Numerical Analysis on Continuum Damage Mechanism of Thermoplastic Composite Pipe (TCP)

by

Aina Syahira binti Mohd Nor Azami

22633

Dissertation submitted in partial fulfilment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical)

JANUARY 2020

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Numerical Analysis on Continuum Damage Mechanism of Thermoplastic Composite Pipe (TCP)

by

Aina Syahira binti Mohd Nor Azami

22633

A project dissertation submitted to the

Mechanical Engineering Programme

Universiti Teknologi PETRONAS

in partial fulfilment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL)

Approved by,

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

(Assoc. Prof. Dr. Abdul Rahim bin Othman)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

January 2020

CERTIFICATION OF ORIGINALITY

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 undertaken or done by unspecified sources or persons.

____

AINA SYAHIRA BINTI MOHD NOR AZAMI

ABSTRACT

The purpose of this paper is to produce a material study on Thermoplastic Composite Pipe (TCP) which consists of PE80, Glass reinforced polymer and PE100 materials. The proposed model used stress-distribution analysis and thermal-stress distribution analysis for onshore pipeline environment under internal pressure load by using Finite Element Analysis in Abaqus CAE software. The composite model has been tested under internal pressure value of 1.72 MPa and 42 MPa and variation temperature dependent data. The stress-strain diagram of the TCP composite has been obtained to characterize the mechanical properties of the composite under internal pressure load. Mesh sensitivity and element type sensitivity study has been applied on the model to verify the model accuracy of the structure. The effect of high and low temperature applied on the model has been investigated. This paper provides knowledge to achieve further continuity study on TCP composite to produce comprehensive analysis on overall characterization of the composite material.

ACKNOWLEDGEMENT

I would like to express my highest gratitude to my supervisor, Assoc. Prof. Dr. Abdul Rahim bin Othman for his guidance and support throughout the completion of my Final Year Project. He has given his most commitment and assurance during my supervision time despite his position as an important person in UTP. I would not have reached to this far if not for his advice and guidance.

I would like to thank Mr. Amirul Ariff for his guidance and reference throughout working on this project. He has shown me persistent and determination in finding root and cause if one analysis is failed during the analysis simulation.

I would also like to thank UTP for given me the opportunity of completion this dissertation paper as a mark for my final year of study in this university. Thank you to FYP Coordinators and Mechanical Engineering Department for giving me such opportunity to contribute a small effort in material study.

To my parents and family, thank you for your prayer and believe in me to complete my 4 years of study in UTP.

To all my fellow friends, thank you for the impromptu theory and analytical discussions we always have as that has solve many difficulties and hiccups during my final year. I truly appreciate and cherish the moment.

Although it is beyond our expectation to finish our study during this pandemic happened in 2020, I surely believe that we got this together and will walk through this tragedy with strong heart and strong mind. Thank you.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1:
INTRODUCTION

1.0 INTRODUCTION

Pipelines serve the important structure in oil and gas industry where it affects the business revenue in collecting crude oils and gas, transportation stocks to refinery plants and distribution products to end-users. Best quality of materials for pipelines must be installed to maintain the efficiency of the product supplied and high full operational capability to operate so less time will be spent on maintenance wise with maximum capacity. Industry has started to approach composite materials as an alternative to replace steel pipelines where it has high beneficiary cost in term of pipelines installation and operational in field. Manufacturing of thermoplastic materials as pipelines will mostly consist of three different layers of materials, depending on the environment introduced. In this study, a thermoplastic composite pipe which contains the layers of PE 100, glass fibre-reinforced polymer (GRP) and PE 80 material will be studied. PE 100 is a third-grade thermoplastic pipe which fulfils mechanical properties demand inside the pipelines where it contacts with the environment of crude oil, gas, water and other microbiological lives from the well supply while PE 80 is a lower grade type of polyethylene which falls in medium-density polyethylene (MDPE) categories. GRP is a type-E glass fibre which is impregnated material with polymer mixture to form a higher flexibility and strength composites.

Composite study is essential in identifying the mechanical properties and respond when subjected to mechanical loads. Composite materials are an adaptation of combination from different material in term of composition, structure and others. These combinations of structures are important to achieve certain resistance towards particular condition and improve the lifetime of the material while in operation. Composite material structure can be a solution to the limitation of one individual's material properties which cannot meet the requirement of specific condition. Fibre reinforced polymer are one of the common materials that is subjected to composite structure as it can contribute to high strength and stiffness of the composite on weight basis. In composite structure, one material does not necessary to dissolve and mix completely with other material in microstructure scale. Individual material can still be identified on its own physical attribute to as a combined material in composite [24].

In this analysis, thermoplastic composite pipe is one of the composites that is made from combination of three different material that can still be identified on physical structure. It has different behaviour when subjected to different composition and material reinforced used where it cannot depend on generalized properties from its discipline characterization. Damage mechanisms in continuum study will be focused in this project to examine the behaviour and natural phenomenon of the composite in a static analysis. Analysis and prediction of continuum damage mechanism on the material will be concluded by using finite element analysis on Abaqus CAE software.

1.1 Problem Statement

This study is motivated by studies available found on the respective aspects: -

Lack of finding in stress distribution analysis on TCP in continuum damage mechanisms. As for one composite material, different modification and alteration of material in term of structure sized, percentage of element, orientation of material direction and others will provide different characterization properties although the material applied is a common type of material used in the industry. None of characterization of mechanical properties has been found for a specific thermoplastic composite pipe which consists of PE80, Glass reinforced polymer and PE100 material combined.

Another problem statement for this study is deficiency founding in characterization of material properties on TCP in continuum damage mechanisms. Material properties data of one element is important especially for design-resistance process where the material is needed to know the maximum allowable value applied on the material before rupture. Lacking of stress distribution analysis on TCP composite model also produced inadequate data for mechanical properties of the composite.

1.2 Objectives

This study is performed to achieve on following objectives: -

To determine stress analysis and thermal stress on TCP under CDM failure mode by using finite element analysis.

Finite element analysis by using Abaqus Software is capable to analyse and process the mechanical response and properties of a material due to advanced software development for material study. Stress distribution and thermal-stress distribution analysis will be determined for the specific material composite of TCP.

Another objective is to characterize mechanical properties on analysis done on TCP under continuum damage mechanism failure mode. Once the stress distribution analysis has done, appropriate value of exerted stress will be generated by the software in order to estimate the stress value with strain rate applied on a mechanical load for the TCP composite model.

1.3 Scopes of Study

This research is limited to: -

- Material of Thermoplastic Composite Pipe (Layers of PE 80, Glass reinforced polymer and PE 100).
- Failure mode on continuum damage mechanisms.
- Conditioned on onshore/buried pipeline for environmental surrounding.
- The pipe is subjected to the flow of fluid in a full stream pipeline.

CHAPTER 2
LITERATURE REVIEW

2.0 LITERATURE REVIEW

Multiple reference and sources have been referred throughout the research on mentioned project to ensure the affirmative founding on the topics are relevant and up to state-of-the-art study. Literature review helps in providing information and knowledge-based approach supported by the proven theory of any experimental data gained. Some of the references will support the idea while contradict studies will help in producing better analysis.

Advantages of composite material has given an improvement in pipeline production in oil and gas industry where mechanical properties of composite can be modified based on the material percentage reinforced in the matrix of one's material. Benefits of reinforced composite material can be applied in producing better mechanical properties of pipelines in transporting different types of fluids. The idea of replacing the usage of steel pipe in oil and gas industry to non-metallic pipe has been debating where non-metallic pipe has high resistivity towards highly corrosive liquids [1] and higher strength-to-weight ratio than steel [2]. However, drawback of replacing steel pipe to non-metallic pipe is it has lower pressure retain and micro crack developing during spooling process of transporting the pipes [2]. Sahin, Akdemir, Avci, and Gemi [3] also suggested that surface crack can be initiated due to fatigue degradation when composite pipe is in stress and corrosive environment.

Various failure mechanisms can be studied to produce better analysis on composite material behaviour under certain limitations.

2.1 Thermoplastic Composite Pipe:

2.1.1 PE 100

Thermoplastic Composite Pipe (TCP) is one of the innovations in composite pipe where reinforced material is substituted inside the core matrix of polymer plastic to improve its strength and corrosion resistance while improving in strength-to-weight ratio value compared to steel pipe. As a common practical for non-metallic pipelines to be spool-abled during installation in field, TCP has been made to compensate high bending stress where its bending strains can achieve from 1% to 3.5% percent. Inspection on the pipelines can be improved throughout the time where no pigging inspection is needed onto the surface of TCP. Inspection method onto TCP is replaced with coupon measurement or wall-thickness comparison will be made from the datum to inspect rate of degradation happened [6].

PE 100 is widely used as inner layer inside thermoplastic composite pipeline. Multiple layers of composite pipelines are used to produce high resistivity material so it will be inert to the reaction for the flowing fluids inside the pipe. PE 100 material is one type of High-Density Polyethylene that has been widely used for its high mechanical properties in term of strength, inert temperature and pressure. It has better crack-growth resistance compared to PE 80 which is suitable for longer service time [4]. Experimental test gathered from [4] has shown that PE 100 has higher strain hardening which will improve resistance of the material to degradation. Amabipi et. al [5] has proven PE 100 is capable to provide more than or equal to 5-year time of service with improved internal corrosion caused by leaks in a metallic pipe before.

2.1.2 PE 80

PE 80 is a medium density polyethylene that is graded as PE 80 as it can sustain to 8 MPa of minimum requirement length. It is common to compare both graded polyethylene, PE 80 and PE 100 since they are the most utilized polymer type in pipeline transportation. The comparisons made for both PE graded pipe are mostly in term of mechanical properties, influence of pre-strain on relaxation behaviour and modulus values. In 1980, PE 80 was one of the first polymer material pipeline that was installed in the field. It is because PE 80 has passed certain requirements and standards as a gas pipeline transportation with sustainable hoop stress on reference standard [ISO STANDARD]. PE 100 has different percentage in matrix ratio as it can sustain up to 10MPa of minimum requirement strength. Industry is prominent to choose PE 100 compares to PE 80 as PE 100 has credit values on crack growth resistance, higher yield strength and elastic modulus. PE 100 is also predicted to have longer service time compares to PE 80. However, PE 80 has advantages on fracture toughness more than PE 100 under compact tension and single edge notched bending testing. PE 80 is also high tolerable and less damaged in squeeze of process in contrast with PE 100. [4]. Squeeze process is a safety feature to shut down a running polyethylene-type pipeline when emergency in risky condition that required shut-off the lines. The polyethylene-pipeline will be flattened into parallel bar to obstruct the flow of the pipeline [27].

In one research [4], polyethylene, especially graded polyethylene PE 80 and PE 100 has comprehensive record as oil and gas pipelines. Their features which are chemically and thermally stable has high point in overcome the absence characteristic of conventional pipes which are chemical corrosion inlet. Furthermore, polyethylene can be modified into advanced polymer that will contains fluorine that will generate special properties to polymer structure such as Ethylene tetrafluoroethylene (ETFE) and polyvinyl difluoride (PVDF).

2.1.3 Glass Reinforced polymer (GRP)

GRP exhibits high corrosion resistance to chemical harm, high in strength and flexibility. GRP is also a common material chosen in material selection for pipeline industry where it adapts to deficiency properties of conventional pipelines such as carbon steel in term of corrosion resistance [18]. They added that polymer with reinforced glass present excellence performance as insulator and protecting riser from etching corrosion. However, GRP has lacked studies found on the damage mechanism and methods in analysing the degradation happened. The characteristics and properties of mixture element in GRP in content of polymer, resin and glass fibre variables held many failures causes from different mechanical characteristics of said materials. Formation of bubbles in between the polyester layer and surface film also stated as one of many failures occurs on GRP. Exothermic reaction during pipeline operation will result in crack formation on the pipeline in between the layers of GRP [13].

Studies on effect of temperature on GRP has concluded that operation temperature is one of the effects on deterioration causes that contributed to damage mechanism for GRP. In the research, [13] stated that moisture, bubbles and heat reaction on GRP is mainly caused from operation temperature that may up to 120°C. Long term and short-term effect on failure mechanisms has been discussed where the factor of temperature rising will influence in changes for ageing material. Long term effect of temperature variable will result in creep phenomenon and fatigue rupture.

2.2 Failure Mode in TCP

Failure mode analysis is one of assessment in conducting and verifying the quality and lifetime value of one's material. The information on failure mode will provide rank of critical issues happens on a material and improvements made to decrease the failure potential [7]. Failure mode is used to identify the type of failure happened on one material to cause it starting to fail. Failure mode analysis is important to study the characteristic and behaviour of the material, and actions taken to reduce prevention of the failure mode to happen. Chaboche [10] added that failure mode analysis is significant in predicting the lifetime structure. Design analysis is also depending on failure mode analysis to produce one assembly of products.

Continuum damage mechanism (CDM) is one of failure mode happens on thermoplastic composite pipe. CDM can conclude the accumulation of failure mode happens on TCP where the structures of composite laminates are based on layers of material combining together. CDM is capable to reconstitute the damage material with homogenous material failure mode by connecting damage mechanisms on the mechanical properties of the materials and its effect [8]. Another theory of CDM is the strain energy density found in in inelastic theory is used to measure the damage variable. Creep and fatigue behaviour are studied in condition of cyclic creep and static which resulting in ductility exhaustion process [9]. CDM is an interaction creep and fatigue failure mode to identify the corresponding the macroscopic crack initiation caused by fatigue [10].

2.3 Simulation FEA on ABAQUS CAE

Finite element analysis (FEA) is one of the prominent ways in investigating the reaction and behaviour of a material when conditioned to a certain aspect. FEA is also capable in calculating and predicting failure mode analysis on one material, providing the condition and environment exposed to the material. Simulation on the reaction of the material is also possible in providing enhance evaluation of visual interpretation. Almeida [11] utilized finite element analysis in her study to identify different failure modes on dry-filament tubes subjected to pressure which will be evaluated on computational analysis. In [11] acquired test, FEA has proved that tubes with lower than 20:1 d/t ratio will fail from buckling and failure in plane-shear plane if the d/t ratio was too high. The amount of data gained from FEA has provide with precise output om the analysis ran.

For running a study on TCP behaviour on CDM properties, a FEA software Abaqus/CAE (Complete Abaqus Environment) was used as it provides sufficient data needed to study failure modes of CDM. Abaqus/CAE is a software tool to simulate and study the condition of one material under certain circumstances providing mechanical components and environment selections available in the software package. To identify the mechanic properties and mechanical response of fibre glass material, Nurhaniza et. al [12] was working with Abaqus software where it is equipped with sufficient modelling of the structure which are macro and microscopic, mixed, discrete and others. Data result such as strain-stress diagram was collected and simulations on strain dispersion was shown in the result section. FEA analysis can be compared to experimental result acquire from physical experiment conducted to improve error analysis. One of drawbacks in FEA analysis is, the simulation on the specimen is expected to be prefect zero deformity. Other than that, with its precise value of data gained from the analysis, some insignificant errors are counted into the analysis such as volume of air trapped, insignificant heat loss which can be negligible with sensitivity control in the software.

CHAPTER 3:

METHODOLOGY

3.0 METHODOLOGY

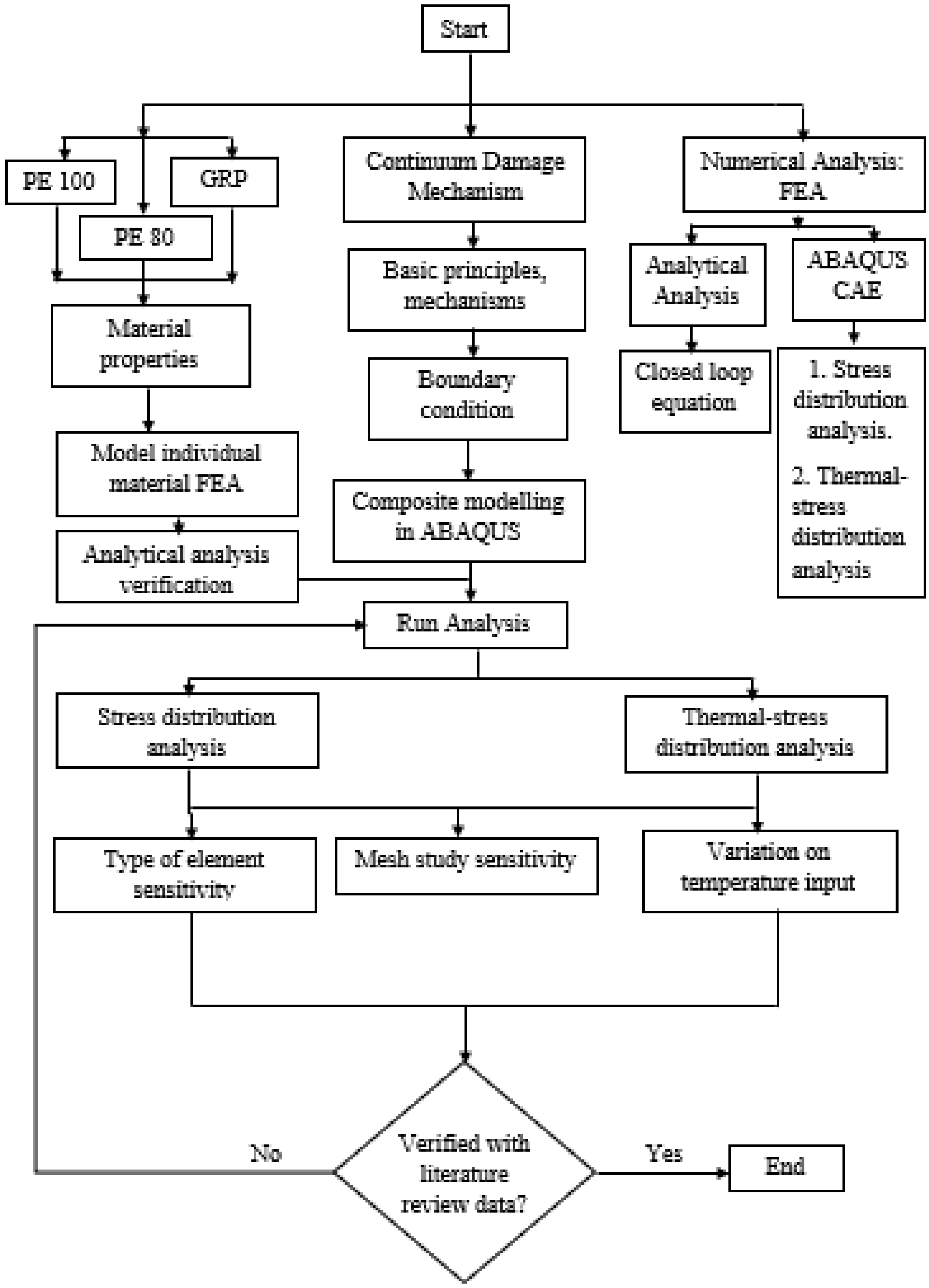


Figure 1: Flowchart diagram for methodology

3.1 Thermoplastic Composite Polymer Scope Study

Those three different materials, which are PE 100, GRP and PE 80 will be analysed individually on their material properties and behaviour study when subjected to bending load. These materials exhibit different characteristics as they are matured differently in a mixture of respective matrix and resin configuration. Study and research done on these materials are guided by literature reviews and finding on discussions cited in this report. It is important to understand different composition of materials exhibit different materials properties. Understanding on the material failure mechanism, focusing on continuum damage mechanism type, is important to understand how one material deteriorates. The material properties needed upon these materials are stress-strain value, Poison’s ratio, elastic modulus and others.

Modelling the materials individually will be one of the methods in analysing the material structure and reaction when subjected to certain condition. In finite element analysis, the material is modelled to obtain mechanical properties value. Acceptable model with correct modelling structure will be guided by comparing the mechanical properties value obtained with literature review findings on the value gained from their analysis. Borchak and Aid (2016) has published values for stress-strain on a fatigue curve subjected to constant stress on PE 100. Same modelling condition can be imitated to prove the quality of modelling designing in finite element analysis.

3.2 Continuum Damage Mechanism (CDM) study

The concept and mechanism of CDM is studied to understand the failure happened and chronological events that will lead to rupture. Diverse brittle material has undergone evolutions which many failure modes are designated to perform the failure mechanics happened on materials. CDM is on type of failure formed in macroscopic crack formed at damaged zone or fracture process zone [15]. Boundary condition and scope of work have been described in Chapter 1.5 to limit the findings into more comprehensive perimeters. After validation of the individual modelling to verify the accuracy of the analysis, modelling on composite material merging those materials (PE 100, GRP and PE 80) into three layers, laminated together to study the behaviour of it when subjected to certain conditions. Mechanical properties of the composite will be attained to be validated with literature reviews related to composite behaviour diagram.

3.3 Finite Element Analysis familiarization and study

Modelling on material structural will be done on a finite element analysis software, ABAQUS CAE which is reliable and has been established to perform computational fluid dynamics. Interface on ABAQUS CAE is introduced for the student's familiarization and tutorial on the exercises has been gone thoroughly to understand the function and operation keys. The objective of utilizing ABAQUS CAE software is to generate thermal stress distribution analysis when the model is synthesized by using identified mechanical properties.

3.4 Structure Modelling

For individual structure, the mechanical properties and dimension of the pipelines is as according to Table below: -

Table 1: Mechanical properties of individual material

Material	Inner diameter (mm)	Outer diameter (mm)	Young's Modulus (MPa)	Poisson's Ratio	Thermal Expansion (K^{-1})
PE 80	187	205	953	0.42	1.8×10^{-4}
GRP	174	187	72.4	0.21	9.81×10^{-4}
PE 100	152	174	1178	0.42	1.44×10^{-4}

GRP material is considered anisotropic where the mechanical properties of the material is direction-dependence where the value of the property is different in every direction.

Table 2: Mechanical Properties for GRP material

Material	Longitudinal modulus, E1 (GPa)	Transverse in-plane modulus, E2 (GPa)	Transverse out-plane modulus, E3 (GPa)
GRP	41	10.4	10.4

Table 3: Mechanical Properties for GRP material

Material	In-plane shear modulus, G12 (GPa)	Out of plane shear modulus, G23, (GPa)	Out of plane shear modulus, G13 (GPa)
GRP	4.3	3.5	4.3

Table 4: Mechanical Properties for GRP material

Material	Major in-plane Poisson's Ratio, v12	Out-of-plane Poisson's ratio, v23	Out-of-plane Poisson's ratio, v13s
GRP	0.28	0.5	0.28

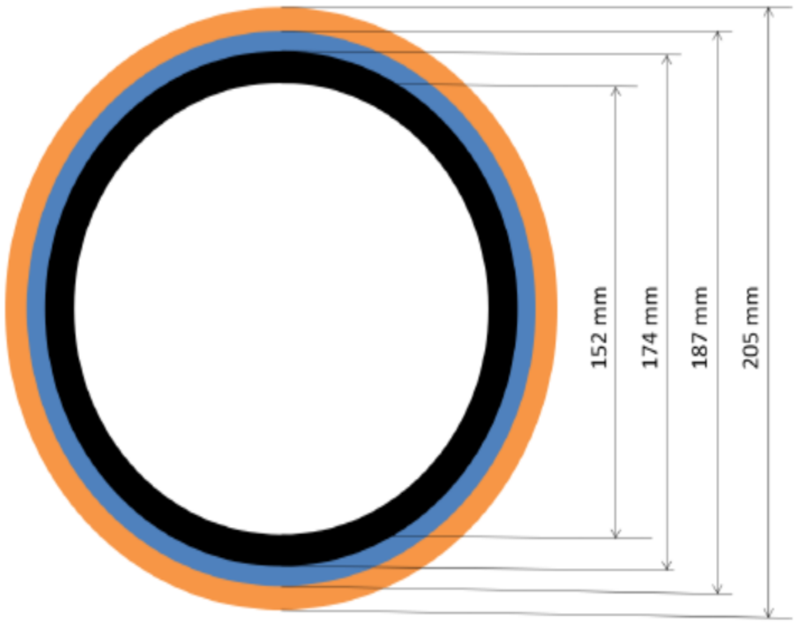


Figure 2: Dimension of pipeline model

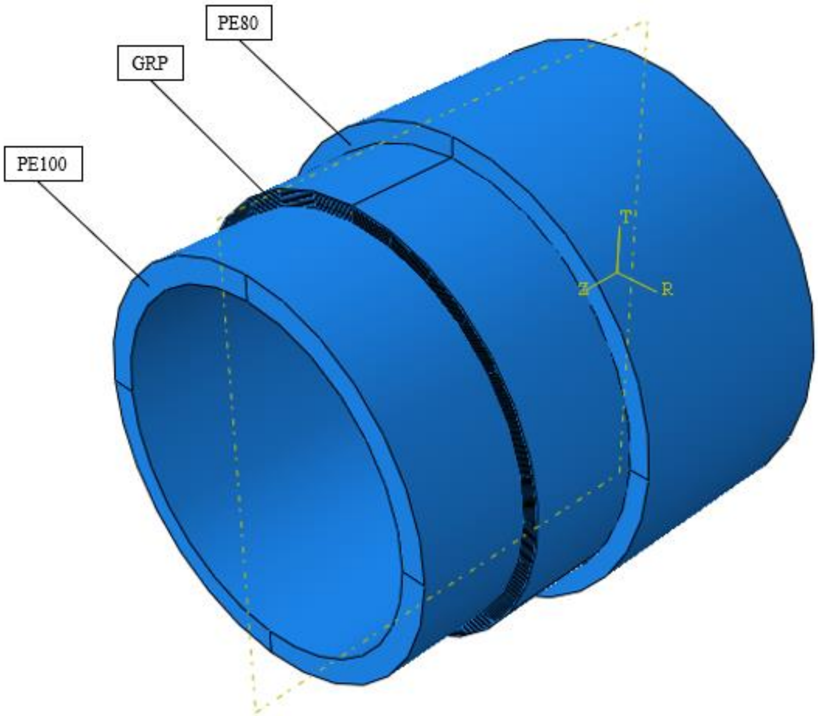


Fig. 4: Arrangement of materials as a thermoplastic composite pipeline in modelling

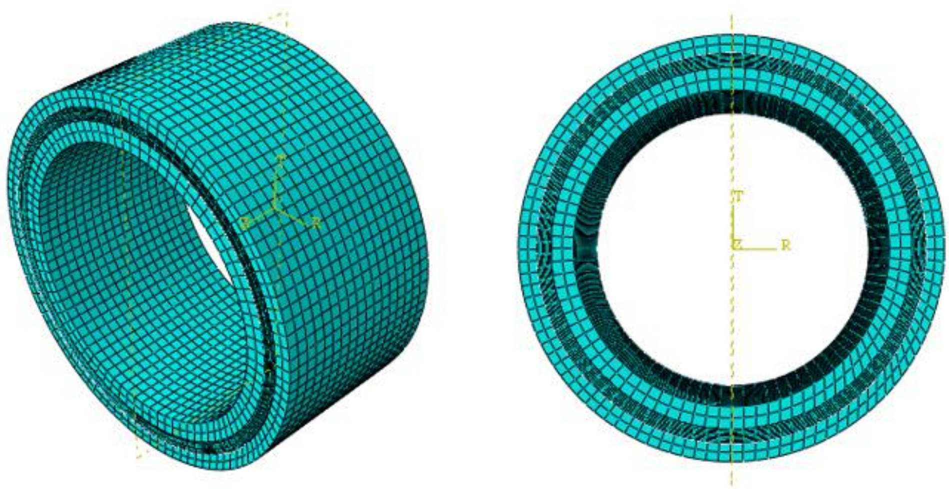


Fig. 3: Mesh picture of the model

3.4.1 Boundary condition of the model

Assumption made on the model is it is a portion of operating pipeline in full stream operation. The boundary condition at the cutting plane of the model is considered to be fixed in displacement, orientation, and in Encanstre condition. The model is analysed in term of force reaction on radial and axial direction inside the inner diameter of the composite model.

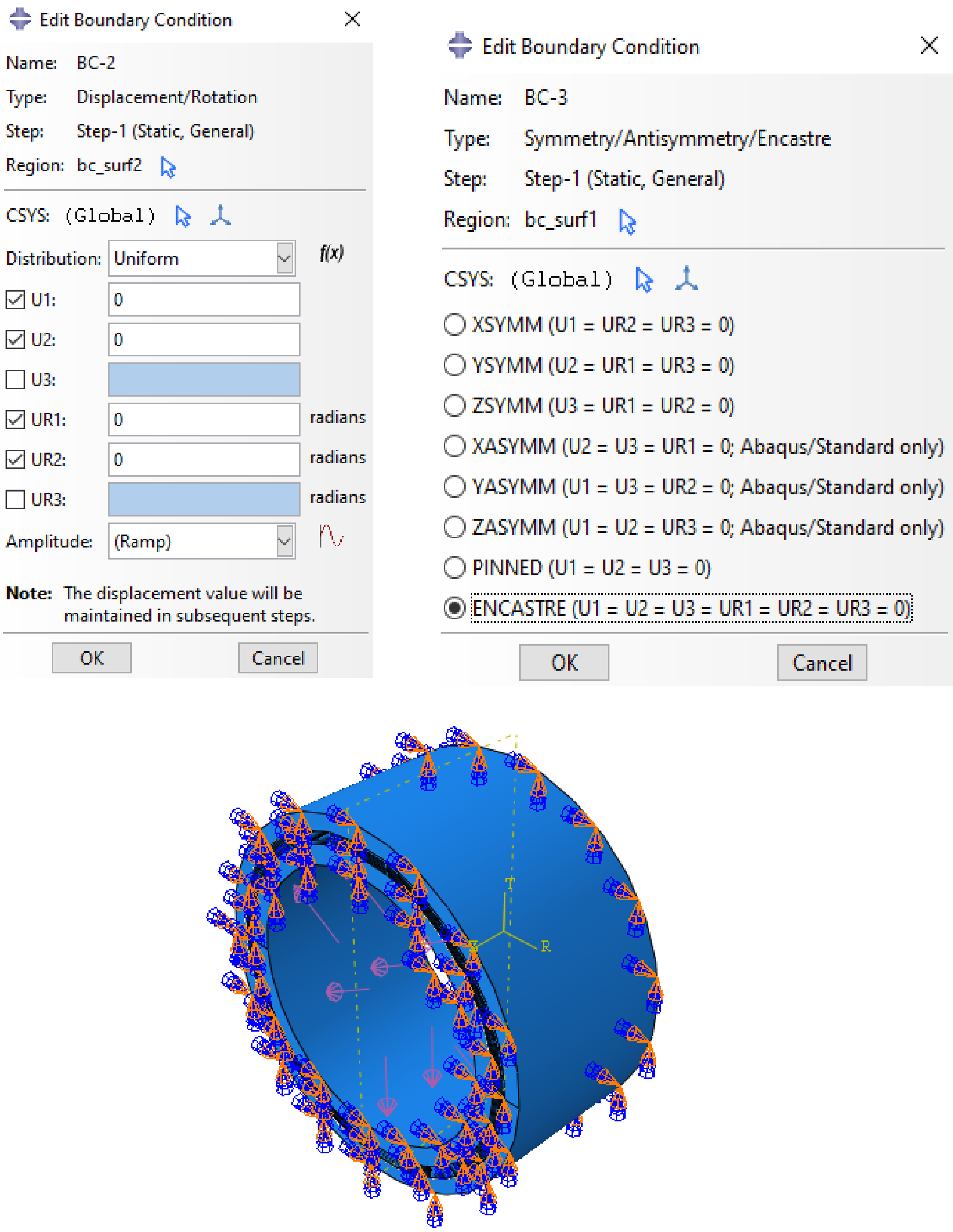


Figure 5: Boundary condition on the model

3.4.2 Internal pressure applied into the model

The internal pressure value applied is and 1.72 MPa based on Table 2 as an operating operation for buried underground pipeline. 4.2MPa which is three times the value of operating internal pressure of buried pipeline in onshore location. However, the rational of choosing 4.2MPa of internal pressure for operating pipeline is to imitate high operational flow of full stream of pipelines in offshore industry.

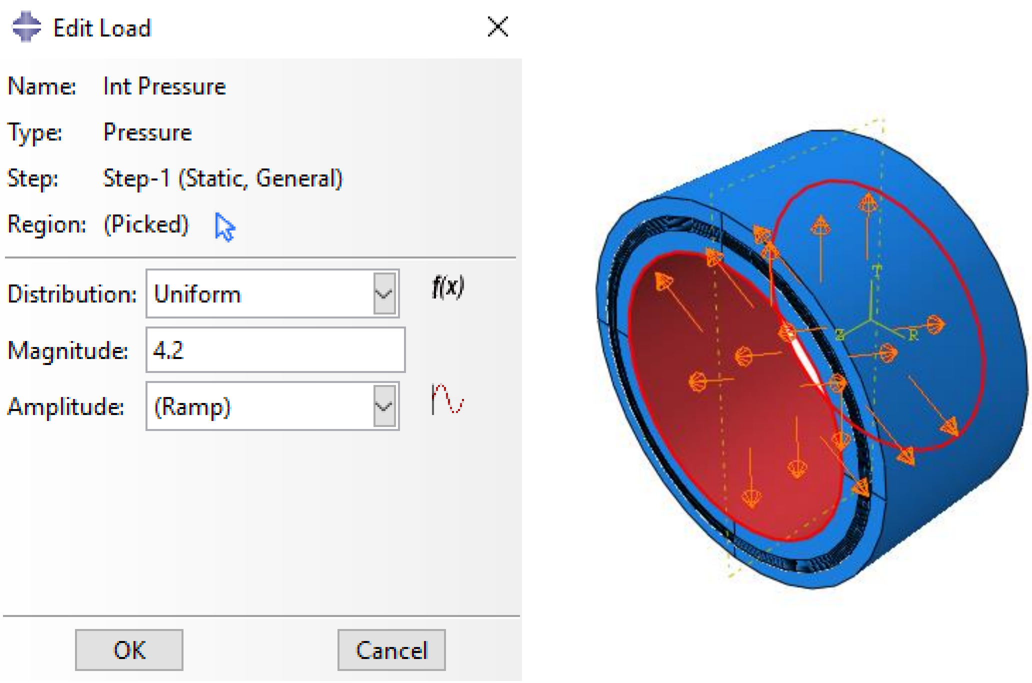


Figure 6: Internal pressure applied on the model

CHAPTER 4

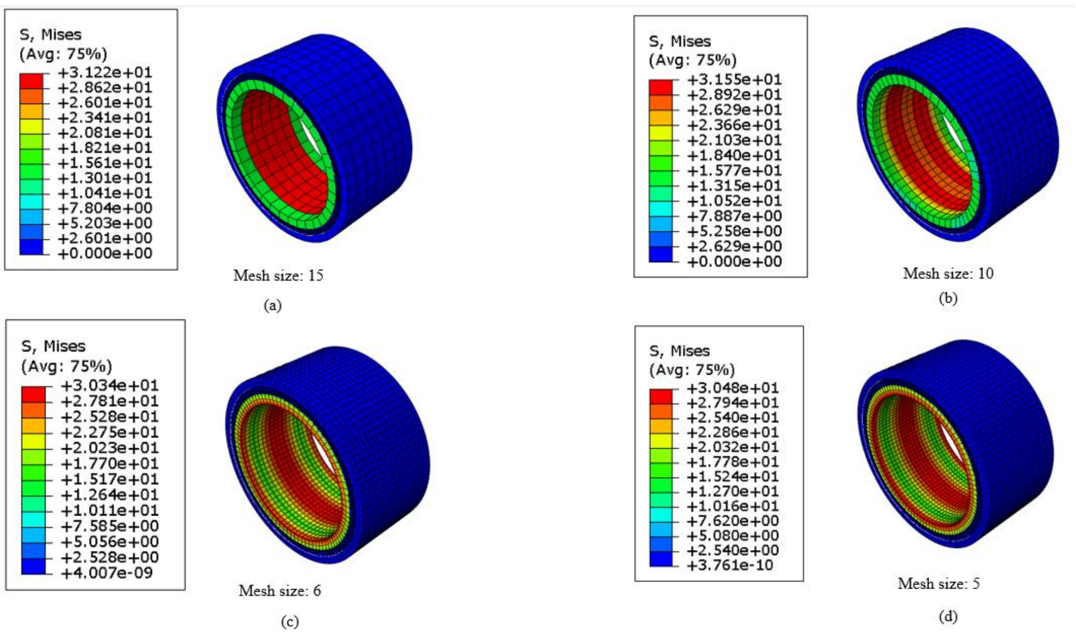
RESULT AND DISCUSSION

4.0 RESULT AND DISCUSSION

4.1 Software Interface Study

4.1.1 Mesh Sensitivity Study

In finite element study, mesh sensitivity will be one of the contributing factors in producing accurate result of model analysis. Mesh sensitivity analysis can be performed by simulating models with different mesh number and element sizes under the condition applied on the model [29]. Number of mesh size can influence the accuracy of the result produced, given it in course or fine mesh. Sensitivity of mesh size can dominate the impact of analysis as finer mesh region is needed to analyse delicate region in the simulation such as plastic zone length. Courser value mesh size which is bigger than the plastic zone size will not precisely analyse the domain hence



leaving the significant analysis on the important part. Sensitivity analysis on TCP model has been ran within the same load applied.

Better distribution on the stress analysis has been observed as we increase the sensitivity of mesh size on the mode. Obvious difference is on the size of stress intensity performed on Fig. 6(c) is more apparent on than Fig. 6(a) and Fig.6 (b) due to finer region of mesh number is adapted.

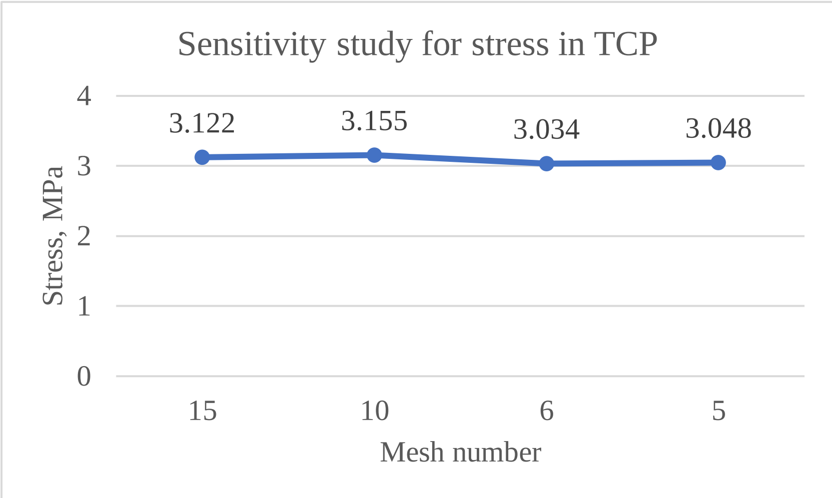


Fig. 9: Mesh sensitivity study graph on TCP model

Table 5: Mesh Sensitivity Analysis Result

Mesh no	von Misses Stress (MPa)	Element number		
		PE80	GRP	PE100
5	3.048	4960	18240	4120
6	3.034	3536	12920	2958
10	3.155	630	4480	530
15	3.122	284	2128	245

Based on Fig. (7), result obtained is defective by using mesh number of 15. The result is increasing in detailed stress value from 15 to 5. Stress value in mesh number of 5 is observed to be common with stress value in mesh number of 6. Choosing to sustain on simulating the model with mesh number 6 will save more in terms of computational time and calculation speed. Stress value has insignificant different when the mesh number is at 5 and lower. As finer mesh number is applied more than 6 on the model, more computational time and high computer hardware specification will be needed to produce stress values that is different in 0.14 MPa. Speed of calculation will be also affected due to unnecessary mesh sensitivity on the model. Region of mesh is best possible to be partitioned with datum plane axis when in subject to heterogenous material combined as TCP model.

4.1.2 Element type sensitivity

Analysis in sensitivity of type element has been done in the simulation for the all model. Objective of this study is to see the difference made and the result produced when the models are subjected to different element type of sensitivity. In Abaqus CAE, types of elements chosen must be corresponding to the topological body of the model structure. Elements type are subjected to types of three dimensional shapes such as tetrahedra, wedges, hexahedra and others. All element types are capable to process on different loads types such as gravity, surface pressure and forces applied. By default, the software will assign on the mesh region and element type when the model is created.

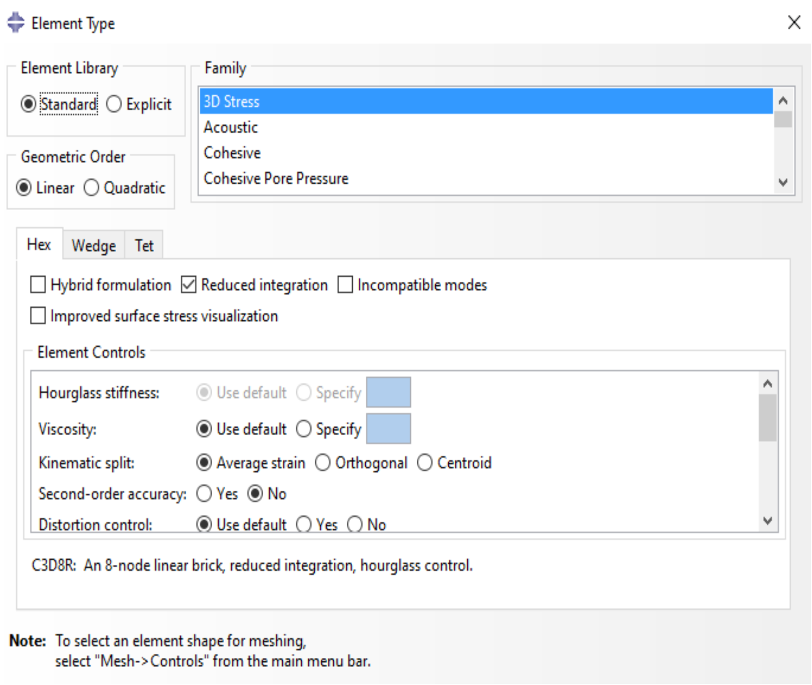


Fig. 10: Mesh element type option for C3D8R

However, there is options on changing to different element type to run on the integrity of the assessment to process the analysis,

To conduct the individual layer of materials and TCP model, common shape of cylinder has been modelled in the platform. An element type has been chosen to run the model in three-dimensional solid element type which is C3D8R type. This type of three-dimensional hexahedral element is working on reduced-integration elements. Fine meshes are needed to run on this type of element to obtain accurate result. This type is described in element type as an 8-node linear brick, reduced integration and hourglass control. Abaqus runs to integrate variety of element overt its volume to allow the simulation on the material behaviour in complete general processing. It quantifies the reaction of the material at each integration point in every element by applying Gaussian quadrature rule. The type of name C3D8R element has meaningful context which C – continuum, 3D – three dimensional, 8 – 8-node brick element and R – reduced-integration element. Reduced-integration linear element depends on the thickness for the material to response accurately. It calculates the element strain energy with hourglass control where it forms element stiffness by applying low order integration form. This will help in reducing the time and speed calculation of the model when it is analysed.

As geometrical shape of common structure of beam, pipelines and plate can be assumed to have general section behaviours, numerical integration can be applied on the element type to study the mechanical behaviour of the material when subjected to loads. Fig. (7) resembled the model in this element type of C3D8R for the solid type of element analysis. Once the model is run in this element type, the result of said element is shown in Fig. (8).

Another type of element tested in this model is a shell element type. Shell element is limited to analyse large-strain analysis where this option can be select in thick or shell problems. Shell element can be applied for three-dimensional and axisymmetric model analysis which TCP model has of these features. Abaqus shell element is implying linear interpolation in its analysis and response to mechanical loading when the conditioned is applied.

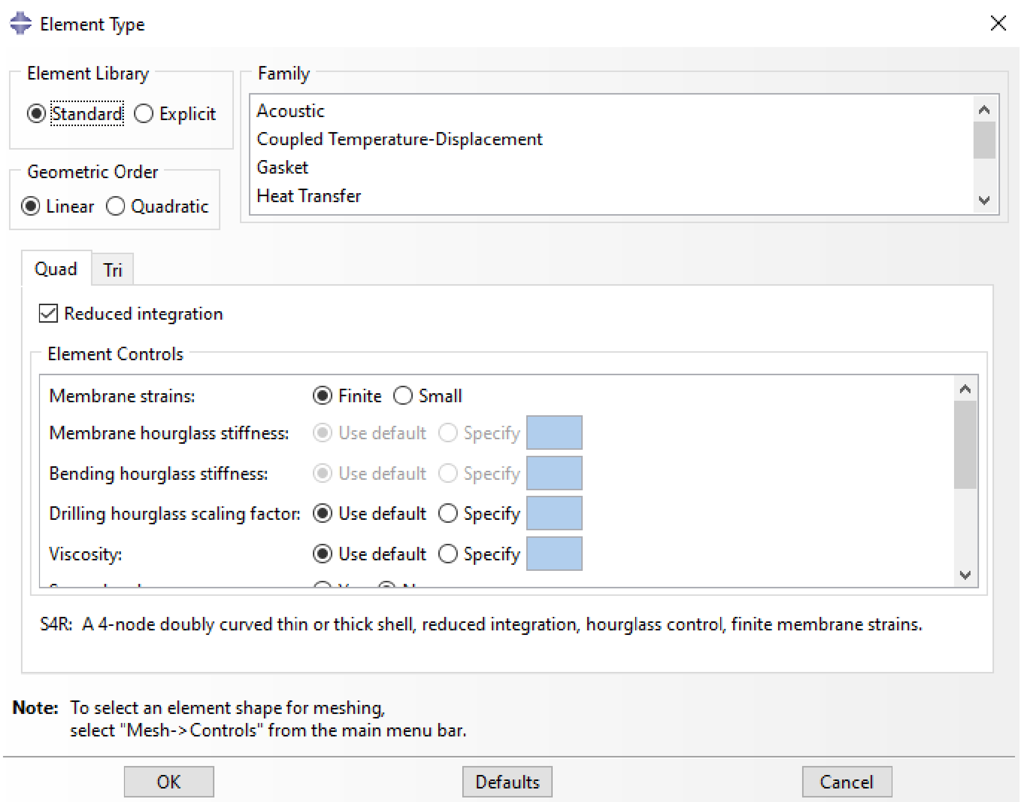


Fig. 11: Mesh element type option for shell element

Element type chosen in this element is namely S4R which indicates shell element type, 4 nodes and reduced-integration processor. This element is restricted to application that applied transverse shear deformation. Accurate result will be processed if this element type is applied on transverse shear deformation and thick shell problem model.

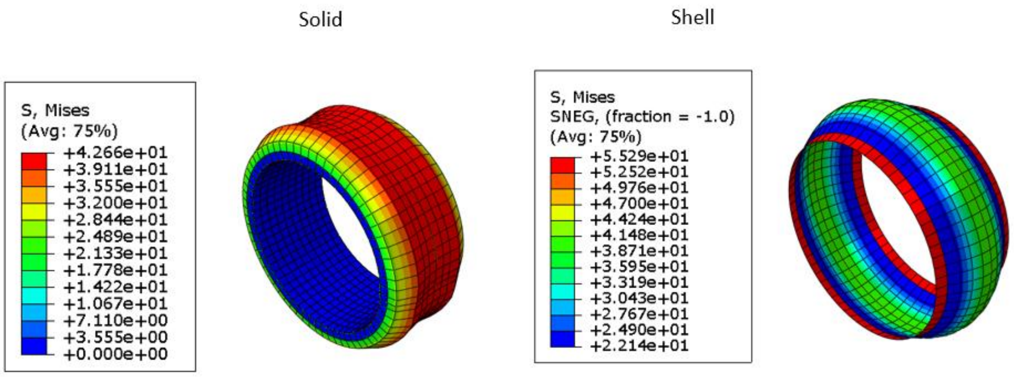


Fig. 12: Comparison analysis between C3D4R and S4R type element in PE80 model

Shell model analysis is seen to be no thickness as the thickness is specified in a specified table in Properties tab. The thickness of the shell is obtained from the difference of size between outer diameter and inner diameter of the pipelines. Large difference is found between the von Misses stresses value between C3D8R type element and S4R. Through thickness stresses that is investigated in this model is preferred to use solid element type compared to shell element type. Thin shell theory is not sufficient enough to apply on the model is applied on dimension to length (D/t) model analysis thus it is favourable to use 3D solid element type instead [22]. Shell element type is concerned on thinner structures than cylinder where cylinder is considered thick as a tubular structural. So, shell element type is not applicable in the study of stress analysis distributions on pipelines models.

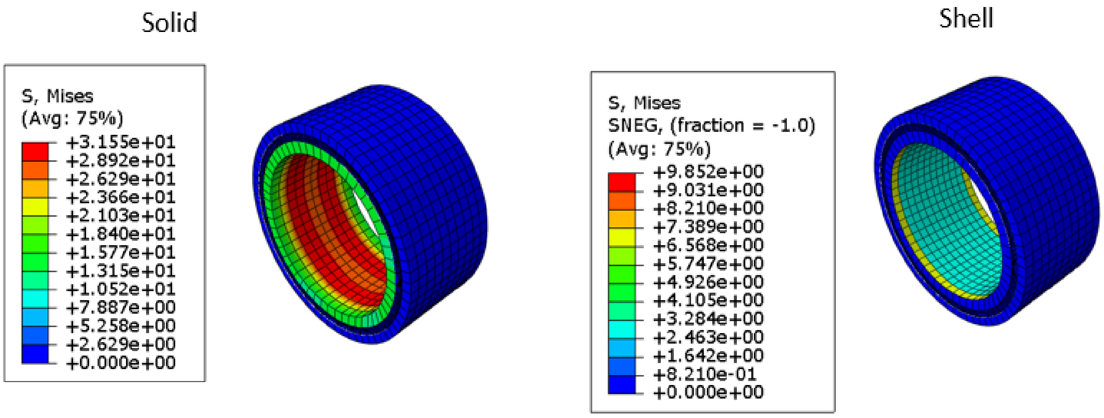


Fig. 13:Comparison analysis between C3D4R and S4R type element in TCP model

Analysis on TCP model using different type of elements is arguable when the interaction on the material to each other (master/slave surface) on shell element is not accurate. The model has specified tab to automatically transform the model parts from solid into shell element and the it is resulted in Fig. (17).

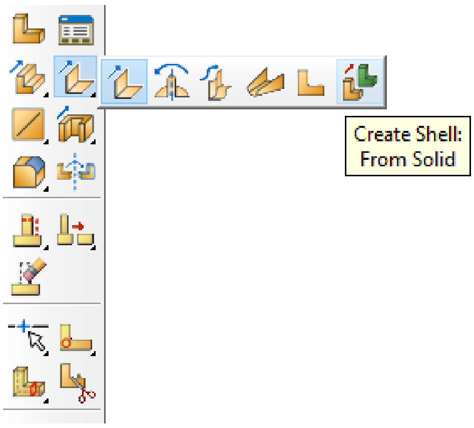


Fig. 14: Command on direct transforming from solid to shell part

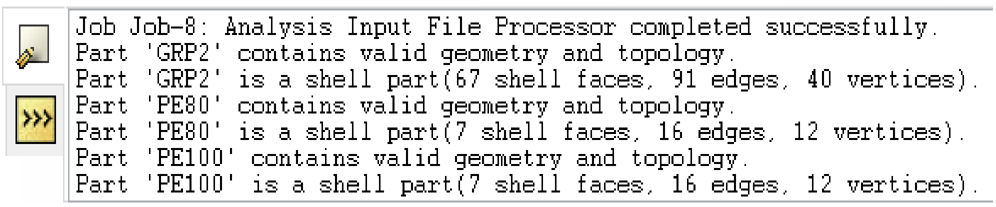

Job Job-8: Analysis Input File Processor completed successfully.
Part 'GRP2' contains valid geometry and topology.
Part 'GRP2' is a shell part(67 shell faces, 91 edges, 40 vertices).
Part 'PE80' contains valid geometry and topology.
Part 'PE80' is a shell part(7 shell faces, 16 edges, 12 vertices).
Part 'PE100' contains valid geometry and topology.
Part 'PE100' is a shell part(7 shell faces, 16 edges, 12 vertices).

Fig. 15: Command message on transformed element model

4.2 Stress Distribution Analysis

4.1.3 Individual Layer Structure

In stress distribution analysis, a common operating condition of pipeline were applied in the analysis model. Operating condition can be extracted from [19] where they explained on monitoring technique on operating oil and gas pipelines in underground.

Table 6: Operating condition. Source: Adapted from [19]

Conditions	Gassy-oil	Oily-gas	Oil-transmission
Gas production rate, m^3/day	17×10^3	122×10^3	This pipe section carries 99.5% oil with less than 0.5% water
Oil production rate, m^3/day	49	28	
Water production rate, m^3/day	170	31	
Gas composition	20% H_2S + 2.5% CO_2, balance hydrocarbon		Less than 0.5% H_2S + 0.5% CO_2
Temperature (°C)	60	55	55
Pressure, psi	250	250	Atmospheric
Replaceable pipe section, in.	6	10	10
Diameter of the operating pipe in the field, in.	6	10	1
Inhibitor, continuous batch	No. 1[a] and No. 3[a] No. 2[b] and No 4[c]	No. 5[d] No. 6[d]	No. 2
Concentration, ppm continuous batch	0, 50, 100, and 200 0 and 2000	0, 100, 250, 500, 1000 0 and 5000	0 and 2000

[a]Water-soluble.
[b]Oil-soluble, water-dispersible.
[c]Oil-soluble.
[d]Compositions not known.

Based on Table. 1, operating internal pressure applied is on 250 psi which equivalent to 1.724 MPa value in the model. Variation between two internal pressures have been made in the model analysis to study the reaction of TCP when it is applied to a high and low operating pressure. Effect on high and low pressure applied will give significant input and value in determine the characterization of the TCP model later on when full composite analysis has been done.

However, internal pressure input gained from [19] is an operating condition for steel pipelines. For thermoplastic material model, condition applied for internal pressure in the model will be 4.5 MPa which is three times higher than the operating internal pressure in underground steel pipelines. This reference is made to have the same operation pressure with operating pipelines in offshore [20]. Other than that, less reaction happened on the model when pressure of 1.7MPa of steel pipelines was applied. Therefore, for individual material study, 4.2 MPa of internal pressure has used throughout the analysis. There is no operating temperature applied in the model as the model ran with a time-independent data gained on the elastic modulus and Poisson's ratio values.

Individual material analysis is important in identifying the characteristic exhibits by the material when the operating conditions were imposed. Different individual model has been structured for every layer of material involved in the composite for its stress distribution analysis study. Different materials exhibited different stress values due to different mechanical properties values. Reaction of the individual model can be observed in term of shape deformation, stress values on different regions on the pipes and distribution of the stress when it is operating condition.

The output of stress distribution on individual materials can be as a reference to thermoplastic composite pipeline model when it was created. The studies of these individual materials can be correlated when the same operation condition is applied on the composite model. Observation on the deformation of the model shape is important to identify the composite deformation. All materials will be structured as a composite and will be layered accordingly to their strength value on exerting forces applied during operation.

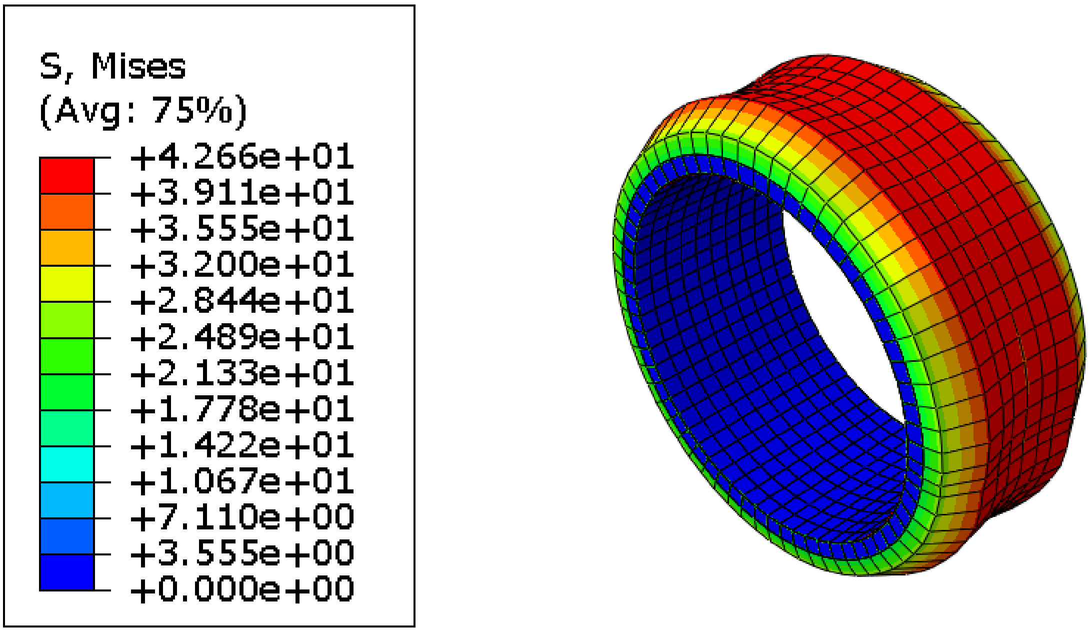


Fig. 16: von Misses stress analysis for PE 80 model

For PE 80, The model has shown that PE 80 has shown significantly impact when subjected to the pressure value. Stress exhibited when the internal pressure of 4.2 MPa is applied into the model is 4.266 MPa. It is can be observed that the model is deformed in radial direction where the model is found to be inflated in radian. The outer surface of the model is found to be in higher stress compared to the inner surface. The value of von Misses stress found in the model is to be lower than the maximum allowable stress applied on the material before pipe failure. The true value of maximum allowable stress with respect to PE 80 material applied is expressed as follows [21]:

$$\sigma_e = \frac{P_i\,[21k^4 + 3.6k^2 + 0.27]^{\frac{1}{2}}}{(k^2 - 1)} \quad (1)$$

In Eq (1), $k = D_o/D_i$ (where D_o is implied as outer diameter and D_i is inner diameter of the pipeline, respectively) and P_i is implied as internal pressure of the pipe.

From (1), the amount of maximum allowable stress derived is 122.94 MPa. The pressure exerted inside the internal region will not cause rupture to the model. PE 80 material is capable to hold the pressure exerted when conditioned to other factors which is in ambient temperature.

Hoop stress is one of significant elements in characterization of the material. Maximum tensional force exerted by the pipeline is depending on the hoop stress value.

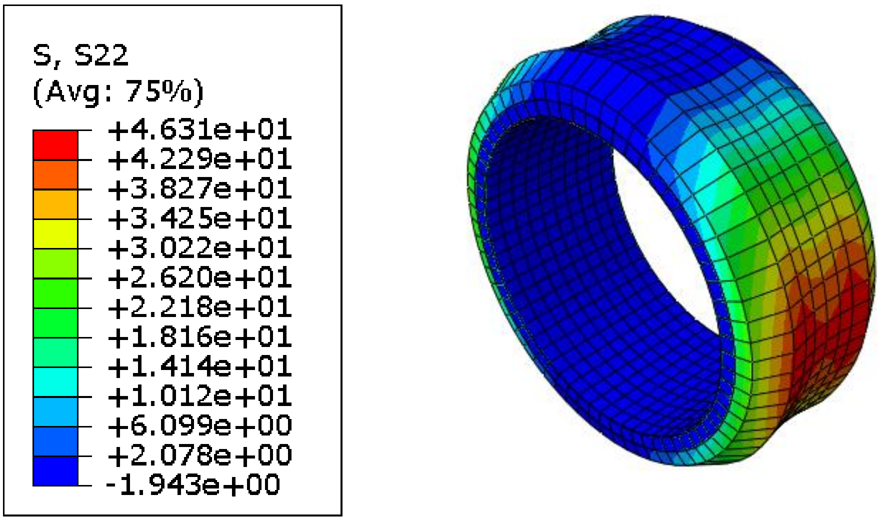


Fig. 17: Hoop stress analysis for PE 80 model

Based on Fig. (2), high stress distribution was observed on the outside of the pipeline model. High stress on the outside was concluded to be the result of high compressive force from ambient pressure compared to internal pressure applied. The force is acting on a radial direction as a result from the internal pressure reaction force.

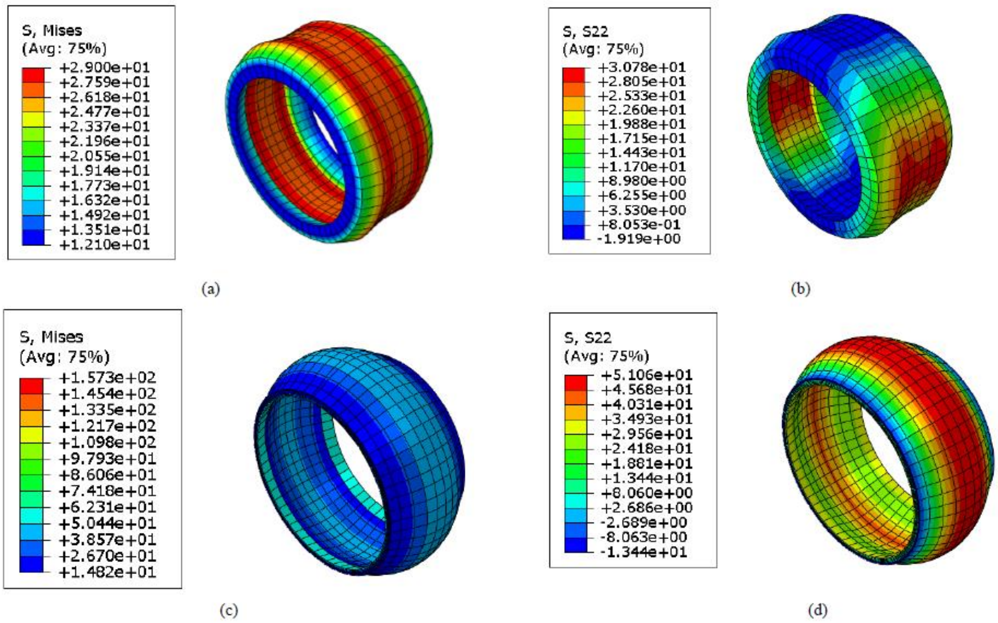
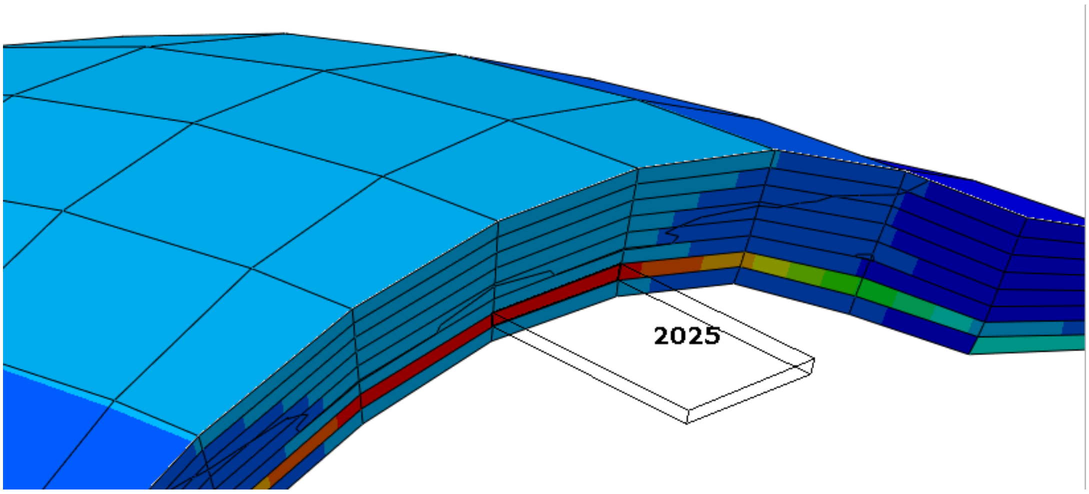


Fig. 18. Result of stress analysis modelling: (a) von Misses stress on PE100 (MPa), (b) hoop stress for PE100 (MPa), (c) von Misses stress for GRP (MPa), and (d) hoop stress for GRP.

Fig. (3) shows observation gained from simulation on material PE 100 and GRP with the same condition applied as previous model. In Fig 3(a), PE100 resulted in lower von Misses stress as compared to von Misses stress value on PE 80 in Fig. (1) due to high elasticity modulus value and strength. PE100 is considered as high-density polyethylene while PE80 is categorized as medium-density polyethylene. However, the internal pressure value of 4.2 MPa applied on PE100 model has resulted in high hoop stress intensity distribution from the inner to the outside region. High hoop stress distribution inside the model has resulted in more compressive deformation observed on PE 100 model in Fig 3(b).

For GRP material, the highest stress presented in the model to be 155.83 MPa in on the second layer of fibre orientation of +90^0 angle can be observed in Fig. (5). Perpendicular fibre orientation to the applied load will result in lesser toughening strength compared to fibres that are not perpendicular. The respective fibre that is perpendicular to the load will contribute in failure of the model such as breakage and



unbinding with other layers. Parallel orientation however will have higher strength, toughness and can carry better load capacity more than other orientation.

Fig. 19: Partition of GRP model into half, element number indication of high stress value on fibre orientation

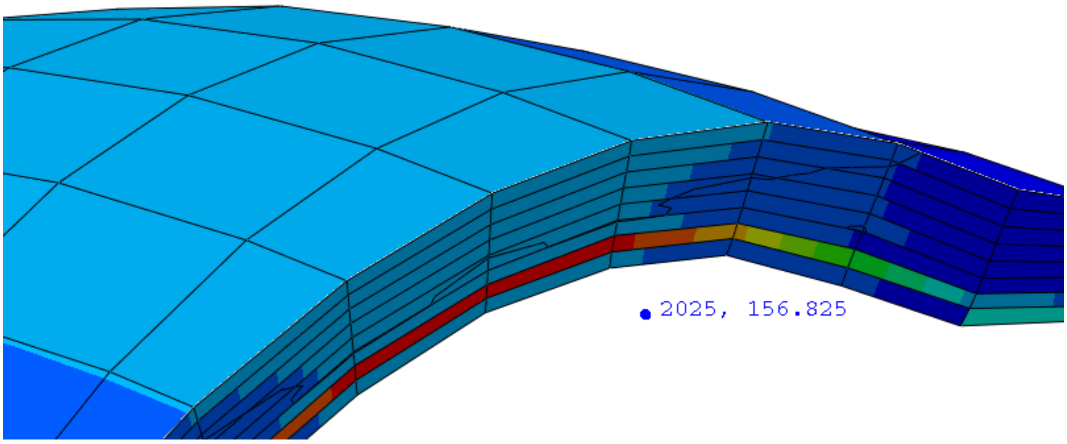


Fig. 20: High stress value at 90^0 fibre orientation in GRP model.

4.1.4 Thermoplastic Composite Pipeline

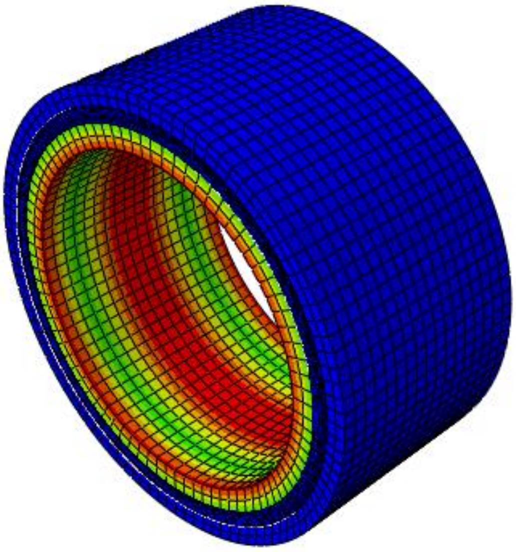


Fig. 21: von Misses stress distribution in TCP model

Stress distribution analysis was done on a structure consist of PE 80, GRP and PE 100 according to the arrangement of the pipe material in Fig. (6). The stress distribution was only managed to exert into layer of PE 100 without entering to GRP layer region. Partition of the model to approve on the statement before can be refer in Fig. (9). High intensity stress distribution was found in the middle of inner region of the pipe.

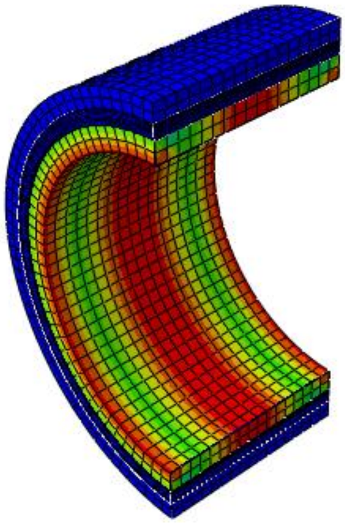


Fig. 22: Partition of TCP model

4.1.4.1 Internal Pressure of 1.72 MPa

For stress distribution analysis study, the figure shown from the model after it is being analysed has the same distribution shown as Fig. 8. The colour intensity of the analysis distribution is also similar, except for the value of the stress generated is different. Generated model and value for stress distribution for 1.72 MPa of internal pressure is shown as below: -

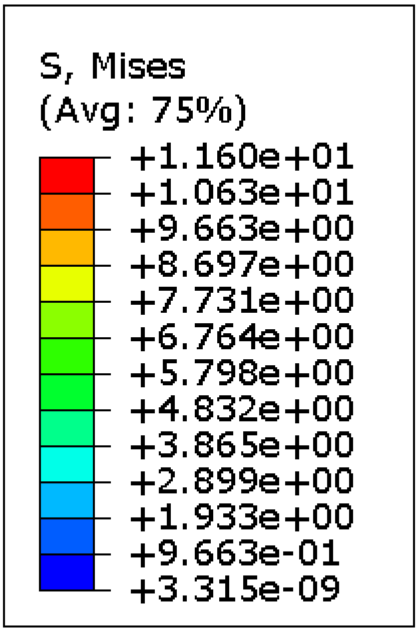


Figure 23: von Misses value presented in the model for 1.72 MPa

The highest value developed by the model in the respective internal pressure value is given as 11.6 MPa. The stress-strain value extracted from the software is presented as Fig. 24: -

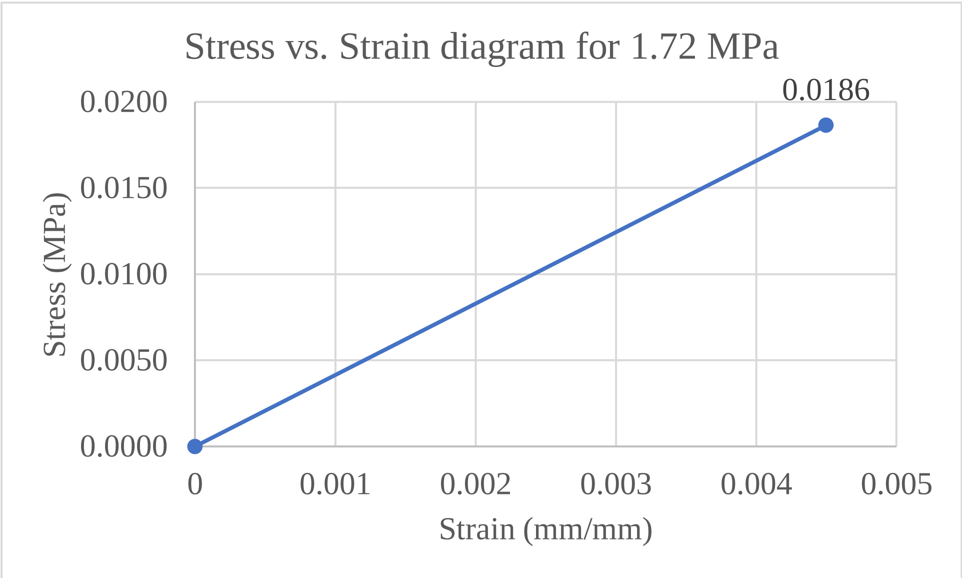


Figure 24: Stress strain diagram for 1.72 MPa internal pressure

Based on Fig. 20, the value of 1.72 MPa of pressure will give a value of stress on the model to 0.0186 MPa. This value is linear to the strain rate of the model as the figure is running. This has concluded that under the internal pressure of 1.72 MPa, the TCP model will be in plastic deformation where it will permanently be deformed after the load is released. Under this load, plastic deformation will happen as before low yield stress region when stress is applied on the model.

As we compared to Fig. 21 where a complete until necking phase of the model has been established, the stress value of 0.0186 MPa from 1.72 MPa of internal pressure as referred to Fig. 20 has been marked to analyse the position of the point coherent with its strain value. The point has not passed the below the low yield point for the composite to initiate plastic hardening process in the model. Plastic deformation of the model is supported by the ductility characteristic by the combined material the TCP. load that can lead it to rupture.

4.1.4.2 Internal pressure of 4.2 MPa

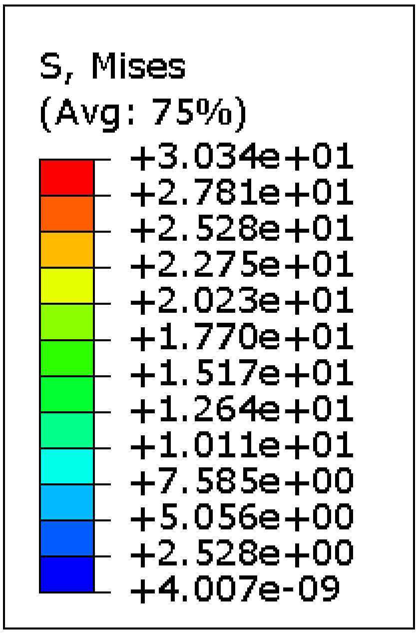


Figure 25: von Misses value presented in the model for 4.2 MPa

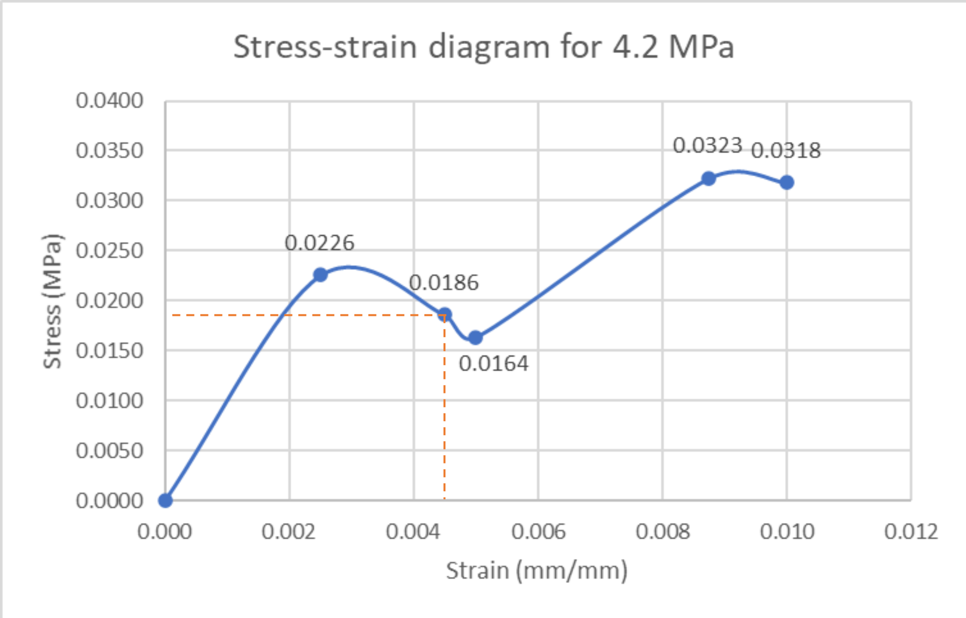


Fig. 26: Stress-strain diagram for 4.2 MPa internal pressure

The TCP model for internal pressure of 4.2 MPa exhibited a plastic toughening behaviour based on the Fig. (26). Thermoplastic composite can be considered to have essential properties of toughness and resilient to fracture which can contribute to lower time for crack propagation to happen. This significant property is giving high value and advantages as an application to pipeline operation that makes them more favourable compared to conventional steel material when subjected to load. Elastic deformation happened at until the stress value of 0.0226 MPa where the material will be permanently plastic when the stress applied is beyond the point. This upper yield stress point is the limit point of stress applied before plastic deformation happened. After this level, material will remain in plastic deformation even when the stress applied is removed. In this level, crystalline structure of the material has changed and started to move that can cause dislocation in microstructure level. Higher dislocation distance between the structure will force them to restrict in further movement. Then, the crystalline will associate with each other to compensate to the longing stress applied on the material. Strain hardening is starting at lower yield strength where the length of crystalline is increasing. Stress value of 0.0164 MPa will be the lowest value of stress required on the model for it to undergoes plastic deformation. Strain hardening development explains on the higher stress applied is still endurable with increasing strain value on which the material is in higher stiffness and strength condition. Lower yield point value is crucial in design process to reduce plastic deformation to happen on the material. Beyond the lower yield point, ductility characteristic will further develop until at ultimate tensile strength, the material will prone to necking and will lead to fracture if it exceeds the true value of allowable maximum applied on the model.

For stress distribution analysis on composite, observation can be made on Fig. (21) where the stress distribution of the model is not penetrating to the GRP layer which contains multiple fibre orientations. It is concluded that glass fibre has contributed to a better performance for the composite compared to polyethylene material [28].

4.3 Thermal-stress Analysis Distribution

In this analysis, thermal-stress analysis distribution on a pipeline flow with temperature and pressure difference has been done to study the characteristic of the material when it is subjected to 4.2 MPa internal pressure. Variable temperature on the difference of 10K and 100K temperature has been applied on the model.

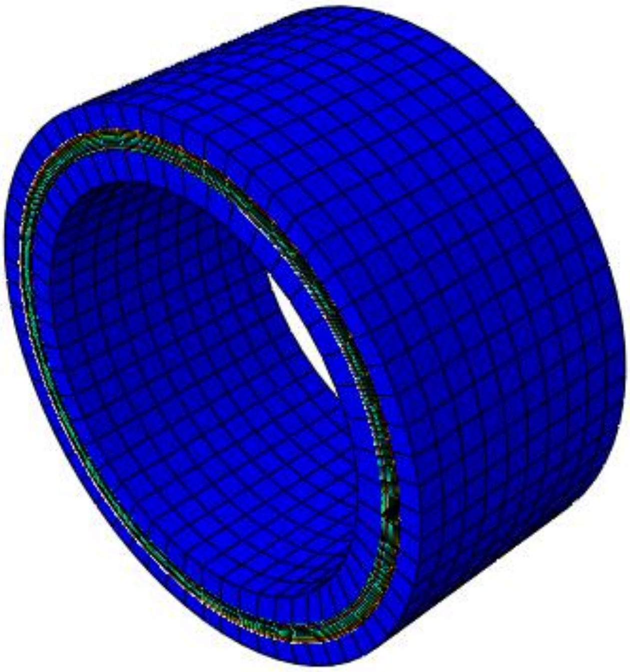


Figure 27: Thermal-stress distribution analysis on TCP model

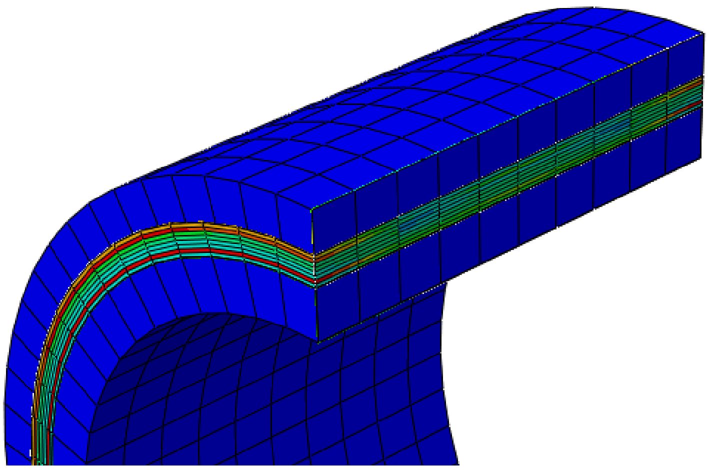


Fig. 28: Partition for thermal-stress analysis model

In Fig. 21, an observation of high stress value was indicated on GRP layer of material at the first and last two layer for the model. In this layer, the material orientation of the GRP is at 90^0 degree. Fibre orientation carries a good factor in determining the mechanical properties of one composite. High stress value at 90^0 orientation is a result as a potential fracture mechanism breakpoint where it cannot bear with the load applied on the composite. The load then will be endured by the breakage of fibres orientation which are parallel to the load. Some of the fibre breakage is fractured at weak cross-sections, which will increase the local stress intensity and increase the load at adjacent fibres e.g. 90^0 fibre. The additional load will be relieved by plastic deformation of the matrix near the broken fibre [23] However, different judgment has been concluded in other experimental data. Based on [24], glass fibre material has a high strength and intensity stress at fibre orientation of 90^0. More extension on the material while applied to certain load at this degree will result in more flexibility of the material. Rigidity of material will be greater on this layer where it can sustain to high yield point of the composite before it goes beyond plastic deformation region. In [25]. they agreed on the high strength and intensity stress as 900 fibre orientation as it acquires more load to fracture the material. As addition to sustain high yield point on 900 fibre orientation, [25] also concluded that Young’s modulus value is more at this orientation which contributes to more elasticity of the material and higher yield point on the material before plastic deformation.

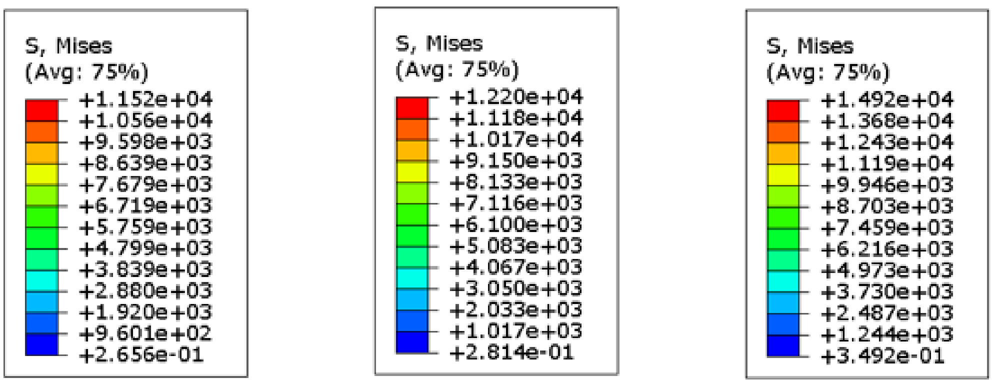


Fig. 29: Comparison of stress value at (a) 338K, (b) 358K, and (c) 438K temperature on composite model

Table 7: Stress Value on the model vs. Temperature

Temperature(K)	Stress value (MPa)
338	11 500
358	12 200
438	14 900

Increasing of temperature has resulted in increasing of von Misses stress on the model. This value is the highest stress value that can apply for respective temperature based on the model before fracture happens. Higher stress value due to elevated temperature can be associated with higher polymerization stress on the model. Polymerization is a process of monomer molecules reaction inside the composite reacted with chemical reaction or condition to produce polymer chains or three-dimensional structures. Consistency of the material during load is decreasing in viscosity system when temperature is elevated which produced monomer chain in the composite as a result for higher polymerization stress. Raised up temperature also gives better molecular flexibility and element diffusion of the material is governed by reaction diffusion, not from one diffusion-controlled than can terminate the mechanism and causing fracture. Additionally, higher temperature can refrain from vitrification process to happen. It is a condition where the glass transition temperature in the composite is moving towards curing temperature. This refrain is a result of higher rate of monomer conversion as polymerization is increasing and bypass the glass transition temperature before reaching the temperature of polymerization reaction. [26].

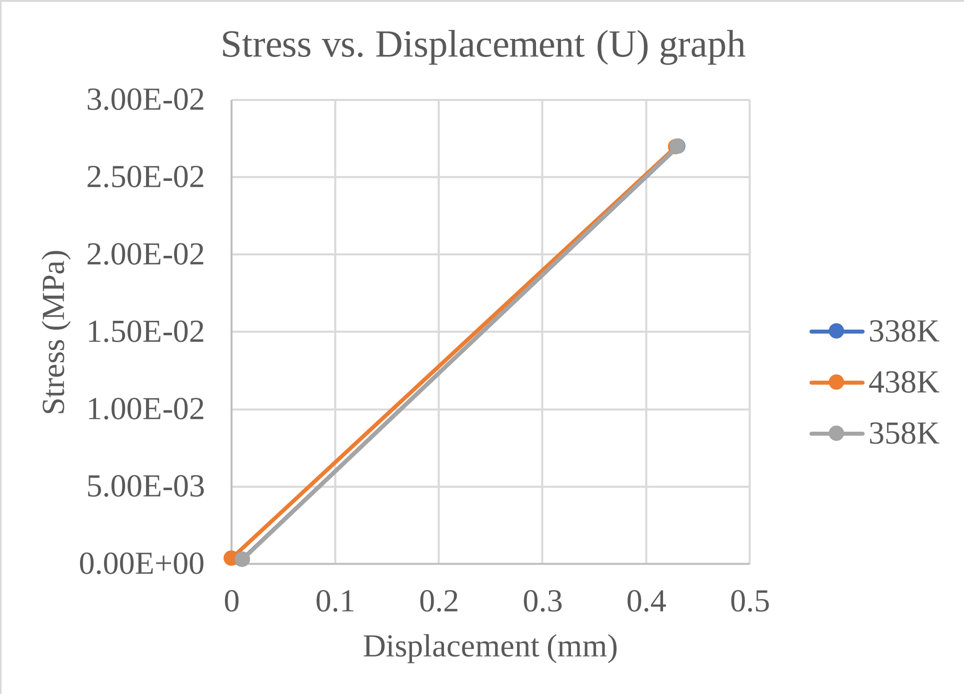


Figure 30: Stress vs. Displacement graph for thermal-stress analysis

In this graph, variation of stress related to displacement position subjected to variation of temperature is represented. A stress-change of radius of the model graph is developed to study the properties of the model in three different temperature. The base temperature is set up as 338K, where elevation of 20 degree of temperature can be compared to 358K and 100 degree difference with 438K. Minimal difference can be observed on 20 degree elevated temperature in the graph.

CHAPTER 5
CONCLUSION & RECOMMENDATION

5.0 CONCLUSION

This line of work has successfully accomplished all the objectives of this project which are to perform finite element study on stress analysis and thermal analysis distribution of 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 distinctive of each material under certain load. Input from individual analysis study has proven on the stress value exerted from the model will not exceed its allowable stress applied before rupture based on Eq. (1). Hoop stress distribution also has been investigated to see the dispersion of the stress on the model and it is resulted as having lower internal pressure compared to ambient outer pressure value. For GRP material, both observation in individual layer and composite has found out that the fibre orientation at 90^0 is having high stress value due to the orientation of the fibre which is perpendicular to the load. At this orientation, fibre of the material tends to be a breakpoint fracture which it cannot endure to the load applied.

Stress distribution study on the composite has produced a stress-strain graph which obeys to the true engineering stress-strain plastic diagram with high ductility process. The analysis has identified on the upper yield stress point which is 22.55 MPa where plastic deformation will happen to the composite is the stress value passed beyond that point. Ultimate tensile strength is also being identified as 32.30 MPa and further necking process is developed in the model without arriving to the fracture point yet. Note that this stress-strain diagram developed from the model is based on an internal pressure load of 4.2 MPa.

Mesh sensitivity study has been carried on to access on the accuracy of the model. Mesh sensitivity study in this model has concluded to 6 number of mesh until it produces insignificant difference in stress value as the number of mesh is more sensitive. Number of mesh in finite element study is one of the important elements in processing the analysis and to ensure all mechanical loadings and properties has been checked in thoroughly to produce accurate result.

Choosing the right type of element is also crucial in determining the accuracy of the result. Solid three-dimensional C3D8R element type is the best to apply on cylindrical model of a pipeline. It works with reduced-integration element which refines the variety of element number over the model's volume to determine the respond and mechanical behaviour displays. Geometrical common shape as in cylinder is more favourable in applying this element type.

Finally, other than static stress analysis distribution, thermal analysis is also been performed to study the reaction of the composite with variation in thermal-load on the composite model. Stress value has increased as the temperature is elevated with 20 degree and 100 degree difference. Polymerization happened on the model as a result of the increasing temperature. This has increased the molecular flexibility of the molecule in a higher temperature.

5.1 Recommendation

Post-process on Abaqus CAE software is capable to provide enhance analysis on variety of mechanical loads for material study. Many utilities values have been provided in the software for the user to discover in producing more precise result for an analysis. Stress analysis distribution on TCP model can made on further improvement when more operational loads and factors is contributing in the analysis. More complex analysis on asymmetrical geometry can be done which includes in pipelines full-stream operation such as flanges, joints, bending load and other to produce more practically result of the material behaviour during operation.

This analysis study on stress distribution of TCP model can be improved by accomplishing an experimental result on the mechanical loads applied and compares with simulation study. This can be further established on the accuracy of the model and produce realistic contribution on the mechanical properties. Experimental study can also be performed on individual material of the composite to distinguish its mechanical properties under respective loads so that the consistency of the material design and properties input in the simulation will be more definitive on the mechanical behaviour on the composite.

This study on stress analysis distribution on TCP composite will contribute in providing more data and knowledge to achieve further continuity analysis so that the establishment of an overall study of the composite which touch every aspects of its operational loading can be applied in industry and help to improve the performance in its real life application.

REFERENCES

[1] J. Baron, "Use of Reinforced Composite Pipe (Non-Metallic Pipelines)," in "CANADA'S OIL & NATURAL GAS PRODUCERS," Canada, 2017, vol. 2007.

[2] I. K. Patrick Laney, Kenway Corporation, Augusta, Maine, "Use of Composite Pipe Materials in the Transportation of Natural Gas," *Idaho National Engineering and Environment Laboratory,* p. 69, July 2002.

[3] Ö. S. Sahin, A. Akdemir, A. Avci, and L. Gemi, "Fatigue Crack Growth Behavior of Filament Wound Composite Pipes in Corrosive Environment," *Journal of Reinforced Plastics and Composites,* vol. 28, no. 24, pp. 2957-2970, 2008, doi: 10.1177/0731684408094068.

[4] Y. Zhang and P. Y. B. Jar, "Comparison of Mechanical Properties Between PE80 and PE100 Pipe Materials," *Journal of Materials Engineering and Performance,* vol. 25, no. 10, pp. 4326-4332, 2016, doi: 10.1007/s11665-016-2274-2

[5] N. Merah, F. Saghir, Z. Khan, and A. Bazoune, "Effect of temperature on tensile properties of HDPE pipe material," *Plastics, Rubber and Composites,* vol. 35, no. 5, pp. 226-230, 2013, doi: 10.1179/174328906x103178.

[6] A. Wilson, "Thermoplastic-Composite-Pipe Flowline Helps Reduce Project and Life-Cycle Costs," *SPE-1116-0082-JPT,* vol. 68, no. 11, pp. 82-83, 2016/11/1/ 2016, doi: 10.2118/1116-0082-JPT.

[7] J. A. Hother, "Risk Minimization by the use of Failure Mode Analysis in the Qualification of New Technology - Recent Project Experience relating to intelligent well completions, permanent downhole gauges, expandables and new sand control technologies," presented at the Offshore Europe, Aberdeen, United Kingdom, 2005/1/1/, 2005. [Online]. Available: https://doi.org/10.2118/96334-MS.

[8] P. F. Liu and J. Y. Zheng, "Recent developments on damage modeling and finite element analysis for composite laminates: A review," *Materials &*

Design, vol. 31, no. 8, pp. 3825-3834, 2010, doi: 10.1016/j.matdes.2010.03.031.

[9] Z. C. Fan, X. D. Chen, L. Chen, and J. L. Jiang, "A CDM-based study of fatigue–creep interaction behavior," *International Journal of Pressure Vessels and Piping*, vol. 86, no. 9, pp. 628-632, 2009, doi: 10.1016/j.ijpvp.2009.04.003.

[10] J. L. Chaboche, "Continuum Damage Mechanics: Part I—General Concepts," *Journal of Applied Mechanics*, vol. 55, no. 1, pp. 59-64, 1988, doi: 10.1115/1.3173661.

[11] J. H. S. Almeida, M. L. Ribeiro, V. Tita, and S. C. Amico, "Damage and failure in carbon/epoxy filament wound composite tubes under external pressure: Experimental and numerical approaches," *Materials & Design*, vol. 96, pp. 431-438, 2016, doi: 10.1016/j.matdes.2016.02.054.

[12] M. Nurhaniza, M. K. A. Ariffin, A. Ali, F. Mustapha, and A. W. Noraini, "Finite element analysis of composites materials for aerospace applications," *IOP Conference Series: Materials Science and Engineering*, vol. 11, p. 012010, 2010/05/01 2010, doi: 10.1088/1757-899x/11/1/012010.

[13] S. Bobba, Z. Leman, E. S. Zainuddin, and S. M. Sapuan, "Failures Analysis of E-Glass Fibre reinforced pipes in Oil and Gas Industry: A Review," *IOP Conference Series: Materials Science and Engineering*, vol. 217, p. 012004, 2017/07 2017, doi: 10.1088/1757-899x/217/1/012004.

[14] A. Djebli, M. Bendouba, A. Aid, A. Talha, N. Benseddiq, and M. Benguediab, "Experimental Analysis and Damage Modeling of High-Density Polyethylene under Fatigue Loading," *Acta Mechanica Solida Sinica*, vol. 29, no. 2, pp. 133-144, 2016/04/01/ 2016, doi: https://doi.org/10.1016/S0894-9166(16)30102-1.

[15] A. Forghani, M. Shahbazi, N. Zobeiry, A. Poursartip, and R. Vaziri, "6 - An overview of continuum damage models used to simulate intralaminar failure mechanisms in advanced composite materials," in *Numerical Modelling of Failure in Advanced Composite Materials*, P. P. Camanho and S. R. Hallett Eds.: Woodhead Publishing, 2015, pp. 151-173.

[16] H. Liu, "Material Modelling of Structural Analysis of Polyethylene," Master of Applied Science, Civil Engineering, University of Waterloo, Waterloo, Ontario, Canada, 2007.

[17] S. Sedmak, Z. Golubovic, A. C. Murariu, and A. Sedmak, "Numerical simulation of tensile testing of PE 80 polymer specimens," *Thermal Science,* vol. 22, pp. 203-203, 01/01 2017, doi: 10.2298/TSCI170219203S.

[18] T. P. Sathishkumar, S. Satheeshkumar, and J. Naveen, "Glass fiber-reinforced polymer composites – a review," *Journal of Reinforced Plastics and Composites,* vol. 33, no. 13, pp. 1258-1275, 2014, doi: 10.1177/0731684414530790.

[19] V. S. Sastri, "5 - Corrosion processes and the use of corrosion inhibitors in managing corrosion in underground pipelines," in *Underground Pipeline Corrosion,* M. E. Orazem Ed.: Woodhead Publishing, 2014, pp. 127-165.

[20] G. Saville, S. M. Richardson, and P. Barker, "Leakage in Ethylene Pipelines," *Process Safety and Environmental Protection,* vol. 82, no. 1, pp. 61-68, 2004/01/01/ 2004, doi: https://doi.org/10.1205/095758204322777679.

[21] R. Khademi-Zahedi and M. Shishesaz, "Application of a finite element method to stress distribution in buried patch repaired polyethylene gas pipes," *Underground Space,* vol. 4, no. 1, pp. 48-58, 2019/03/01/ 2019, doi: https://doi.org/10.1016/j.undsp.2018.05.001.

[22] A. J. Sadowski and J. M. Rotter, "Solid or shell finite elements to model thick cylindrical tubes and shells under global bending," *International Journal of Mechanical Sciences,* vol. 74, pp. 143-153, 2013/09/01/ 2013, doi: https://doi.org/10.1016/j.ijmecsci.2013.05.008.

[23] B. D. Agarwal and J. N. Narang, "Strength and failure mechanism of anisotropic composites," *Fibre Science and Technology,* vol. 10, no. 1, pp. 37-52, 1977/01/01/ 1977, doi: https://doi.org/10.1016/0015-0568(77)90027-6.

[24] J. Senthil and V. Southararasu, "Study of tensile, compressive and impact properties with and without fibre orientation for epoxy glass fibre and carbon fibre composites," vol. 2015, pp. 350-352, 04/01 2015.

[25] P. Banakar, H. Shivananda, and H. B. Niranjan, "Influence of Fiber Orientation and Thickness on Tensile Properties of Laminated Polymer Composites," *Int. J. Pure Appl. Sci. Technol.*, vol. 9, 01/01 2012.

[26] F. Calheiros, M. Daronch, F. Rueggeberg, and R. Braga, "Effect of temperature on composite polymerization stress and degree of conversion," *Dental Materials,* vol. 30, 06/01 2014, doi: 10.1016/j.dental.2014.02.024.

[27] P. Yayla and Y. Bilgin, "Squeeze-off of polyethylene pressure pipes: Experimental analysis," *Polymer Testing,* vol. 26, no. 1, pp. 132-141, 2007/02/01/ 2007, doi: https://doi.org/10.1016/j.polymertesting.2006.09.006.

[28] X. Ming, G.-y. Huang, S.-s. Feng, G. McShane, and W. J. Stronge, "Static and Dynamic Properties of Semi-Crystalline Polyethylene," *Polymers*, vol. 8, p. 77, 03/28 2016, doi: 10.3390/polym8040077.

[29] L. Guo, J. Xiang, J.-P. Latham, and B. Izzuddin, "A numerical investigation of mesh sensitivity for a new three-dimensional fracture model within the combined finite-discrete element method," *Engineering Fracture Mechanics,* vol. 151, pp. 70-91, 2016/01/01/ 2016