Effects on Number of Supports on Vibration and Stress Distribution for Piping System

by

Mohammad Sobir Bin Abdul Basith
22582

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

JANUARY 2020

Universiti Teknologi PETRONAS
32610 Seri Iskandar
Perak Darul Ridzuan
Malaysia

CERTIFICATION OF APPROVAL

Effects on Number of Supports on Vibration and Stress Distribution for Piping System

by

Mohammad Sobir Bin Abdul Basith

22582

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF MECHANICAL ENGINEERING
WITH HONOURS

Approved by,

(Dr. Nabihah Sallih)

UNIVERSITI TEKNOLOGI PETRONAS

SERI ISKANDAR, PERAK

JANUARY 2020

CERTIFICATION OF ORIGINALITY

This is to certify that I am liable for the research submitted in this venture, that the original work is my own except as stated in the references and acknowledgments, and that the original work found herein has not been carried out or done by other sources or persons.

SobirBasith

MOHAMMAD SOBIR ABDUL BASITH

ABSTRACT

Fluid induced vibration are commonly faced by industry that are related with piping and pipeline. This phenomenon needs to take on seriously as it can leads to loss and danger to the plant. Additional of pipe support could reduce the vibrational behaviour of the system. However, the additional of pipe supports may lead to the increase in stress acting on the piping and consequently causes crack and leaking issue. This study investigates the effects of number of supports on pipe stress and vibrational behaviour of the piping system. Modal analysis was conducted to investigate the natural frequency and mode shape of the overall piping system, while static structural analysis was conducted in ANSYS to determine the pipe stress. In this study, vibration issue was managed to be improve by 81.62%. It is considered solved as the value of natural frequency is more than 15Hz which been recommended by ASME 2016. However, their stress value at critical points still exceeded the allowable stress limit by 74%. This shows that the optimizing of natural frequency and pipe stress of the piping could not be achieved through the addition of supports alone.

ACKNOWLEDGEMENT

Despite great challenges and new insights, all praises to Allah S.W.T Almighty have given the high spiritual and dedication to completing the 8 months span of this project. I am indebted to Universiti Teknologi PETRONAS for such a remarkable opportunity to have the optimal atmosphere for carrying out this project's investigation and study. I would like to express my sincere appreciation to Group Technical Solution, PETRONAS for such great cooperation and to support the research and investigation of pipe stress and vibrational activity by providing the raw data for further pipe analysis. I am profoundly grateful to my project supervisor Dr. Nabihah Sallih who has given me a great opportunity and new experience in this field of research. Nevertheless, I would like to express my great appreciation to the Final Year Project Coordinator for the Mechanical Engineering Programme, Dr. Tamiru Alemu Lemma, for all the guidance and support in preparing the seminars that clarified the right techniques for presenting, writing or generating good quality papers that are in line with management standards and requirements. Last but not least, thanks to the direct or indirect parties who contributed to this investigation through the provision of constructive feedback and constant support.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.1 Background

Piping vibration issue has been one of the major reasons for downtime, leakage, explosion and fire in industrial plants over the past 30 years. For example, a piping breakdown in a petrochemical plant in 1974 resulted in property damage of over $114,000,000 due to an explosion [1]. Another example that can be seen is an incident that happened in a nuclear pressurized water reactor power plants which was caused by 80 cases of cracks or leaks occurring in the piping system. It is really important to observe and control piping vibration amplitude such that the levels are acceptable. There are several things that can be modified to cater this issue such as the support structure, span length and the materials that been used for the piping system. Based on statistics made by UK Health & Safety Executive vibration contributes to 21% of piping leakage in plant [2].

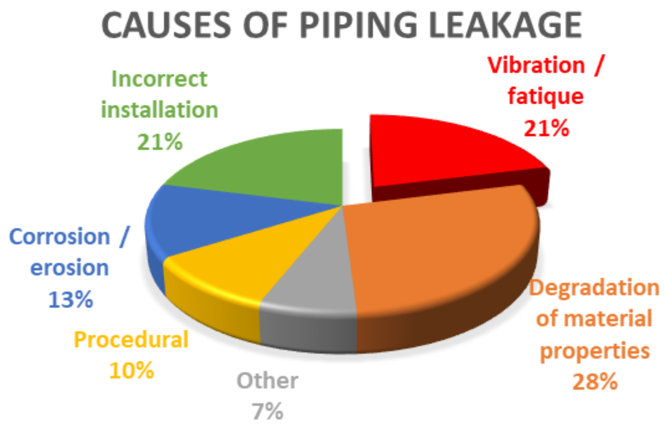


FIGURE 1.1: Chart of Causes of Piping Leakage. Adapted from [2]

A similar issue currently faced by PETRONAS Cari Gali Iraq. Piping on offshore oil and gas production and platform is crumbling which is caused by vibration. When the piping is affected by undesired vibration environment, vibration-induced failures like leakage and fire could occur and seriously give a big impacts and loss to the plant not only in term of production but safety operation of a piping system. The inlet of piping system is connected to the header while the outlet of piping system is connected to the degasser boot. The material of pipe used is pipe seamless ASME B36 have a diameter of 406.4mm and the thickness of the pipe wall is 7.925mm with an approximately 113m total length of the piping system that need to be investigated. There is a certain region in the piping that experienced high vibration issue. The pipe is occupied by the crude oil which have a density of 876.5 kg/m^3, viscosity of 9.4 cSt. The pipe is equipped by several types of support along the pipe which installed to support the pipe from collapsed and avoid excessive total deformation. Figure 1.2 shows the piping system of the plant.

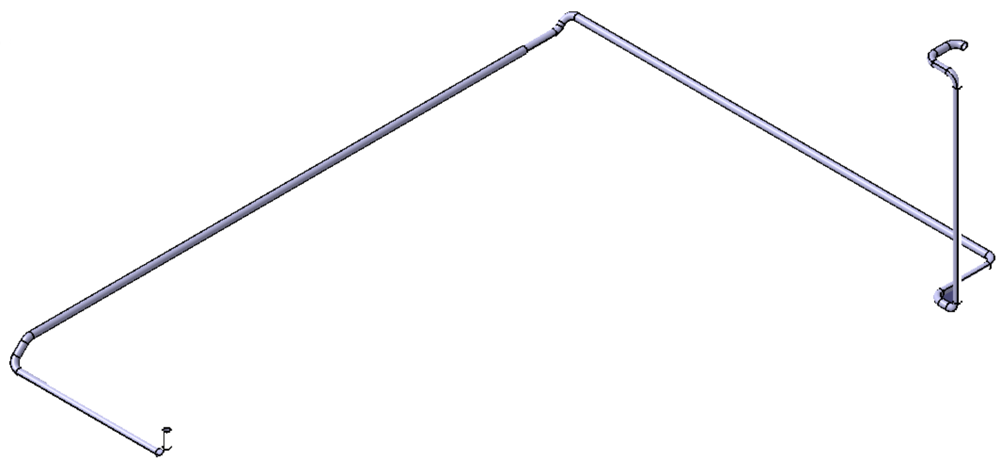


FIGURE 1.2: Piping system of the plant

Vibrations that occurred in the piping can be agitated by a few factors such as external factors, for this case come from the internal factors such as a pulsating by a motion of the medium inside the pipe [3]. Vibration can lead to immediate damage as a short and even a long-term effect. To analyse the effect of vibrations on stress strain state in the pipe profile is a very complex process thus the method of finite element is being applied to analyse the pipe [4].

The length of pipe may be one of the difficulties to determine and analyse the strain of pipe directly as it has approximately 113m of length and it is really hard to use resistance wire stresses in the hazardous profiles. A more acceptable and suitable method can be used which is to measure the vibration amplitude in selected pipe points which are determined by observing the reaction of the pipe by naked eyes.

1.2 Problem Statement

High stress and high vibration amplitude at certain area of pipe that may cause failure of the piping system whether it can collapse or cause danger at the plant. This happens due to the flow of the fluid inside the piping. However, the additional of supports can only cater the issue of vibration not the stress as the supports will limit the movement of the piping hence increase the stress along the pipe. It can cause the pipe to crack and cause dangerous to the plant even more. The current number of supports applied to the piping system are 17 supports.

1.1.1 High Vibration Amplitude

High vibration amplitude can be found at the nodes 62 to 71 which the range of amplitude produced is from 4.1mm/s up to 16.8mm/s. In additional, the natural frequency produced at these regions are from 2.5Hz to 14.2Hz. The natural frequency needs to be 15Hz and above to ensure the system is safe to operate. The location of nodes can be referred at Figure 1.3.

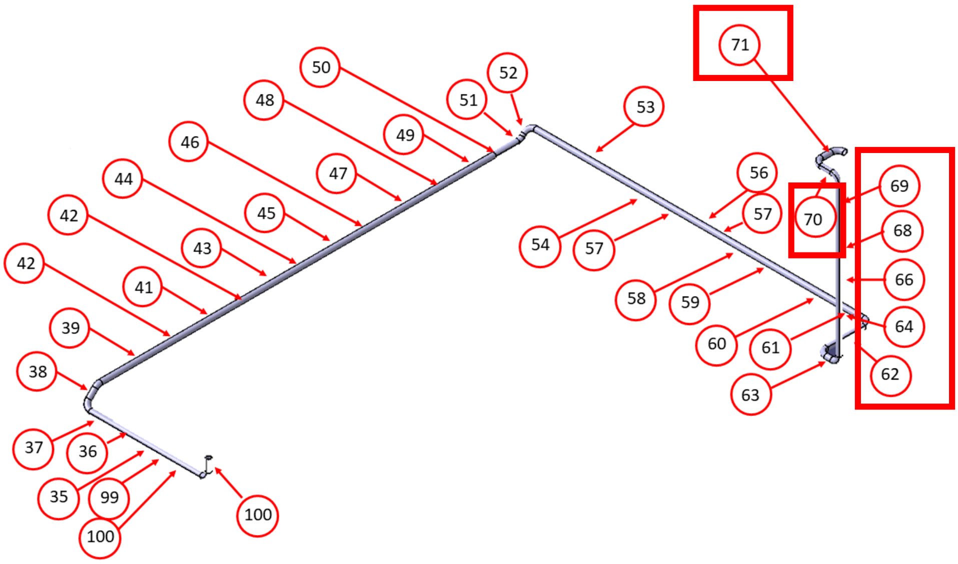


FIGURE 1.3: Nodes in the piping

1.1.2 High Stress

The piping system has a length of approximately 113m which can be considered as long piping system. Due to the long piping system, it can cause high stress at the certain location of the pipe that does not have enough support or suitable support to reduce the pipe stress. Optimum number of supports need to be installed at the piping system to optimise vibration performance and static structural performance.

1.3 Objective and Scope of Study

The objective of this study is to determine the number of support by investigating the relationship between the number of pipes supports and their locations and on stress pipe and vibration at respective area. This study is limited in the simulation by the software scale. This study also neglects the parameters that have low possibility to affect the behaviour of the pipe stress and vibration such as wind velocity and seismic effect. The flow in the piping is assumed to be a single phase flow.

CHAPTER 2
LITERATURE REVIEW

2.1 Fluid Induced Vibration

Highly energized flow of fluid can lead to fluid induced vibrations [5]. Severe or abnormal high piping vibration may be a sign of failure. Piping system can be seriously in threat if vibration level is undetected [6]. Vibrations are undesired and often unanticipated. There is tendency of structures become more flexible as fluid flow in the pipes. There are two different cases which affect the instability mechanism of flexible pipes in conveying fluid which are: (i) unstable vibration caused by the fluid flow when velocity exceeds a critical value, and (ii) vibration that occurred because of oscillating fluid flow [7]. Oscillating fluid flow is where the pipe is connected to reciprocating fluid machines and generates excitation force that can cause vibration. Unstable vibration that caused by the fluid flow is where there is symmetric vortex shedding behind the well. When the critical value is exceeded the probability of resonance to occur become high [8].

2.2 Vortex-induced vibrations (VIV)

Vortex induced vibration (VIV) is a phenomenon that can be observed in many potential areas such as chemicals industry, offshore structures such as heat exchange tubes, bridges, power lines, cables and risers. Due to the force caused by the vortex, the cylindrical structure can be subjected to transverse vibration. As the shedding frequency approaches the natural frequency of the structure, a lock-in effect occurs. It can easily excites the body to lateral resonance, causing relatively large vibrations and lead to fatigue failure to the cylindrical structure [9].

2.2.1 Vortex-induced vibrations of transverse and in-line pipe strain amplitude

Recently, there is vibration characteristics study made by Guo and Lou in 2008 investigated theoretically and experimentally on a riser with an axial internal fluid in the external cross flow. Their results showed that the transverse and in-line pipe strain amplitudes increased with the increase of the internal flow rate [10].

2.2.2 Vortex-induced vibrations of pipes conveying fluid in the subcritical and supercritical regimes

In this study, the dynamic behaviour of fluid transport pipes exposed to vibrations caused by vortices has been analysed. Effects of both subcritical and supercritical internal fluids on the nonlinear dynamics of pipes was been investigated. The inner fluid velocity has been shown to have a strong influence on the pipe's nonlinear dynamics, especially for pipe systems with supercritical fluid flow. The pipe displays intermittent motion and acceleration when the inner liquid velocity is in the subcritical region. The amplitude slowly decreases as the internal liquid velocity in the lock-in region increases. When the internal fluid velocity is in the supercritical region, the lock-in pipe has various dynamic behaviours such as reverse period doubling bifurcation, periodic and chaotic motion [10].

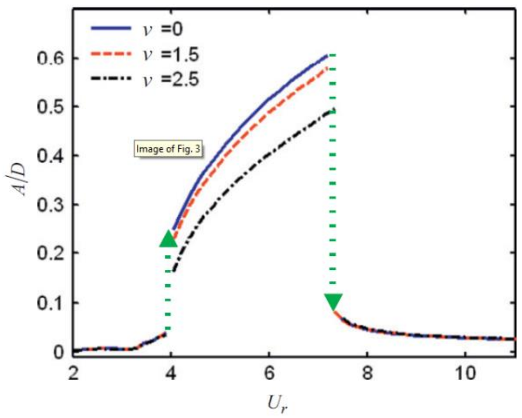


FIGURE 2.1: Relationship between the peak amplitude and the decreased outer fluid velocity at different internal fluid speeds below the critical value of the first mode. Adapted from [10].

2.3 Behaviour of 90° elbow pipe

In the elbow area, the velocity of fluid close to the inside of the elbow increases and the velocity of fluid close to the outside of the elbow slows down and create a large gradient of stress. An unbalanced force occurs in the liquid as a result of the pressure gradient and a secondary flow field is generated downstream of the elbow. Therefore, large pressure gradients produce spikes in the flow, increase friction rates and mechanical disturbances and noise due to motion [11].

2.3.1 Flow characteristics of the fluid at the elbow

Eisinger etc. [12] and Modi and Jayanti [13] carried out several tests and measurements based on CFD to show the pressure drop in the elbow fitted with a guide vane. Experimental results have shown that guide vanes can be used effectively for elbows with a radius ratio (defined as the ratio of the nominal (or medium) elbow radius R to the inner elbow radius *Ri* of the pipe). The position of the guide vane was found to reduce the original elbow pressure loss by about 20%.

Guide vanes in the proper position of the elbow can lower the total vibration level and total sound output level of 90 elbow piping. The volume of reduction depends on the number of Reynolds-the higher the number of Reynolds, the greater the decrease in the total level of vibration and the maximum level of sound energy [11].

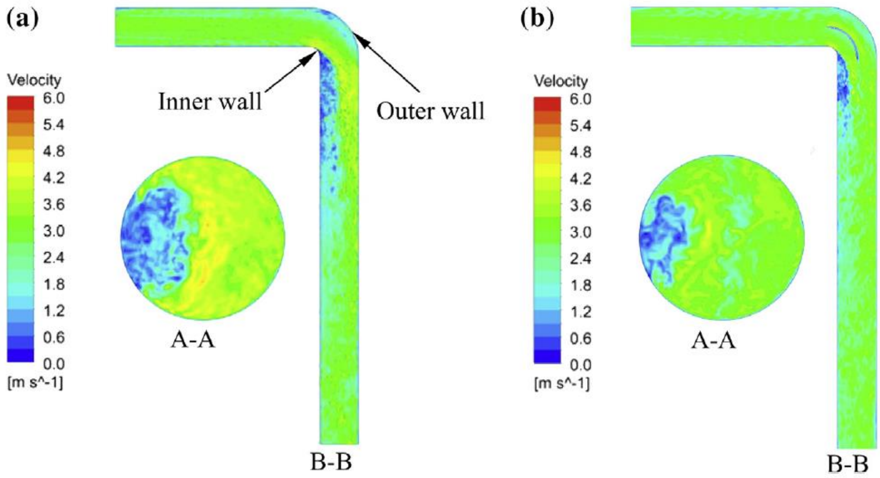


FIGURE 2.2: Fast velocity distribution on the cross-section A-A and B-B at Re =1 x 105. (a) Without guide vane; (b) with guide vane. Adapted from [11].

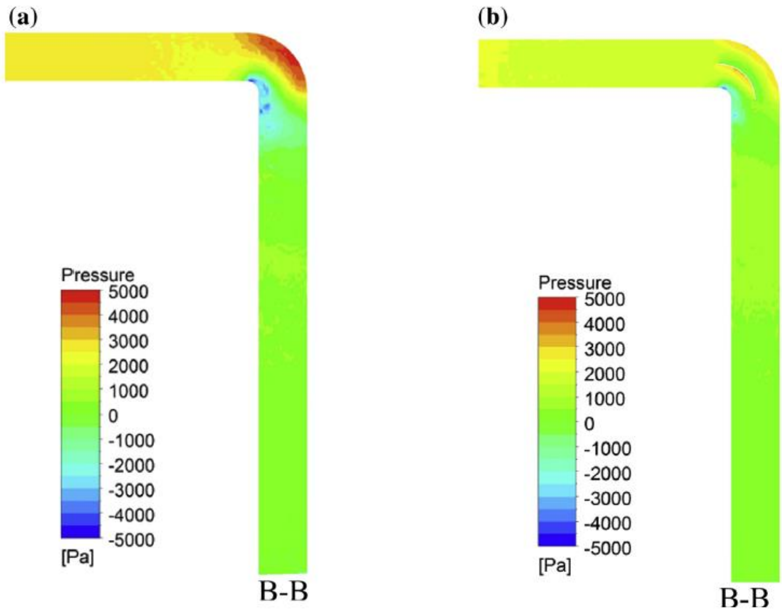


FIGURE 2.3: Instant pressure distribution on the B-B cross section at Re =1x105. (a) Without guide vane; (b) with guide vane. Adapted from [11].

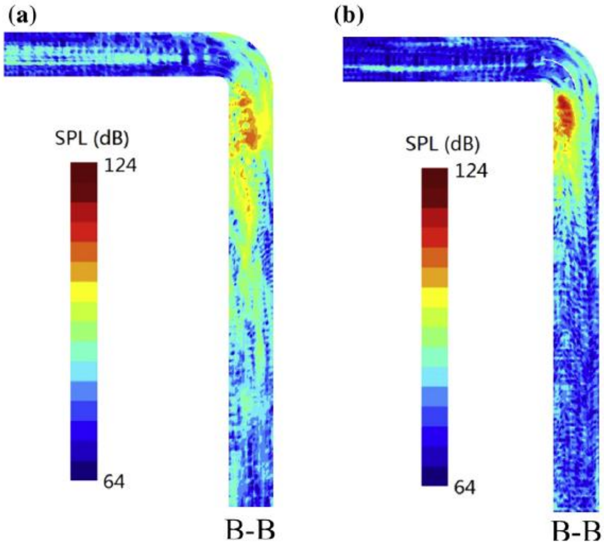


FIGURE 2.4: Sound pressure level contour of bend without/with the guide vane. (a) Without guide vane; (b) with guide vane. Adapted from [11].

2.4 Pipe Stress

Pipe stress analysis need to be analysed to ensure there is no crack at the piping. There are terms need to be discussed when relate with stress which are force and moment. Force is a vector quantity with stress, pressure, or shear impact direction and magnitude. While the quantity of the vector is called the position and intensity of the twisting and bending effects. Different types of loads such as thermal expansion and dead weight will enhance forces and moments. Stress is the force per unit area while the change in length divided by the original length is called as strain [14]. There are 5 major types of pipe stress which are hoop stress, axial stress, bending stress, torsional stress and fatigue stress. The hoop tension is due to the internal or external pressure applied to the tube. Thermal expansion, pressure expansion and applied force are the causes of axial growth. Torsional tension is generated by the resulting stress caused by the moment of rotation around the axis of the tube and by the movements of the organ. Continuous cycling of the stresses present in the pipe causes fatigue stress. [15]. Numerical analysis allows to determine the pipe stress distribution of piping [16]. Deformation of pipe can occur due to pipe stress. It can be estimated numerical simulation by using elasto-plastic finite element [17].

2.4.1 Pipe model and Modification.

Model of the pipe will be built by using finite element method. There are usually two models for pipeline stress analysis: a beam model and a shell model. The shell model is suitable for local analysis of pipelines. Beam models are typically used for stress analysis of long-distance pipelines [18]. Piping supports must be spaced in respect of which they are capable of placing a support at any desired position, keeping it in line within limits that allow drainage and avoiding excessive bending stress from uniform and concentrated loads between supports [19].

2.4.2 Pipe stress in vibrating pipeline

The straight part of the pipe, where d = 273 mm diameter and l = 9 m length is supported at both ends, is taken into account. The amplitude of vibration velocity at the center of gravity of the pipe is A = 200 mm · s-1. When the vibration frequency is f = 3 Hz, the stress in the pipe profile at the maximum vibration amplitude is σ_{zred} = 100 MPa. At f = 9.5 Hz, the vibration velocity with the same amplitude is barely associated with a stress of σ_{zred} = 50 MPa. This shows that the high frequency produced lesser stress on pipe [3].

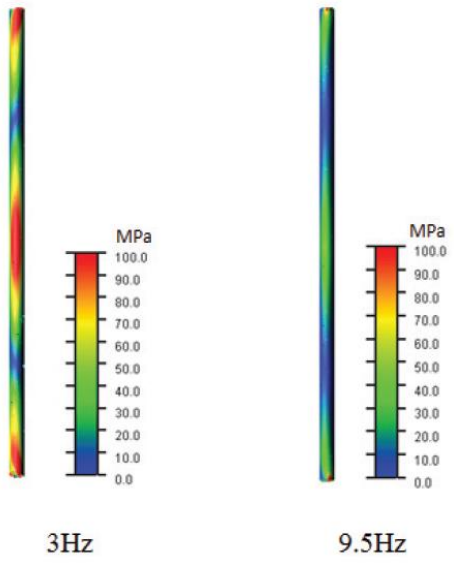


FIGURE 2.5: Stresses in the segment of the pipe. Adapted from [3]

2.4.3 Pipe Stress due to Piping Flexibility

One of the factors that cause pipe stress is thermal. The studies of piping flexibility have been commonly executed on piping system in order to ensure the static force, static stresses and static deflections due to loads and temperature are within the safe limit to be operated. There is a problem when supports are being assumed as the pipe are rigidly anchored as it lead to vibration issue. [20]. This study can be done by using finite element method. The pipe of the body will expand when heated and generally cannot proceed freely in a continuous medium and stresses due to heat. Due to heat, thermal stresses are added to the stress and affects internal and external pressure in pipe material [21].

2.5 Summary of Literature Review

The velocity of the medium inside the pipe can affect the vibrational behaviour of the pipe system. Supercritical regime can cause high vibration at the part. Pipe elbow produce high velocity of the flow and may produce higher vibration. The presence of the support at the elbow may solve the vibration issue however the number of supports may affect the amount of stress applied on the piping. Finite element method is the method that are widely used in vibration and stress analysis. In addition, numerical computational can eases the process of analysis

CHAPTER 3
METHODOLOGY

This chapter describes the procedures that need to be conducted to achieve the main objectives. Model of the piping system was developed in the software called AUTOCAD. The model was then transferred to the software called ANSYS. The geometries and orientations of the model were created follow exactly like the real-life piping system which portraits in the drawing. ANSYS software can be used to get the modal analysis of the system. An earlier stage of this study needs to get the modal analysis and static structural of the pipe. Then need to develop load inside the pipe act as medium fluid and get the modal analysis from the ANSYS. This need to be done so that the pipe system experiences the same situation with the real pipe which when the pipe is filled by the fluid. The parameters then are varied according to the value of pipe stress, total deformation and natural frequency in static structural and modal analysis. The variation that have been created followed by several recommendations [19].

3.1 Material

The material of piping used is ASTM A106 grade B seamless pressure pipe system class of CL 150. It is used to transport fluids and gases that produced high pressure and temperature. The allowable stress limit is 137.89MPa [22]. The physical properties of this material have a density of 7833.43 kg/m^3. The chemical composition of this material can be referred in Table 3.1. [23].

TABLE 3.1: Chemical composition of ASTM A106 grade B. Adapted from [23]

Chemical Composition (%)	
Chemical properties	**Grade B**
Carbon (C) Mx.	0.3
Manganese (Mn)	0.29 – 1.06
Phosphorus (P) Mx.	0.035
Sulphur (S) Mx.	0.035
Silicon (Si) Mx.	0.1
Chromium (Cr) Mx.	0.40
Cooper (Cu) Mx.	0.40
Molybdenum (Mo) Mx.	0.15
Nickel (Ni) Mx.	0.40
Vanadium (V) Mx.	0.08

3.2 Load

Load or self-weight need to be set to represent the fluid inside the piping system. Density of crude oil is decided to be 919 kg/m3 [24]. The load is set to be based on the calculation. Calculation can be made by using this formula:

$$\rho = \frac{m}{v} \quad (1)$$

ρ = Density of crude oil

m = Mass of fluid, kg

v = Crude oil, m^3

The calculated load values are as follows;

m = 14,573kg

3.3 CAD Modelling Using CATIA Based on Isometric Drawing

CAD modelling is been developed by using CATIA based on isometric drawing. The geometries and orientation also portraits the real situation at the plant. Figure 3.1 shows the method to create CAD modelling using CATIA.

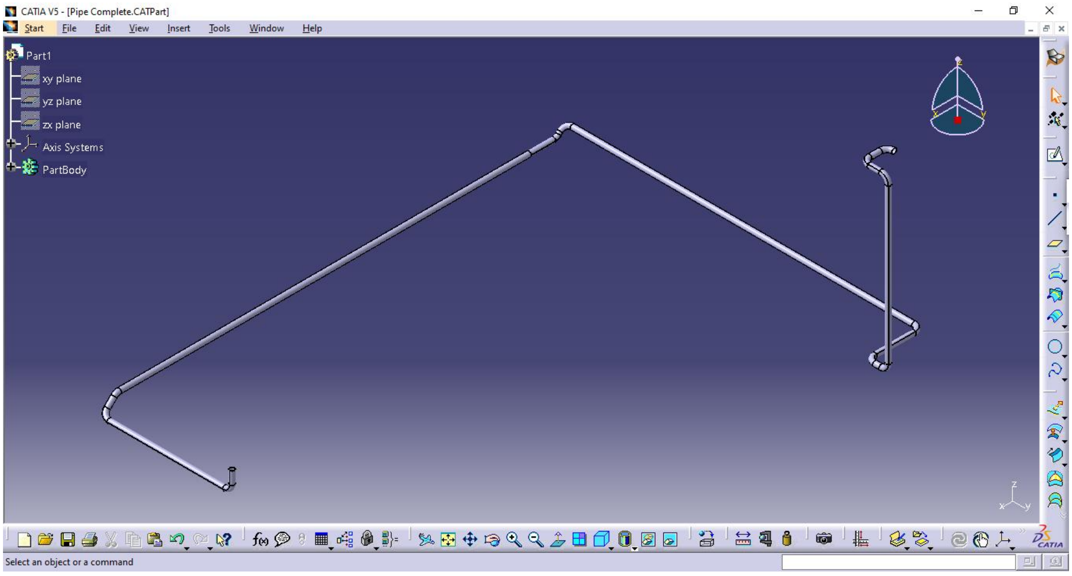


FIGURE 3.1: Model of piping created in the CATIA software

The model needs to be created between anchor at degasser boot to line stop of the piping system. These are because of following reasons:

1. Take into consideration of stresses and total deformation occurring at nozzle which connected from pipe to degassing boot.
2. Take account for thermal expansion at the line stop. Refer Figure 3.2

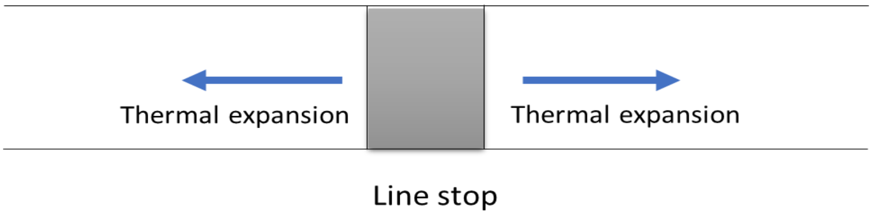


FIGURE 3.2: Direction of thermal expansion at the line stop

3.4 Setting Parameters in ANSYS

Parameters are being set in the ANSYS. For this case the support is the parameters that need to be set. There are several types of support and pipe guide around the piping. Support is used to ensure the piping is not collapsed or facing huge total deformation. Each type of supports has different mechanism and boundary condition that have been created. Table 3.2 shows the supports that is used in the piping. Figure 3.3 shows the method to develop support location.

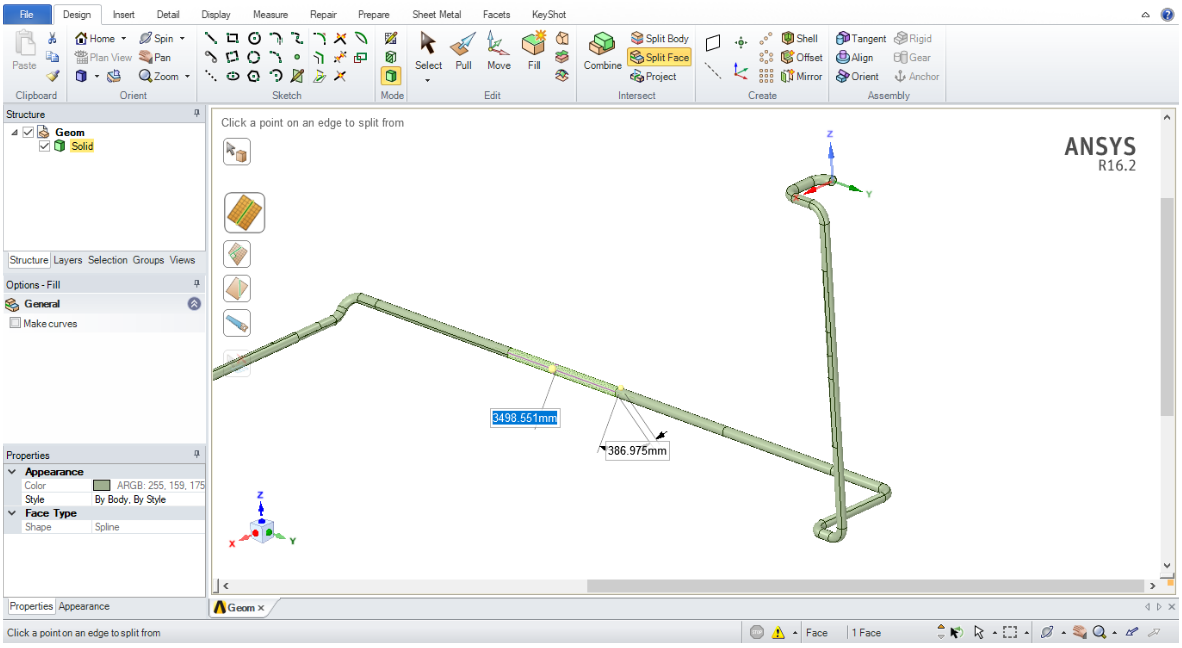


FIGURE 3.3: Develop the location of the support

TABLE 3.2: Existing types of piping supports

Support	**Boundary Conditions**					
	X	**Y**	**Z**	**RX**	**RY**	**RZ**
Fixed Support 1	/	/	/	/	/	/
Vertical Guide 1	Free (limit)	Free (limit)	Free	/	/	Free
Vertical Trunnion	Free	Free	/	/	/	Free
Vertical Guide 2	Free (limit)	Free (limit)	Free	/	/	Free
Saddle Plate 1,2,3	/	Free	/	/	Free	/
Line Stop 1	/	/	/	/	/	/
Saddle Plate 4,5,6,7,8,9	/	Free	/	/	Free	/
Line Stop 2	Free	/	/	Free	/	/
Saddle Plate 10	/	Free	/	/	Free	/
Fixed Support 2	/	/	/	/	/	/

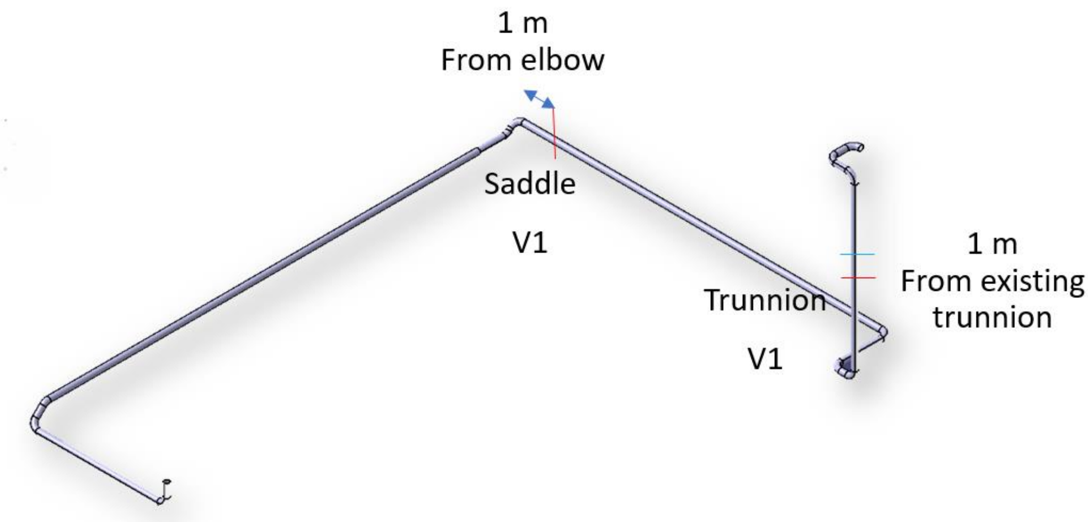


FIGURE 3.4: The target location area to add support

Figure 3.4 shows the location of addition support determined by detecting the critical area which experience high stress and lowest natural frequency along the pipe. The distance for the additional support is fixed which 1m apart from existing support. The type of supports determined by the support that exist as a foundation. Defining boundary conditions by using organized conversion method [25].

Variation of supports are then been created to observe the change of the vibration and stress behaviour. Basically, method that performed are try and error. Based on research done by Zhang T, et al, 2015, additional number of supports in elbow can reduce the vibration produced. Optimum number of supports need to be determined to solve the vibration issue and pipe stress distribution.

3.5 Meshing

Meshing is an integral part of the computer-assisted simulation process. Mesh will affect the accuracy, convergence and duration of the solution. The finer the mesh, the longer time ANSYS takes to get the result and produce the more accurate result when comparing with coarse mesh. In this study, the mesh was first developed by using the coarse mesh. This is to ensure the simulation needs short time to simulate. After the simulation have been done successfully, the finer mesh will be created again to get more accurate result. Convergence study need to be done to determine the optimum mesh size to be used to obtain the shortest computational time with compromising the accuracy of the actual result. Tetrahedral mesh can be used to create high quality of mesh and low mesh density [26]

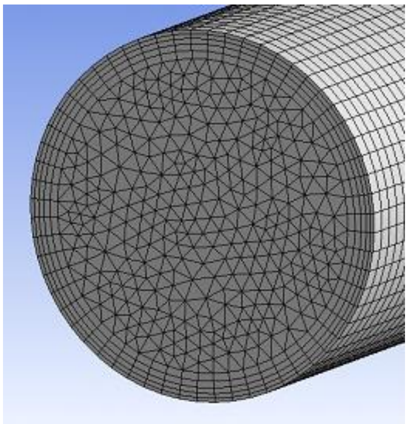


FIGURE 3.5: Mesh in piping using ANSYS

3.6 Static Structural Analysis

Static structural need to be determined by using ANSYS to determine the stress along the piping. It can be shown by contour pattern and easy to determine the location of pipe which have the high stress

3.7 Modal Analysis

Modal analysis is a method to investigate and study the dynamic properties of the systems in the frequency domain. For this study, the modal analysis that need to be discovered is to identify the “hot spot” for stress and vibration.

3.8 Outputs

The outcome that need to acquire need to tackle the objective of the study which are to come out with the modal analysis and static structural such as total deformation and stress analysis. There will be 4 results that need to be the outcome of this study which are stress analysis with and without load applied, and total deformation with and without load applied. Figure 3.6 shows the flow process to get the output required.

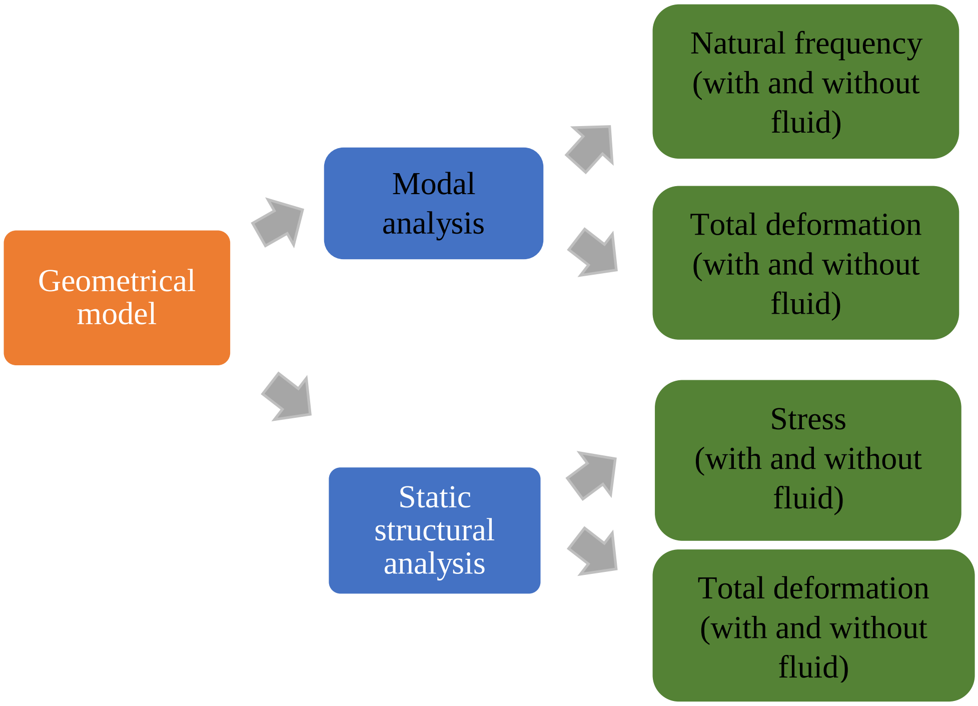


FIGURE 3.6: Flow process to get the output

3.9 Gantt Chart

This section shows the key milestones and expected progress for both final year projects 1 and 2.

3.9.1 Gantt Chart for Final Year Project 1

TABLE 3.5: Gantt chart for final year project 1

Activities	Week													
	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Introduction														
FYP title familiarization														
Literature review														
Support at the pipeline and related study														
Extended proposal drafting						11/10/2019								
Extended proposal submit														
Simulation														
Data extraction														
CAD modelling														
Support tracing														
Proposal defence								21/10/2019	31/10/2019					
Compile result and analysis														
Interim report drafting														
Interim draft report submit													29/11/2019	
FYP 1 Summary														
Interim report submit														06/12/2019

Indicator : Completed task   Date Key milestone

3.9.2 Gantt Chart for Final Year Project 2

TABLE 3.6: Gantt chart for final year project 2

Activities	Week													
	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Simulation														
Model and simulate model														
Draft progress report														
Submit progress report							14/02/2020							
Analysis														
Result correlation														
Result fine-tuning														
Analyze study														
FYP 2 Finalization														
Pre-Sedex preparation														
Pre-Sedex										06/03/2020				
Draft final report														
Final draft report submission											13/03/2020			
Dissertation														
Submit dissertation (Soft-copy)												20/03/2020		
Draft technical paper														
Submission of technical paper												20/03/2020		
Viva presentation														
Viva													27/03/2020	
Dissertation submission (Hard-copy)														03/04/2020

Indicator : Completed task | Date Key milestone | Expected progress

3.9.3 Key Milestones

This section shows the key milestones set for final year project 1 and also final year project 2

3.9.3.1 Final Year Project 1

TABLE 3.7: FYP 1 key milestones

Key milestones	Due Week	Date
Extended proposal submission	Week 6	11/10/2019
Proposal defence	Week 8-9	21/10/2019 - 31/10/2019
Interim report draft submission	Week 13	29/11/2019
Interim report submission	Week 14	6/12/2019

3.9.3.2 Final Year Project 2

TABLE 3.8: FYP 2 key milestones

Key milestones	Due Week	Date
Progress report submission	Week 7	14/02/2020
Pre-Sedex	Week 10	6/3/2020
Final draft report submission	Week 11	13/3/2020
Dissertation submission (soft-copy)	Week 12	20/3/2020
Technical paper submission	Week 12	20/3/2020
Viva	Week 13	27/3/2020
Dissertation submission	Week 14	3/4/2020

3.10 Project Flow

Figure 3.6 illustrates the project flow of this study. There are two methods were conducted to get the outcome which are modal analysis and static structural analysis. Self-weight then needs to be inserted into the procedure to get the static structural analysis and modal analysis with the present of fluid inside the piping. All the results taken will be documented before the end of this project.

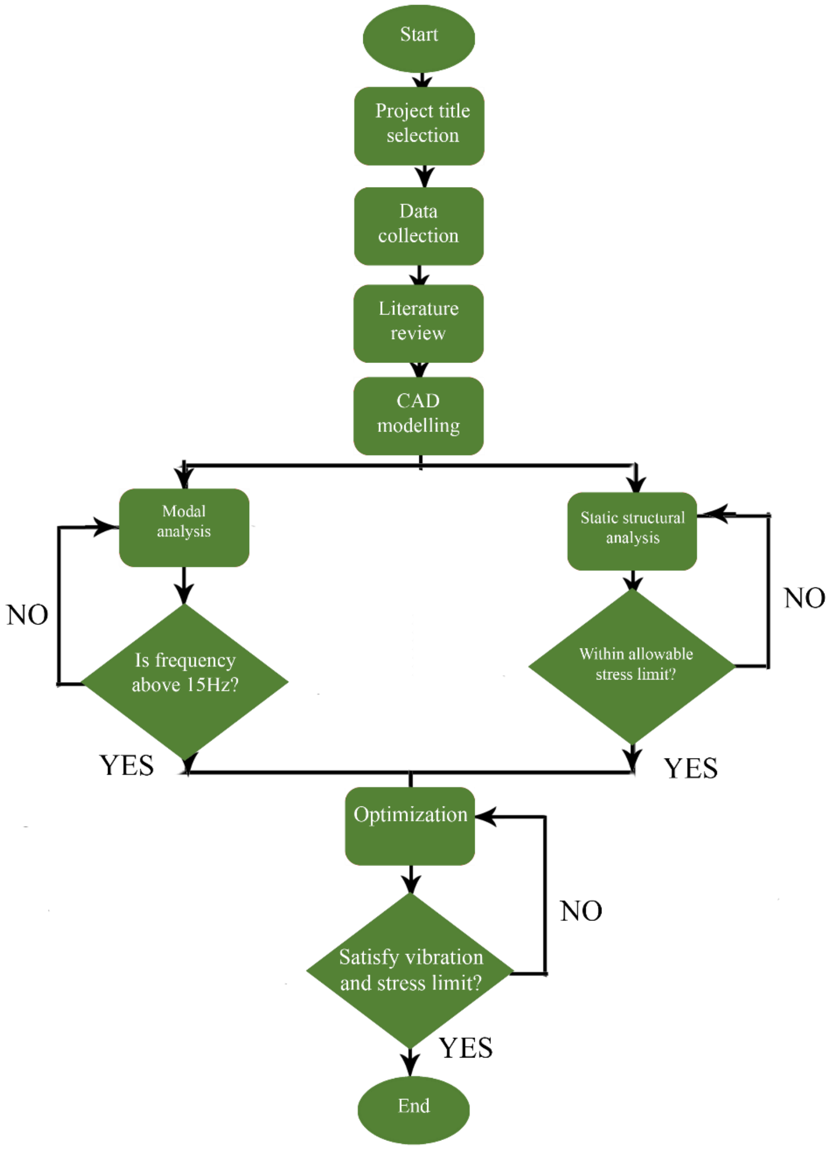


FIGURE 3.7: Flow chart of this study

CHAPTER 4
RESULTS AND DISCUSSION

This chapter covers all results and discussions that have been obtained through simulations that has been done. There are total of 72 sets of simulations that has been created for this study. In this chapter, it will be sectioned by several process. Firstly, pipe modelling based on real drawing and specification. Next, results will be shown and discussed accordingly. Based on objective, results that need to be recorded are static structural and modal. From static structural analysis, the result of stresses and maximum total deformation will be determined while in modal analysis, the result of maximum total deformation, mode shape and natural frequency will be obtained. Both static structural analysis and modal analysis will have the condition of pipe with self-weight and without self-weight. Value of pipe stress must be lower than 137.89MPa [22] while natural frequency must be higher than 15Hz [27]. The effects of number of supports will be studied and discuss based on variations of number of pipe that have been created.

4.1 Model

Piping with the length of 113m with the existing of support was being drew follows the specification of the real model including the elbow and orientation of the pipe. Figure 4.1 shows model of pipe with existing pipe supports. This model was used for further study. There are total of 17 pipe supports that complete the whole model of pipe.

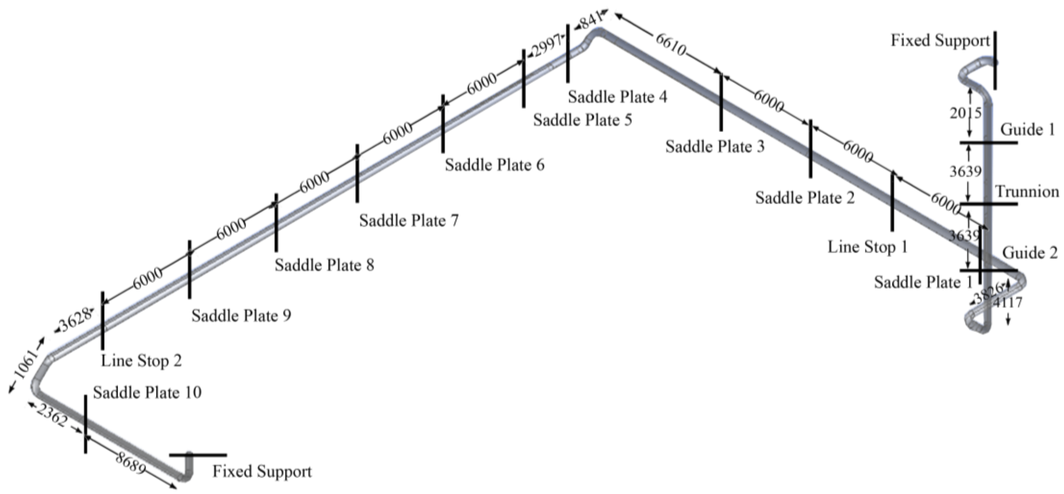


FIGURE 4.1: Model of pipe with existing pipe supports

4.2 Convergence Study

Since the model has been completed, mesh was created to improve the accuracy of data [28]. The type of mesh used is tetrahedral. This is because it will be more accurate for some reasons. Coarse and big meshes were used in the earlier stage of the simulation to save more time, however, it resulted in a low accuracy. A finer mesh was used at the next stage of the simulations. Mesh independency study was done to identify the optimum number of elements to be applied for all models.

Based on Figure 4.2, results start to converge into a constant number. It is shown at the range number of elements between 445,146 to 597,862. Percentage different of result of total deformation from number of elements 445,146 and number of elements 521,504 is 0.12% while the percentage different of total deformation from number of elements 521,504 and 597,862 is 0.08%. Number of elements of 445,146 selected to be used for further study as it is considered to be an optimum number elements. It will benefits with low computational time with high accuracy of result.

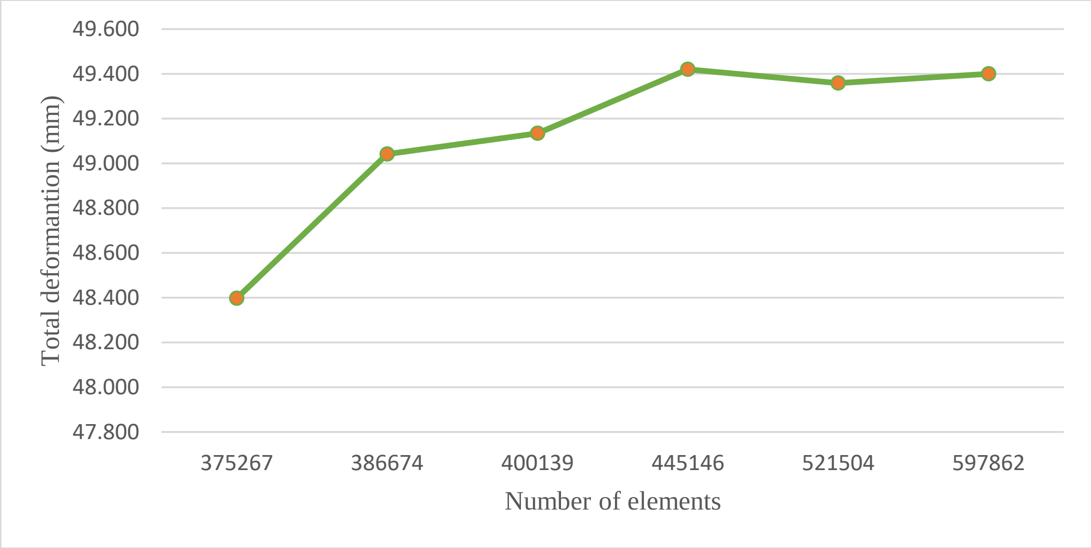


FIGURE 4.2: Graph of Mesh independency study

4.3 Static Structural and Modal Analysis without Modification

Model was being created based on specification shown in Figure 4.1. Type of result that had been obtained from static structural analysis are stress and total deformation while in modal analysis, the result obtained are maximum total deformation and frequency (Hz). Boundary condition of the pipe are shown in Table 3.2. Both of the analysis are being sectioned into two parts which are with self-weight and without self-weight. Self-weight are considered as fluid inside pipe which weight 14,573kg.

4.3.1 Location of Points to Observe Stress Value

Location of pipe to observe stress value are shown in Figure 4.3. These locations are selected after considering the pipe support distribution and orientation of the pipe which may produce high stress value. There are total of 7 points located along the pipe. All result of stress value will use the same position as points to study the effect of number of pipe supports to the pipe stress.

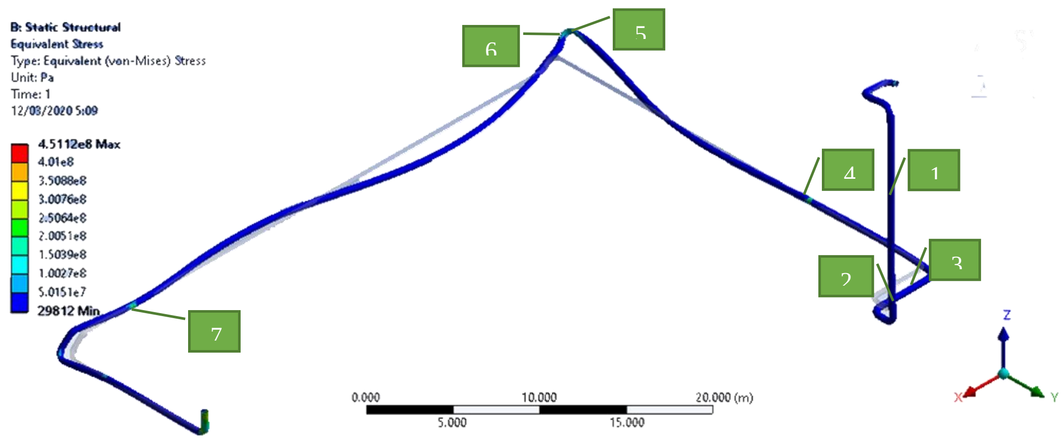


FIGURE 4.3: Location of pipe to observe the stress value

4.3.2 Static Structural with Self-Weight from Original Version

Based on Figure 4.4, points that exceed allowable stress limit are point 4 and point 7. Stress value produced from point 4 is 145.12MPa while point 7 produced 207.15MPa. Both of these points are where line stop are being placed. Rigid support creates greater force than saddle support [29]. Boundary condition that set has caused it over constraint and produce high stress value at particular area. The set up for boundary condition line stop is shown in Table 3.2.

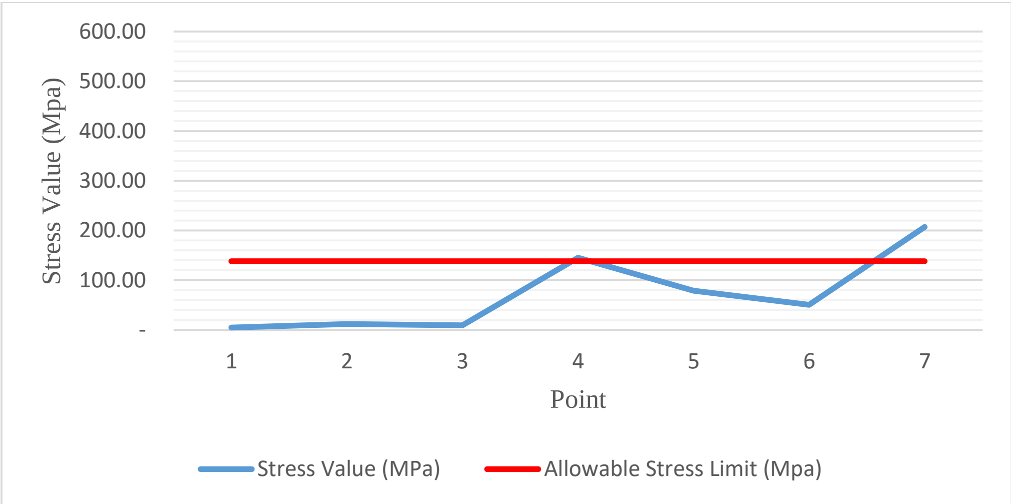


FIGURE 4.4: Graph of Stress value for original condition with self-weight

Figure 4.5 shows total deformation of the pipe. The maximum total deformation is marked by red colour which deformed by 49.42mm. This is because of pipe support distribution and design of the pipe itself. Pipe support distribution has been shown in Figure 4.1. Distance between saddle plate 3 and saddle plate 4 is 7.45m which has the highest distance between supports in the system. In addition, the design of the pipe which it goes incline and have 90° turn after the incline part effect the deformation of the pipe.

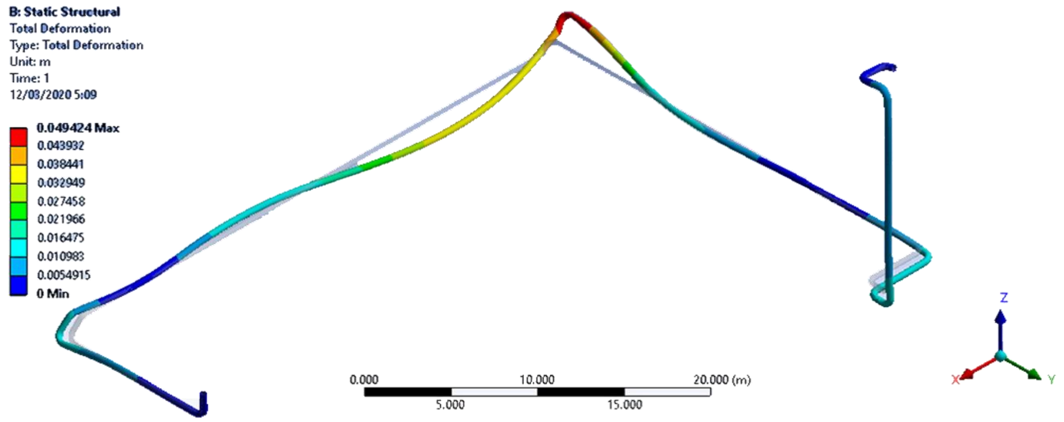


FIGURE 4.5: Total deformation of pipe with self-weight

4.3.3 Static Structural without Self-Weight from Original Version

Figure 4.6 shows the result of stress value for original condition without self-weight. The result shows that the stress value at point 4 and point 7 exceed the allowable stress limit. The reason behind this has been discussed in section 4.3.2.

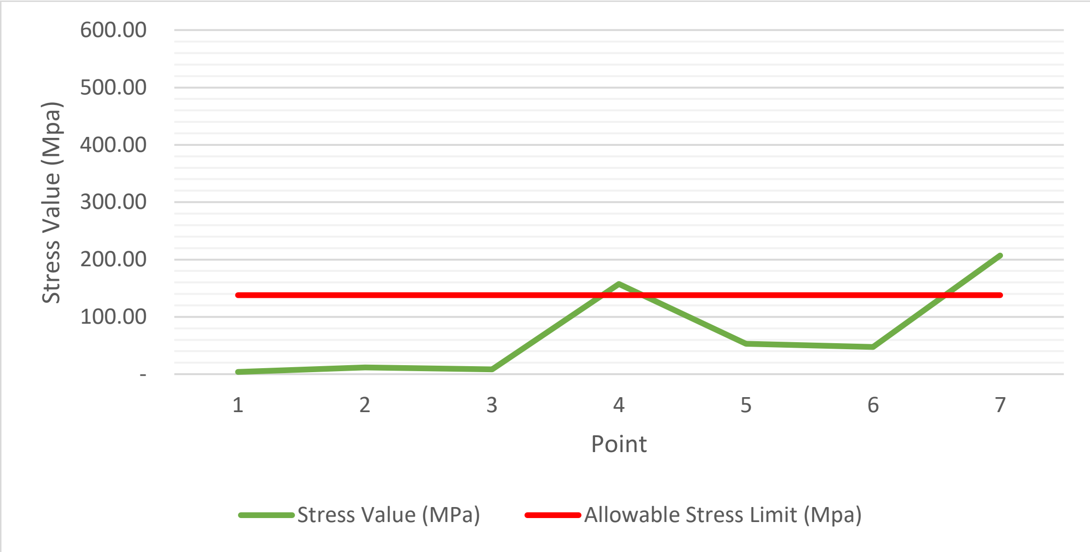


FIGURE 4.6: Graph of Stress value for original condition without self-weight

Figure 4.7 shows total deformation of pipe without self-weight. The maximum value of total deformation is 49.06mm.

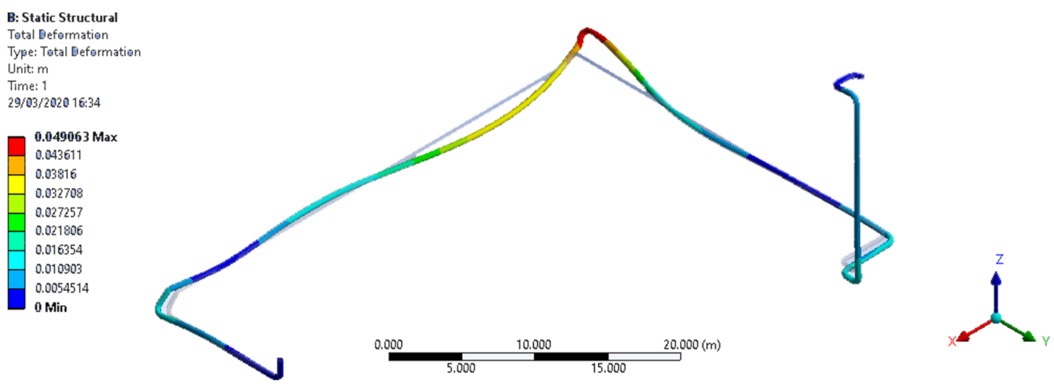


FIGURE 4.7 Total deformation of pipe with self-weight

4.3.4 Comparison between Result of Static Structural with self-weight and without Self-Weight from Original Version

Figure 4.7 shows the comparison between results of stress value with self-weight and without self-weight. There is only small difference between the results mentioned. However, stress value without self-weight is smaller than stress value with self-weight. The average percentage difference of stress value between these two conditions is 8%. In addition, difference of total deformation between both conditions is 0.36mm equivalent to 0.73%.

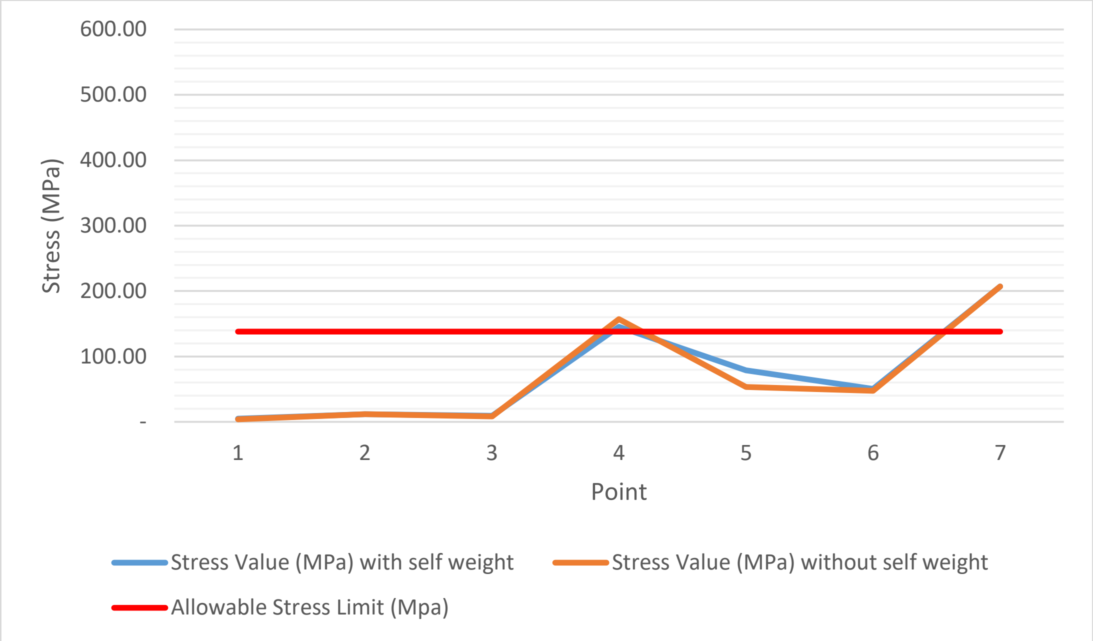


FIGURE 4.7: Graph of Comparison between result of stress value with self-weight and without self-weight from original version

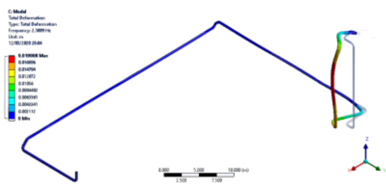
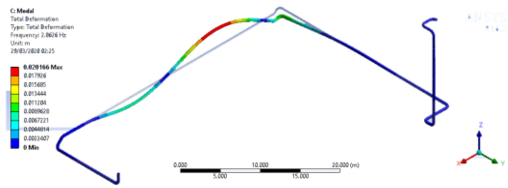
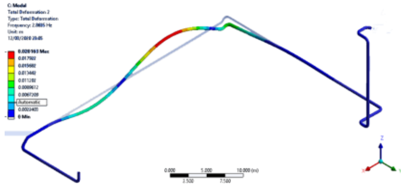
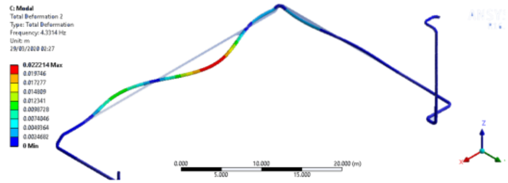
4.3.5. Limitation of Boundary Condition for Modal Analysis.

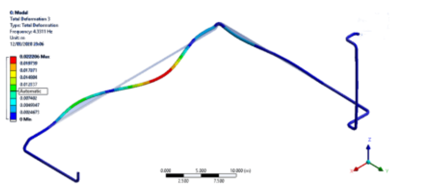
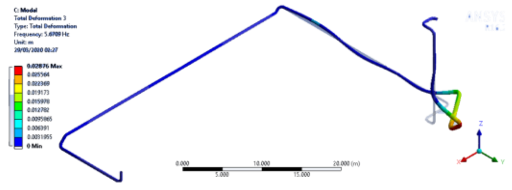
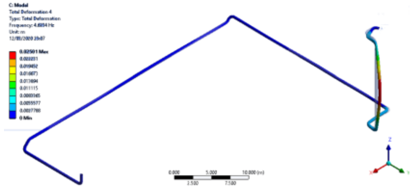
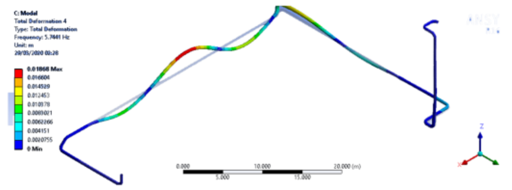
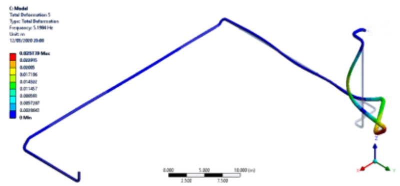
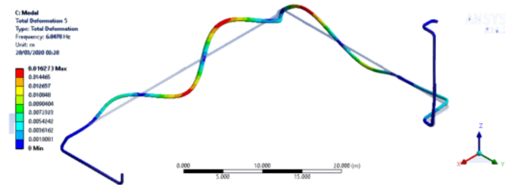
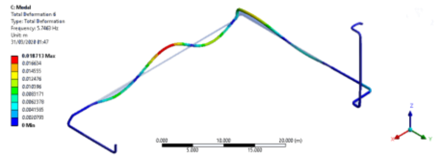
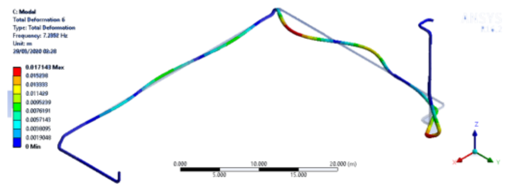
Boundary condition criteria has been shown in Table 3.2. However there is limitation in Ansys which the behaviour of guide cannot portrays the real function of guide. The guide actually allows 5mm movement in X and Y direction. Unfortunately Ansys only allows the user to input the boundary condition without allowable movement. In this case, there are two setup for guide which is fix in X and Y direction and allow movement in X and Y direction without limitation of 5mm. The reading of results are taken in between of both conditions.

4.3.6. Modal Analysis with Self-Weight from Original Version

Table 4.1 shows result of modal analysis with self-weight follows the original condition. Minimum frequency of vibration in guide that allows X and Y axis is 2.39Hz. It occurs in vertical orientation of pipe. However the minimum frequency of vibration in guide that fix X and Y direction is 2.86Hz. Instead of occurred at vertical piping, it occurs at different location which happened because of pipe support distribution. Distance between pipe supports in critical (red colour) area of mode 1 is 6m. The range of maximum total deformation is 25.78mm to 28.76mm.

TABLE 4.1: Result of modal analysis with self-weight from original condition

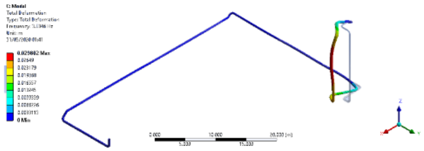
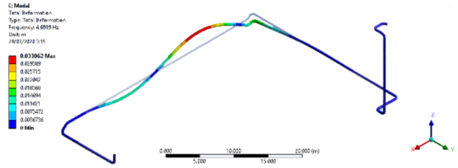
Guide (Allow: X and Y direction)				Guide (Constraint: X and Y direction)		
Mode	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape
1	2.39	19.00		2.86	20.16	
2	2.86	20.16		4.33	22.21	

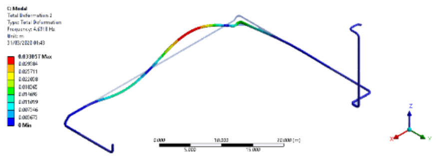
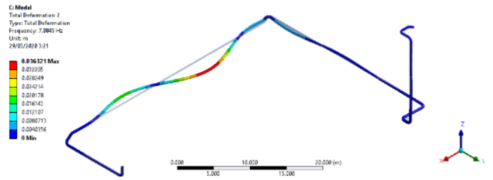
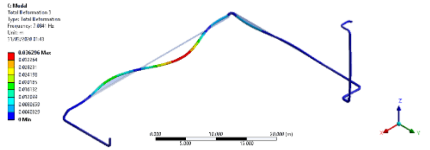
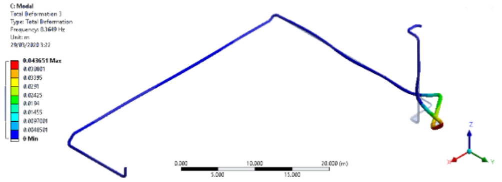
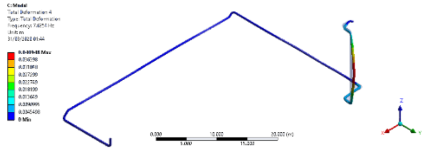
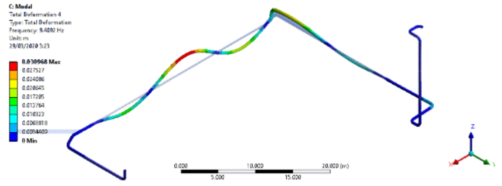
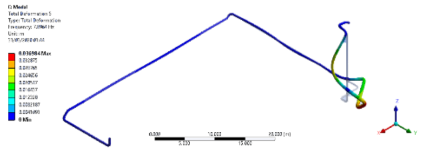
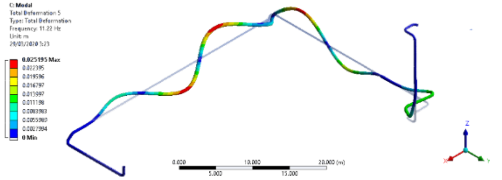
3	4.33	22.20		5.67	28.76	
4	4.62	25.01		5.74	18.68	
5	5.20	25.78		6.85	16.27	
6	5.75	18.71		7.24	17.14	

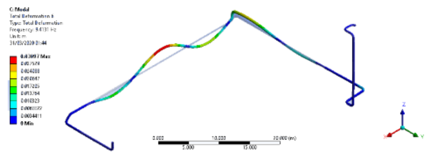
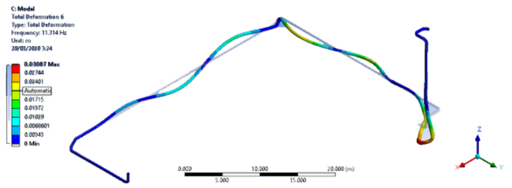
4.3.7. Modal Analysis without Self-Weight from Original Version

Table 4.2 shows result of modal analysis without self-weight from original version. It shows that range of maximum deformation are between 37.00mm to 43.65mm. These range are taken after considering the location of mode shape of the pipe that experience highest total deformation. It occurs at the bottom of the vertical pipe. The lowest natural frequency of vibration for guide that allows X and Y direction is 3.83Hz. It occurs at vertical pipe orientation. While lowest natural frequency of vibration for guide that constraint in X and Y direction is 4.69Hz. It does not occur at vertical pipe orientation but occurs at location of pipe that have large distance of pipe support distribution. The distance of pipe support distribution is 6m.

TABLE 4.2: Result of modal analysis without self-weight from original condition

Guide (Allow: X and Y direction)				Guide (Constraint: X and Y direction)		
Mode	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape
1	3.83	29.80		4.69	33.06	

2	4.19	33.06		7.08	36.32	
3	7.08	36.30		8.36	43.65	
4	7.63	40.95		9.41	31.00	
5	7.90	37.00		11.22	25.20	

6	9.43	30.97		11.31	30.90	

4.3.8 Comparison between Result of Modal Analysis with self-weight and without Self-Weight from Original Version

Figure 4.8 shows the comparison between results of total deformation with self-weight and without self-weight from original version. Pipe without self-weight experienced higher total deformation than pipe with self-weight. This is because there is lower load in pipe without self-weight than pipe with self-weight. The concept is object with higher load need more energy to make it move. Average range percentage difference of total deformation between pipe with and without self-weight is 60%-63%.

(a)

(b)

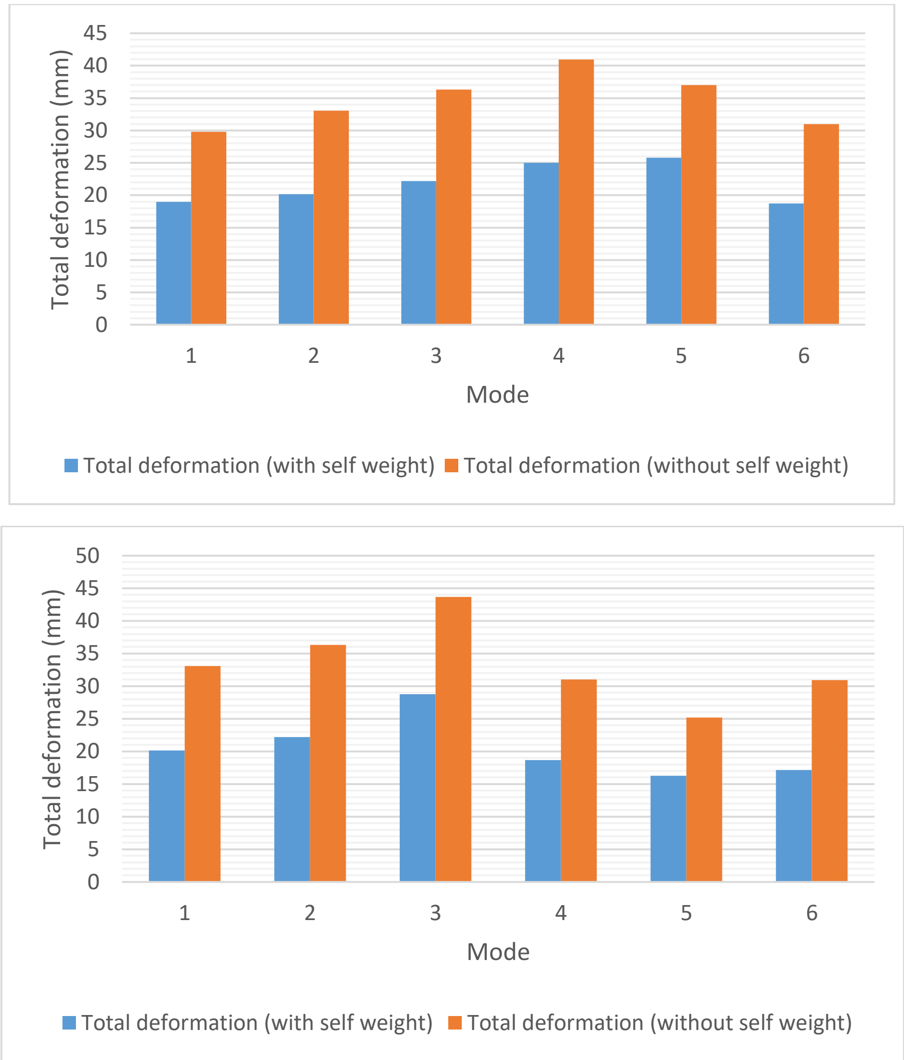
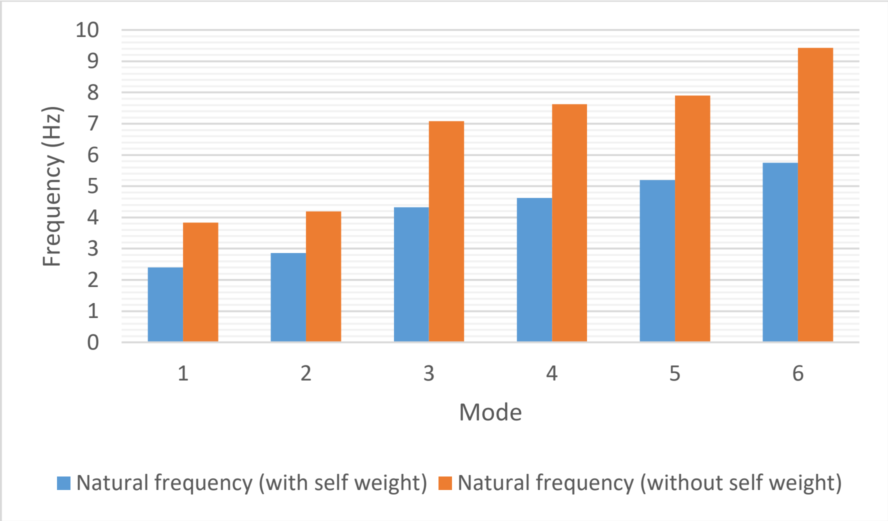


FIGURE 4.8: Graph of Comparison between results of total deformation with self-weight and without self-weight from original version. (a) Allows in X and Y direction; (b) constraint in X and Y direction.

Figure 4.9 shows the comparison between results of natural frequency with self-weight and without self-weight from original version. Pipe with self-weight has lower natural frequency than pipe without self-weight. Percentage different of natural frequency between pipe with self-weight and without self-weight is 60%. Frequency decrease when mass increase [30].

(a)



(b)

FIGURE 4.9: Graph of Comparison between results of natural frequency with self-weight and without self-weight from original version. (a) Allows in X and Y direction; (b) constraint in X and Y direction.

4.4 Result Validation

Results that have been obtained from original model has been validated to ensure the results are correct and model can be used for further study. Results are being compared with solid reference taken from reliable source.

4.4.1 Validation of Static Structural Analysis Model

Results that have been obtained for total deformation is 49.42mm. There is source from RTS PETRONAS report mentioned that total deformation at that particular area is 44mm [31]. There is difference of 5.42mm with the percentage different of 11%. The small difference of results obtained between this model and RTS PETRONAS report decided to use this model for further study.

4.4.2 Validation of Modal Analysis Model

Beside computational way in finding natural frequency there is also numerical method that can be used to find natural frequency. The formula used is shows below [1].

$$f\ pipe\ span = \frac{\lambda}{2\pi}\sqrt{\frac{gEI}{\mu l^4}} \qquad (2)$$

λ = Frequency factor,

g = Gravitation constant,

E = Modulus of elasticity,

I = Moment of inertia,

μ = Weight per unit length of beam (including fluid and insulation)

l = Span length,

TABLE 4.3: Input value for numerical analysis

Mode	λ	2π	g	E	I	μ	l	Frequency (Hz)
1	20.6	6.29	9.81	1.95E+11	1.97E-04	2203.18	24	**2.35**

The result of natural frequency shows from Table 4.3 for mode 1 is 2.35Hz while. Results of mode 1 that have been obtained from the model is 2.39Hz. The difference between result obtained from this model and numerical method is 1.7%. This model is considered reliable to be used for further study.

4.5 Model for Variation 1

Based on previous result, hotspot area being detected considered the most critical part of the pipe. Additional number of supports are being introduced to solve the issue. Figure 4.10 shows the number, location and pipe of supports that been introduced for this model. 1 saddle plate pipe supports which 1m apart from elbow and 1 trunnion which set 1m apart from existing trunnion is been set in this model.

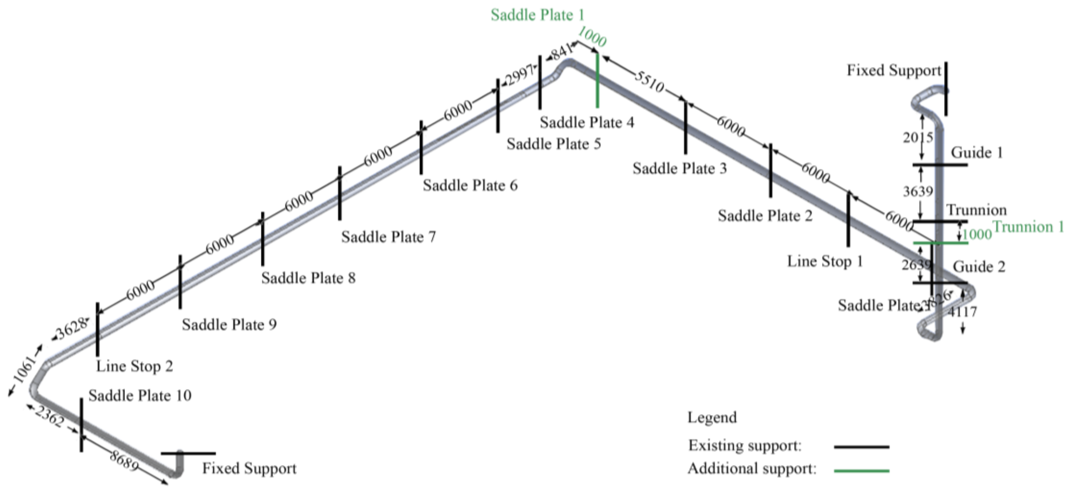


FIGURE 4.10: Pipe support distribution for variation 1

4.5.1 Static Structural with Self-Weight from Variation 1

Based on Figure 4.11, points that exceed allowable stress limit are point 4, point 6 and point 7. Stress value produced from point 4 is 156.20MPa, point 6 is 271.96MPa while point 7 produced 271.96MPa. Stress value at points 4 and 7 still exceed the allowable stress limit because of the set up in boundary condition. The increment of stress value at point 6 which before this below stress limit is because it is over constraint due to existence of additional Saddle Plate 1. The stress is high at this point is also because of the design of the pipe which has been explained in section 4.3.2.

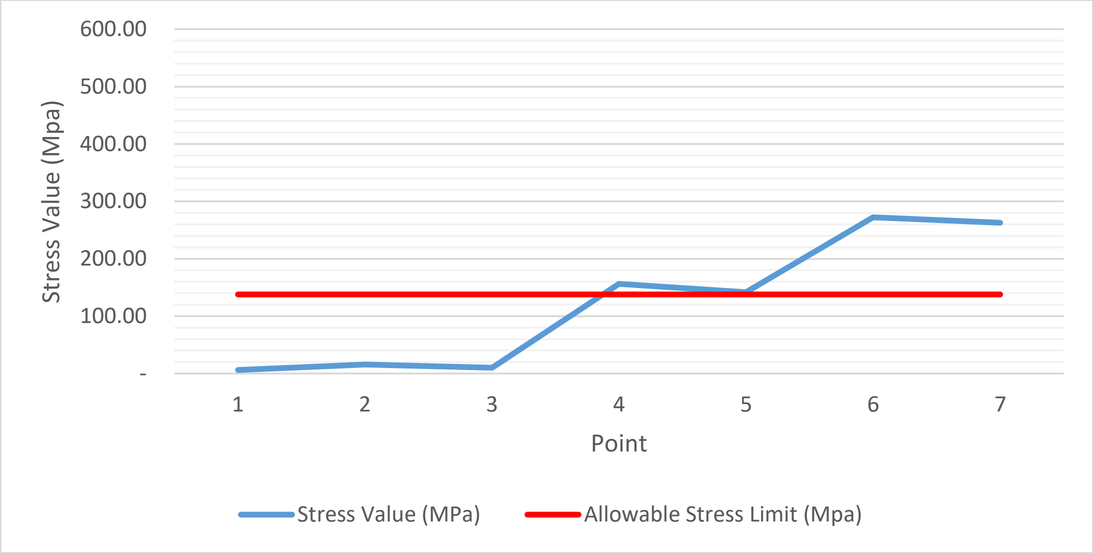


FIGURE 4.11: Graph of Stress value for variation 1 with self-weight

Figure 4.12 shows total deformation of the pipe for variation 1 with self-weight. The maximum total deformation is marked by red colour changed its location after saddle plate 1 is add in the pipe system. From original condition result, the previous red colour area has been recorded to have 49.42mm of total deformation. It is decreased to 28mm. It is been improved by 43.34%. However it cause spike of total deformation at new area. Total deformation at red circle area is 47.63mm. Overall, the maximum total deformation has been decreased by 3.6%

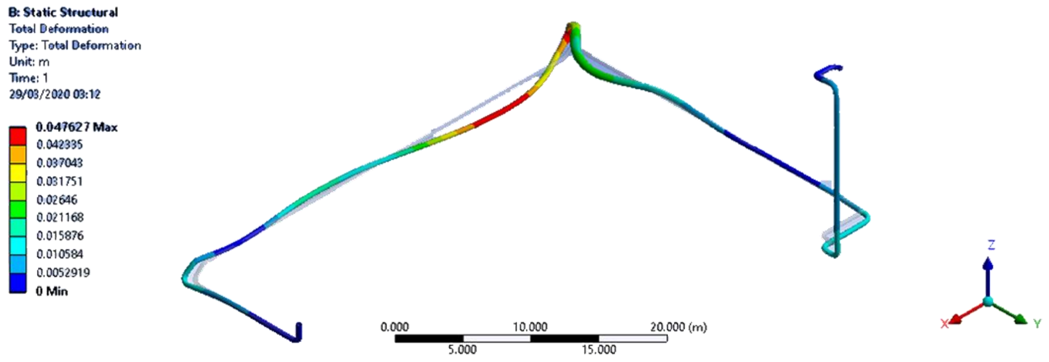


FIGURE 4.12: Total deformation of pipe for variation 1 with self-weight

4.5.2 Static Structural without Self-Weight from Variation 1

Figure 4.13 shows the result of stress value for original condition without self-weight. The result shows that the stress value at point 4, point 6 and point 7 exceed the allowable stress limit. The reason behind this has been discussed in section 4.5.1.

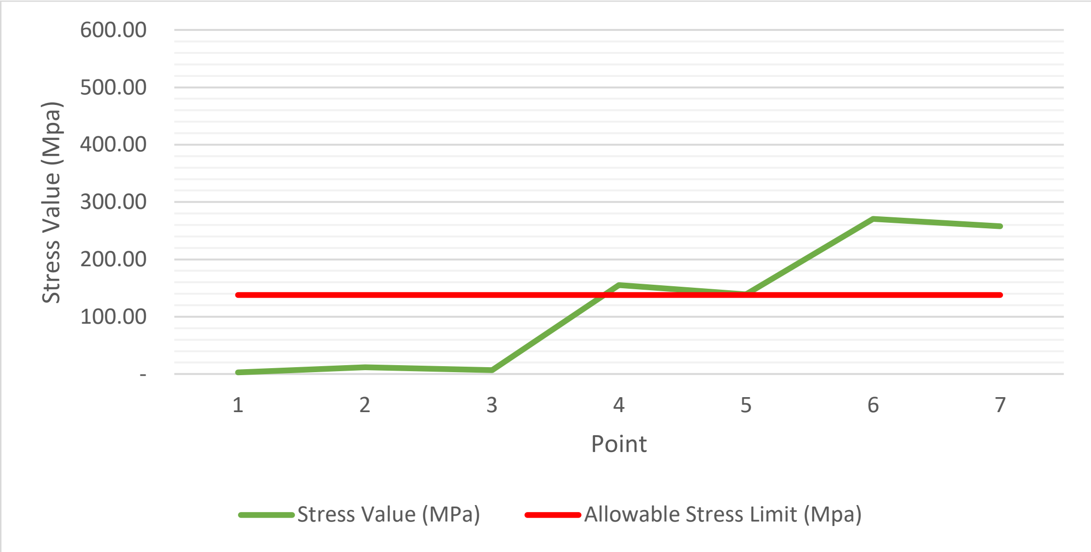


FIGURE 4.13: Graph of Stress value for variation 1 without self-weight

Figure 4.14 shows total deformation of pipe without self weight. The maximum value of total deformation is 52.23mm.

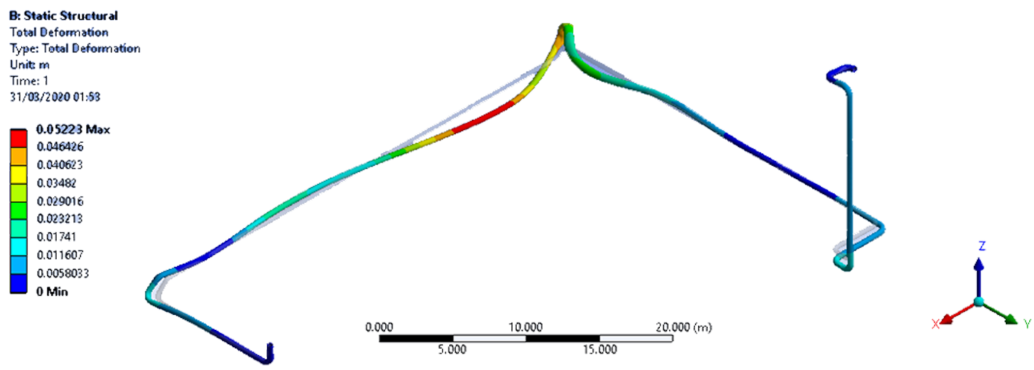


FIGURE 4.14 Total deformation of variation 1 pipe without self-weight

4.5.3 Comparison between Result of Static Structural with self-weight and without Self-Weight from Variation 1

Figure 4.15 shows the comparison between results of stress value with self-weight and without self-weight. There is only small difference between the results mentioned. However, stress value without self-weight is smaller than stress value with self-weight. The average percentage difference of stress value between these two conditions is 15.3%. In addition, difference of total deformation between both conditions is 4.6mm equivalent to 9.67%.

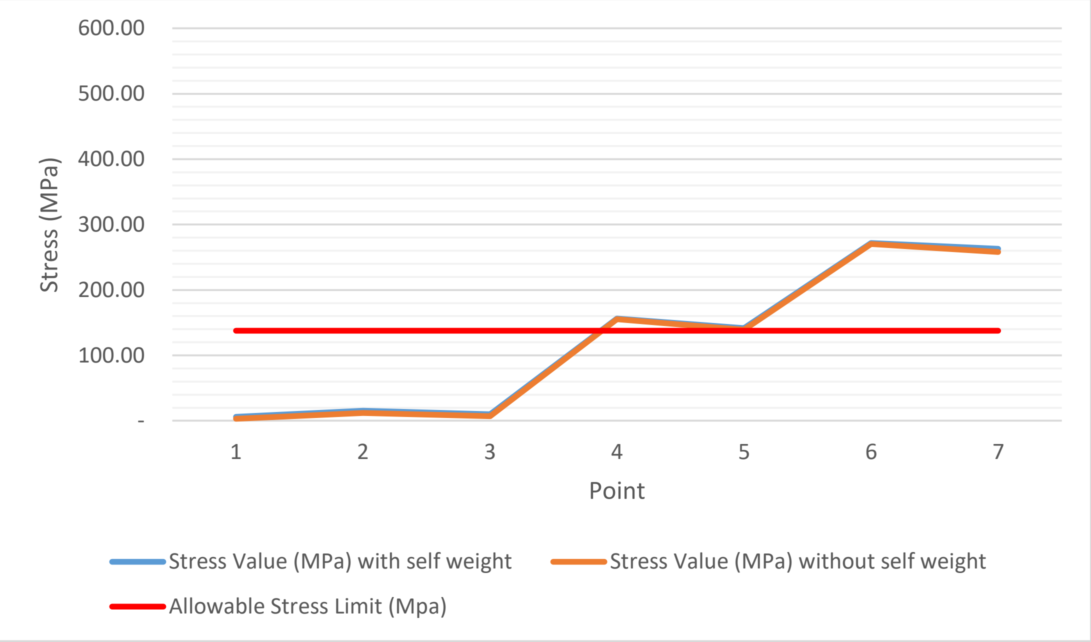
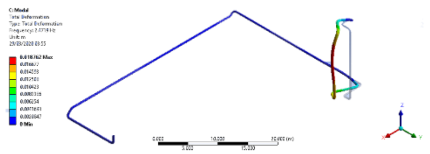
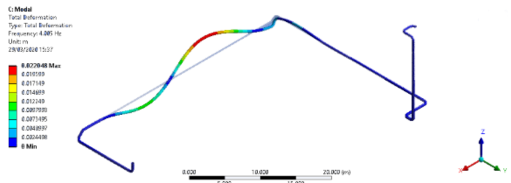
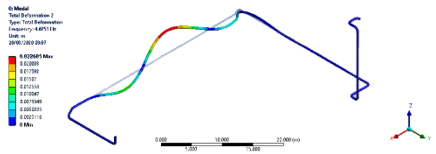
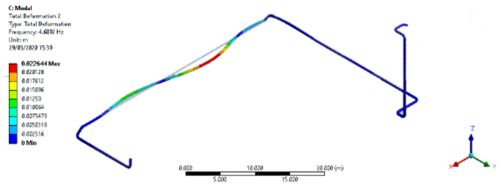
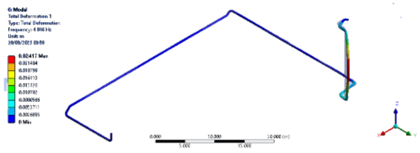
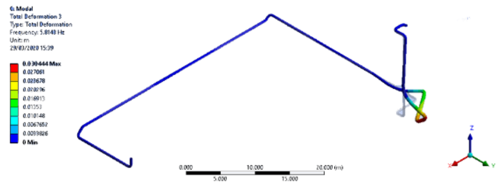


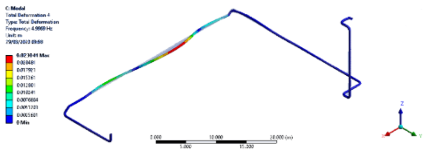
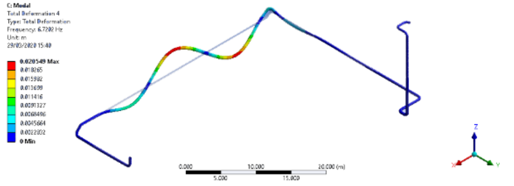
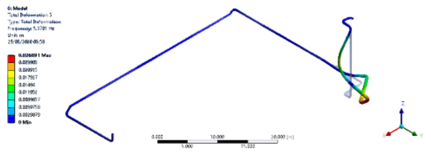
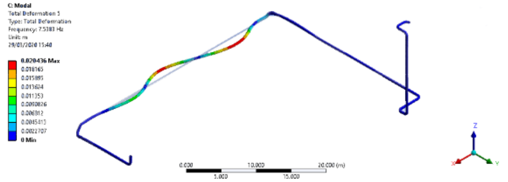
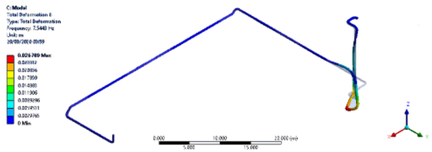
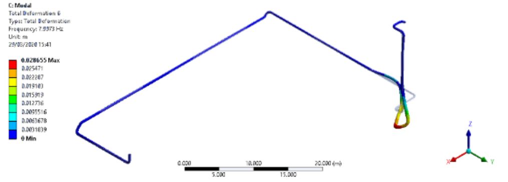
FIGURE 4.15: Graph of Comparison between results of stress value with self-weight and without self-weight from variation 1

4.5.4 Modal Analysis with Self-Weight from Variation 1

Table 4.4 shows result of modal analysis with self-weight follows the original condition. Minimum frequency of vibration in guide that allows X and Y axis is 2.47Hz. It occurs in vertical orientation of pipe. However the minimum frequency of vibration in guide that fix X and Y direction is 4.0Hz. Instead of occurred at vertical piping, it occurs at different location which happened because of pipe support distribution. Distance between pipe supports in critical (red colour) area of mode 1 is 6m. Range of maximum total deformation is in between 26.89mm to 30.44mm.

TABLE 4.4: Result of modal analysis with self-weight from variation 1 condition

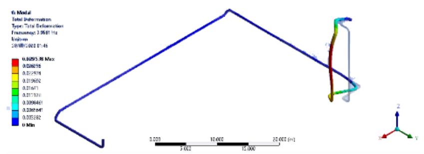
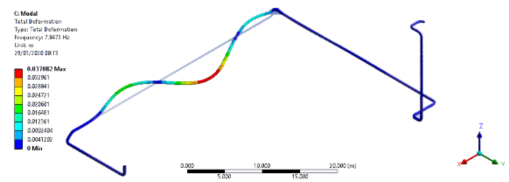
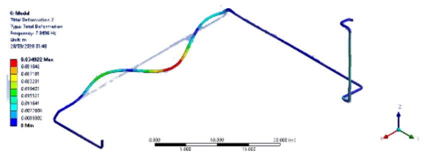
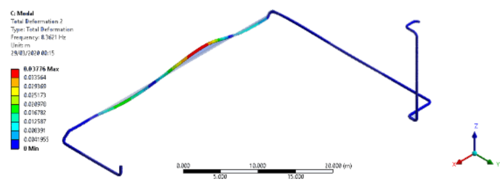
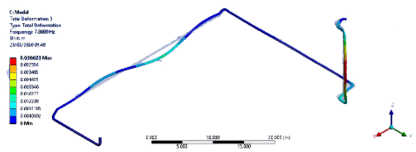
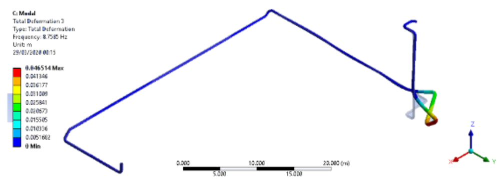
Guide (Allow: X and Y direction)				Guide (Constraint: X and Y direction)		
Mode	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape
1	2.47	18.76		4.00	22.05	
2	4.68	22.60		4.61	22.64	
3	4.82	24.17		5.81	30.44	

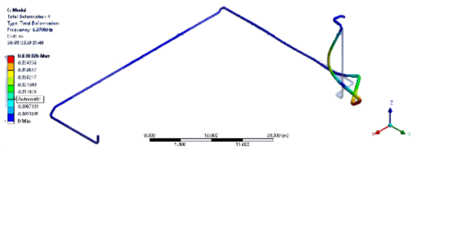
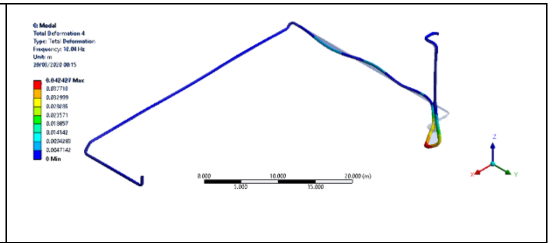
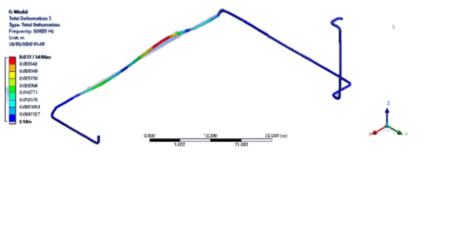
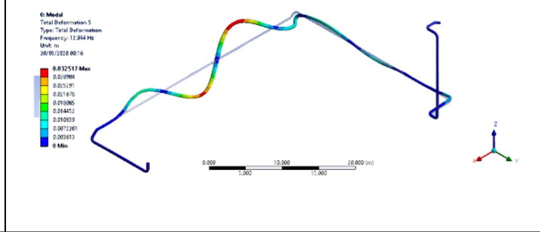
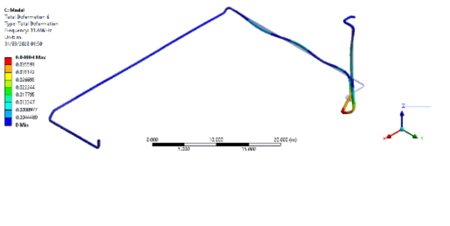
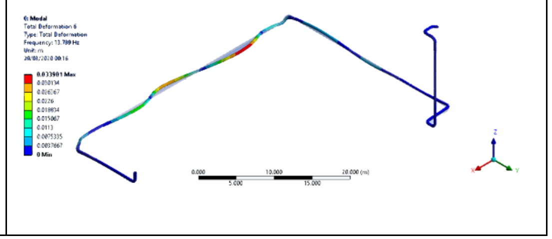
4	5.00	23.04		6.72	20.55	
5	5.37	26.89		7.54	20.44	
6	7.54	26.79		8.00	28.66	

4.5.5. Modal Analysis without Self-Weight from Variation 1

Table 4.5 shows result of modal analysis without self-weight from variation 1 version. It shows that range of maximum deformation are between 39.33mm to 46.51mm. It has been increase after trunnion is applied at that area. These range are taken after considering the location of mode shape of the pipe that experience highest total deformation. It occurs at the bottom of the vertical pipe. The lowest natural frequency of vibration for guide that allows X and Y direction is 4.00Hz. It occurs at vertical pipe orientation. While lowest natural frequency of vibration for guide that constraint in X and Y direction is 7.85Hz. It has been increased by 3.16. Saddle Plate 1 has effects the natural frequency at that area. It does not occur at vertical pipe orientation but occurs at location of pipe that have large distance of pipe support distribution. The distance of pipe support distribution is 6m.

TABLE 4.5: Result of modal analysis without self-weight from variation 1 condition

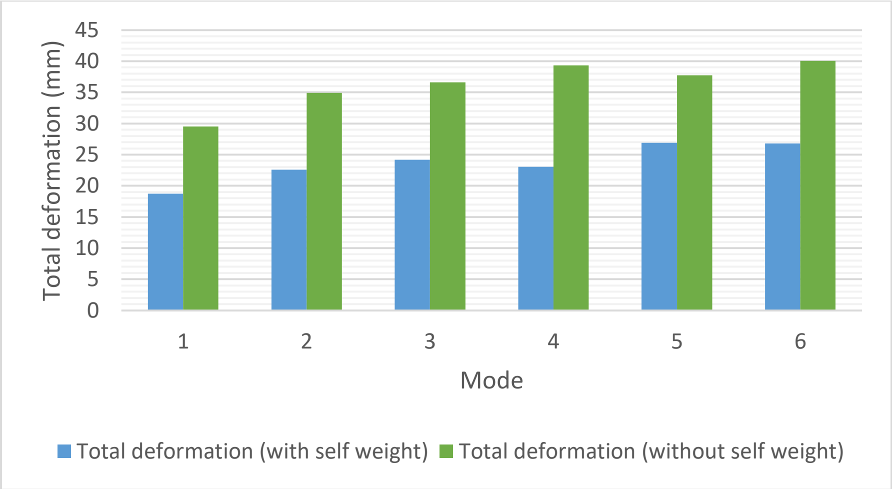
	Guide (Allow: X and Y direction)			Guide (Constraint: X and Y direction)		
Mode	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape
1	4.00	29.54		7.85	37.08	
2	7.85	34.92		8.36	37.76	
3	7.86	36.62		8.76	46.51	

4	8.27	39.33		12.04	42.43	
5	8.36	37.73		12.91	32.52	
6	11.45	40.04		13.74	33.90	

4.5.6 Comparison between Result of Modal Analysis with self-weight and without Self-Weight from Variation 1

Figure 4.16 shows the comparison between results of total deformation with self-weight and without self-weight from variation 1. Pipe without self-weight experienced higher total deformation than pipe with self-weight. This is because there is lower load in pipe without self-weight than pipe with self-weight. Discussion has been stated in section 4.3.7. Average range percentage difference of total deformation between pipe with and without self-weight is 54%-62%. Larger than original model which have range of 60%-63%.

(a)



(b)

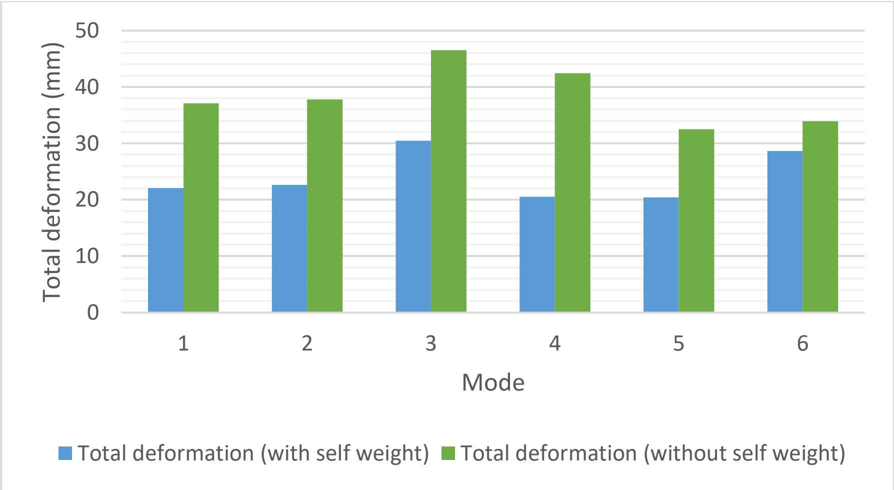


FIGURE 4.16: Graph of Comparison between results of total deformation with self-weight and without self-weight from variation 1. (a) Allows in X and Y direction; (b) constraint in X and Y direction.

Figure 4.17 shows the comparison between results of natural frequency with self-weight and without self-weight from original version. Pipe with self-weight has lower natural frequency than pipe without self-weight. The range of percentage different of natural frequency between pipe with self-weight and without self-weight is in between 61%-75%. Larger than original model which has percentage difference of 60%.

(a)

(b)

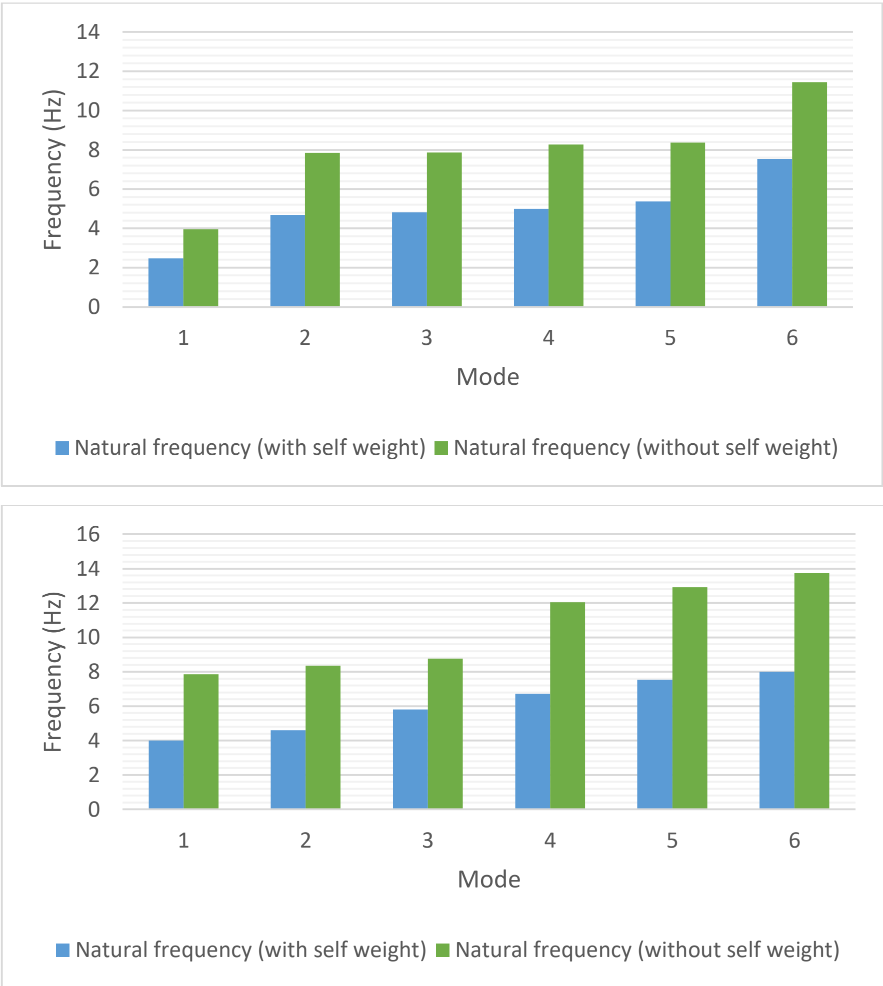


FIGURE 4.17: Graph of Comparison between results of natural frequency with self-weight and without self-weight from variation 1. (a) Allows in X and Y direction; (b) constraint in X and Y direction.

4.6 Model for Variation 2

Based on previous result, hotspot area being detected considered the most critical part of the pipe. Additional number of supports that have been introduced in variation 1 is kept to be there. Figure 4.18 shows the number, location and pipe of supports that been introduced for variation 2 model. 2 saddle plate pipe supports and 1 trunnion is been set in this model. Total additional pipe supports are 3 saddle plates and 2 trunnion. Saddle plate is been set 1m apart from saddle plate 6 and saddle plate 5. 1 Additional trunnion is been add 1m apart from previous addition trunnion support mentioned in variation 1.

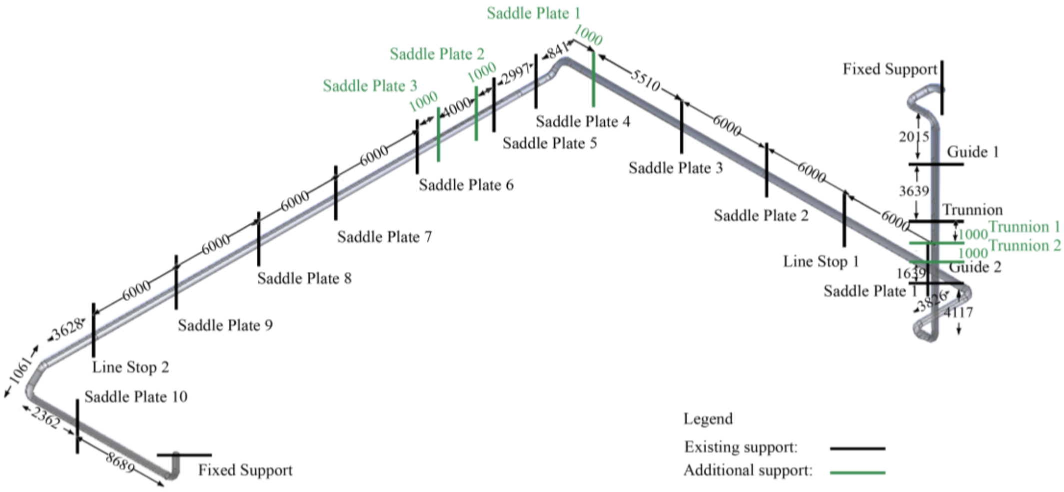


FIGURE 4.18: Pipe support distribution for variation 2

4.6.1 Static Structural with Self-Weight from Variation 2

Based on Figure 4.19, points that exceed allowable stress limit are point 1, point 5, point 6 and point 7. In variation 1, stress value for point 4 is more than allowable stress limit. However, in this variation of number of pipe supports, the stress value for point for decrease from 156.20MPa to 128.01MPa. Which is good improvement as it bellows the allowable stress limit. Stress value produced at point 1 is 227.12MPa, point 5 with 182.17MPa, point 6 with 275.19MPa while point 7 produced 212.3MPa. Point 7 improved by 60MPa. Stress value at points 4 and 7 still exceed the allowable stress limit because of the set up in boundary condition. Stress value at point 4 spikes up by 220.12MPa because of there are too many pipe trunnion support has been set at really close distance.

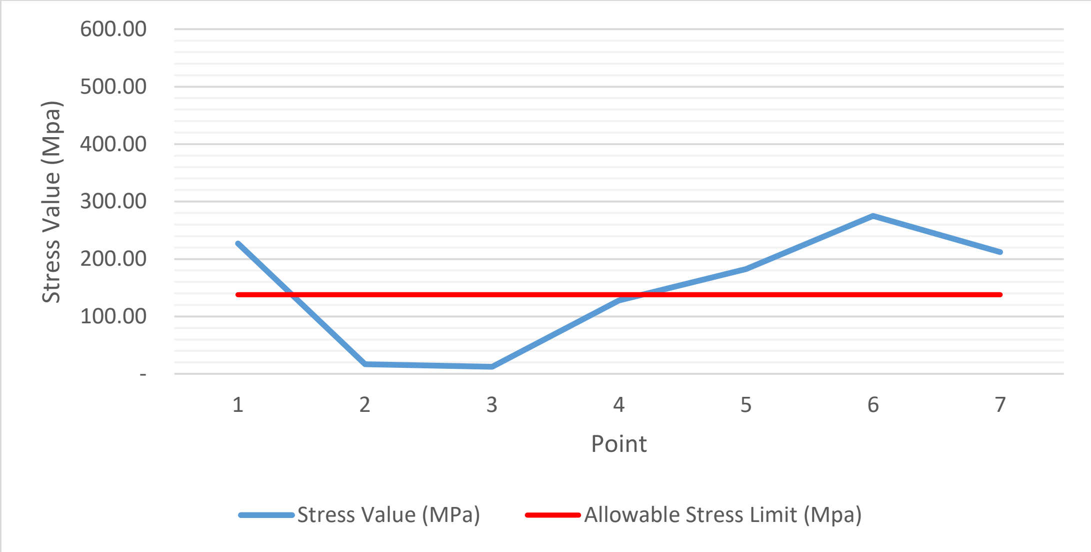


FIGURE 4.19: Graph of Stress value for variation 2 with self-weight

Figure 4.20 shows total deformation of the pipe for variation 2 with self-weight. The maximum total deformation is marked by red colour changed its location after saddle plate 1 is add in the pipe system. Value for maximum total deformation has improved from 47.63mm to 42.20mm. This is because of the present of additional Saddle Plate 2 and Saddle Plate 3. Maximum total deformation is improved by 11.4%.

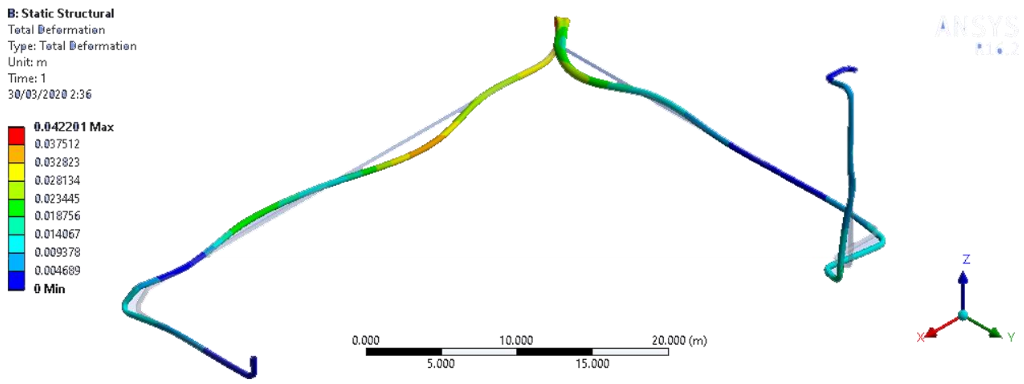


FIGURE 4.20: Total deformation of pipe for variation 2 with self-weight

4.6.2 Static Structural without Self-Weight from Variation 2

Figure 4.13 shows the result of stress value for original condition without self-weight. The result shows that the stress value at point 1, point 5, point 6 and point 7 exceed the allowable stress limit. The reason behind this has been discussed in section 4.5.1.

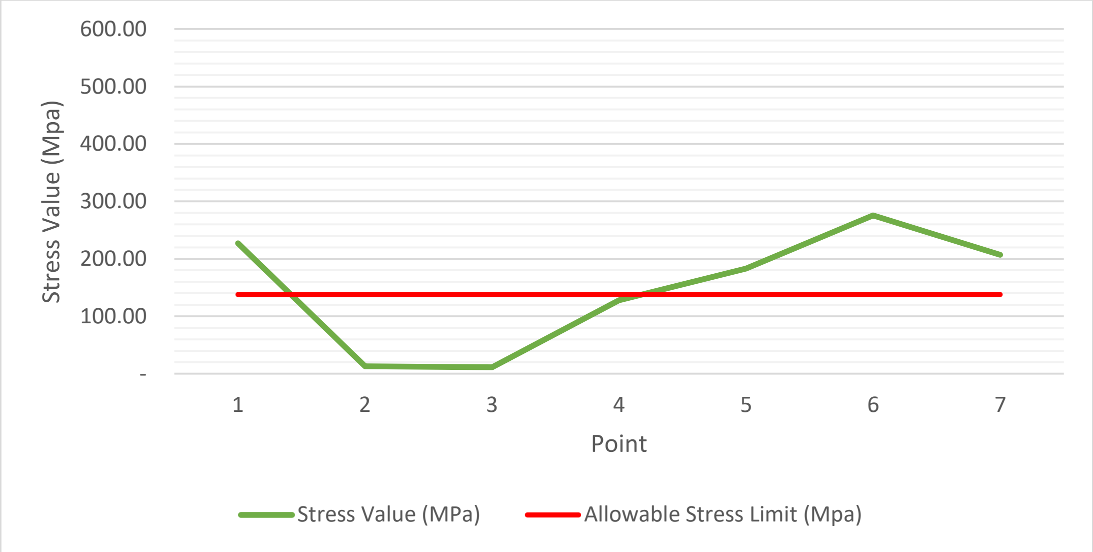


FIGURE 4.21: Graph of Stress value for variation 2 without self-weight

Figure 4.22 shows total deformation of pipe without self-weight. The maximum value of total deformation is improved from 52.23mm to 42.17mm. Two saddle plates that been set at the location for variation 2 improved the maximum value of total deformation as it support the pipe and reduce the gap between supports.

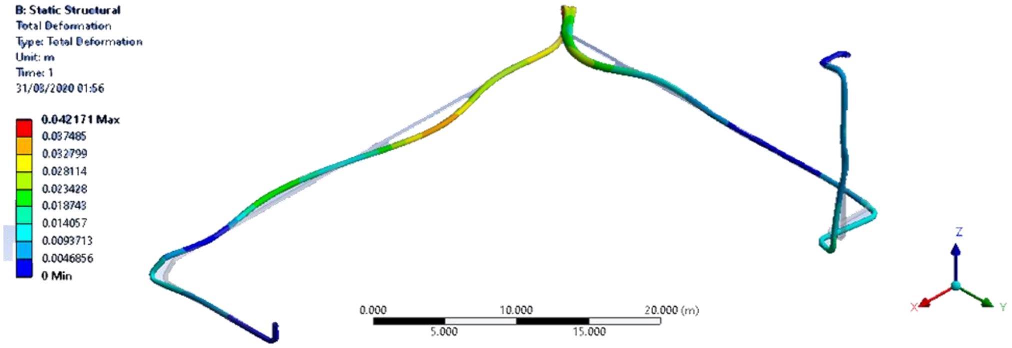


FIGURE 4.22 Total deformation of variation 2 pipe without self-weight

4.6.3 Comparison between Result of Static Structural with self-weight and without Self-Weight from Variation 2

Figure 4.23 shows the comparison between results of stress value with self-weight and without self-weight. There is only small difference between the results mentioned. However, stress value without self-weight is smaller than stress value with self-weight. The average percentage difference of stress value between these two conditions is 4.68% where else in variation 1, it has 15.3% of average percentage value. In addition, difference of maximum total deformation between both conditions is 0.03mm equivalent to 0.07%. The percentage difference become lower as there are total of 3 saddle plates support at the area.

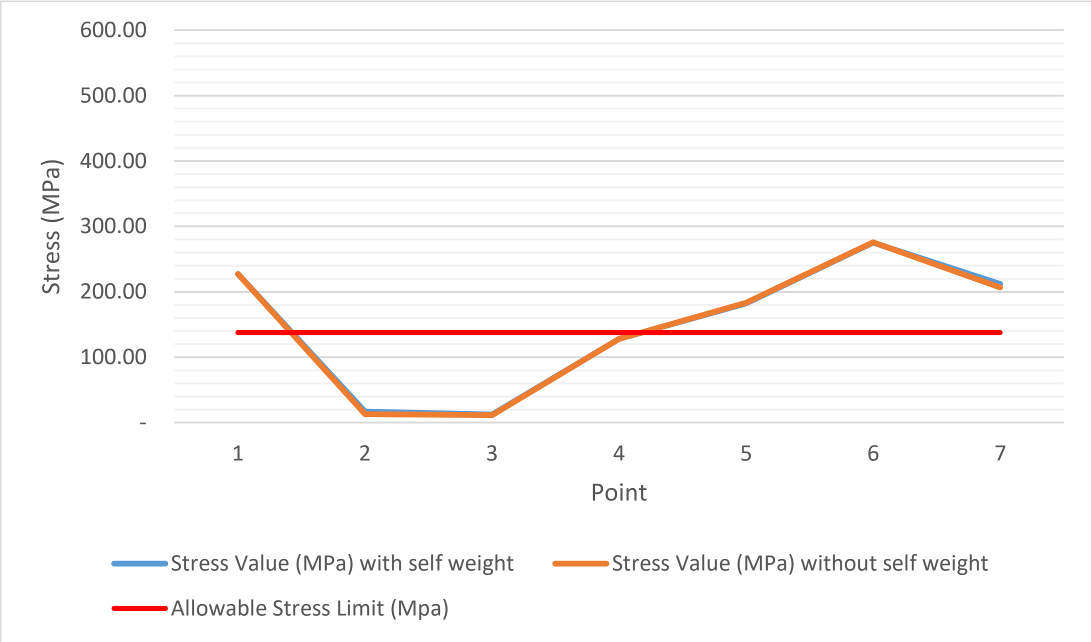
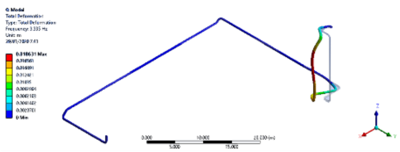
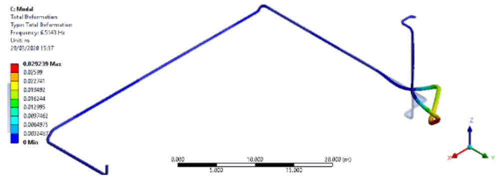
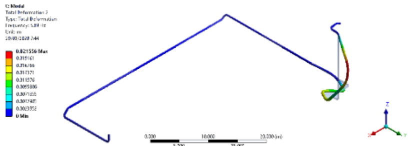
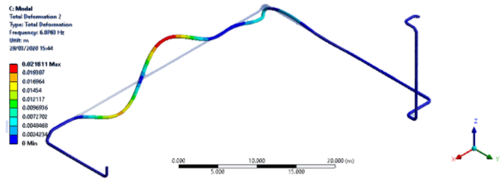
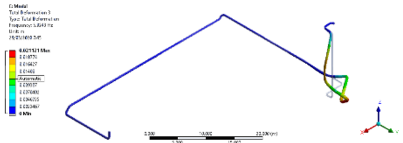
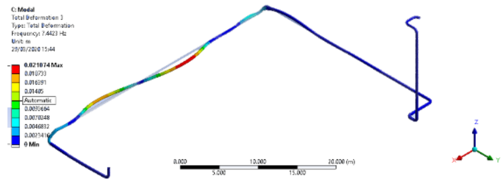


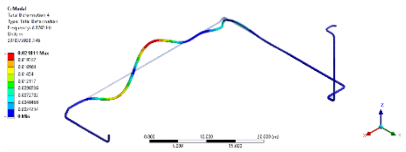
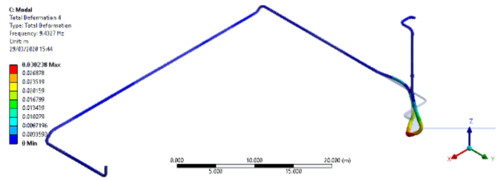
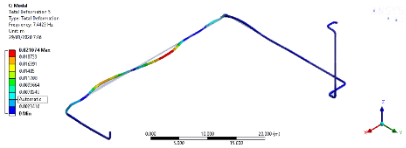
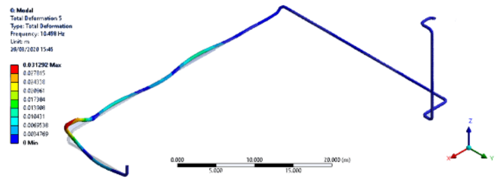
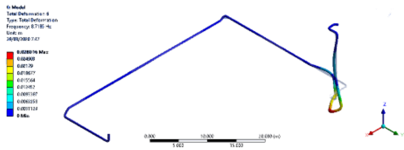
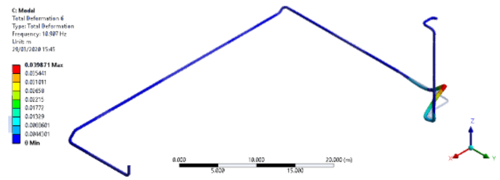
FIGURE 4.23: Graph of Comparison between results of stress value with self-weight and without self-weight from variation 2

4.6.4 Modal Analysis with Self-Weight from Variation 2

Table 4.6 shows result of modal analysis with self-weight follows the variation 2 boundary condition. Minimum frequency of vibration in guide that allows X and Y axis is 2.47Hz. It occurs in vertical orientation of pipe. However the minimum frequency of vibration in guide that fix X and Y direction is 4.0Hz. Instead of occurred at vertical piping, it occurs at different location which happened because of pipe support distribution. Distance between pipe supports in critical (red colour) area of mode 1 is 6m. Range of maximum total deformation is in between 26.89mm to 30.44mm.

TABLE 4.6: Result of modal analysis with self-weight from variation 2 condition

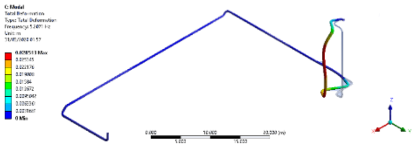
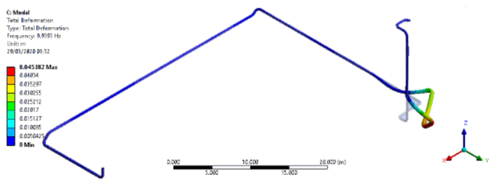
	Guide (Allow: X and Y direction)			Guide (Constraint: X and Y direction)		
Mode	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape
1	3.34	18.63		6.51	29.24	
2	5.69	21.56		6.88	21.81	
3	5.92	21.12		7.44	21.07	

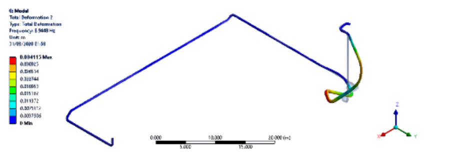
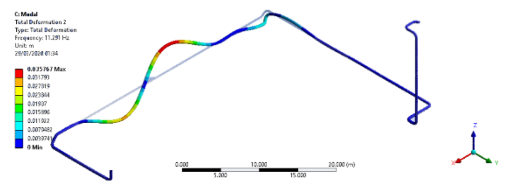
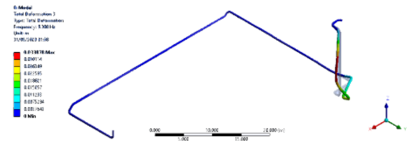
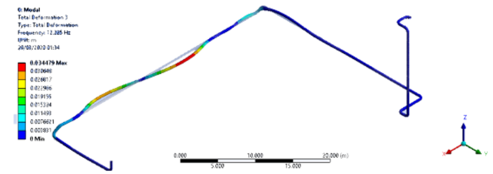
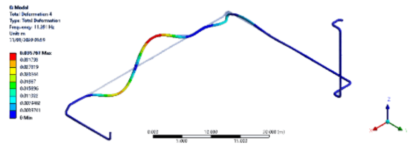
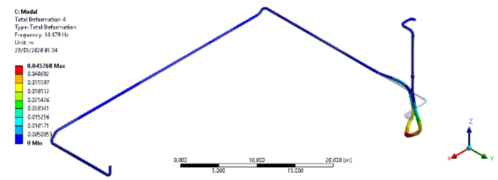
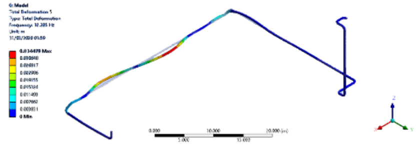
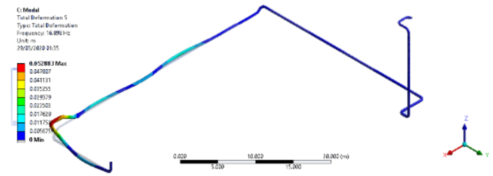
4	6.88	21.81		9.44	30.24	
5	7.44	21.07		10.46	31.29	
6	8.72	28.02		10.91	38.71	

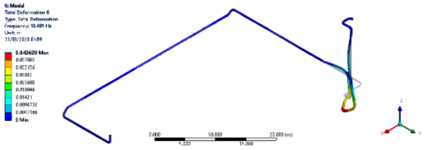
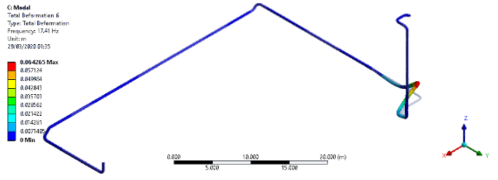
4.6.5. Modal Analysis without Self-Weight from Variation 2

Table 4.7 shows result of modal analysis without self-weight from variation 2 version. It shows that value of maximum deformation is 64.27mm. It increase by 17.76mm equivalent to 27.6% compared to variation 1. This occurs at the area of existing saddle plate 1 support. The lowest natural frequency of vibration for guide that allows X and Y direction is 5.21Hz. Improved by 1.21Hz equivalent to 23.22% compared to variation 1. It occurs at vertical pipe orientation. While lowest natural frequency of vibration for guide that constraint in X and Y direction is 9.94. It has been improved by 2.09Hz equivalent to 21.03% compared to variation 1.

TABLE 4.7: Result of modal analysis without self-weight from variation 2 condition

Guide (Allow: X and Y direction)				Guide (Constraint: X and Y direction)		
Mode	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape
1	5.21	28.51		9.94	45.38	

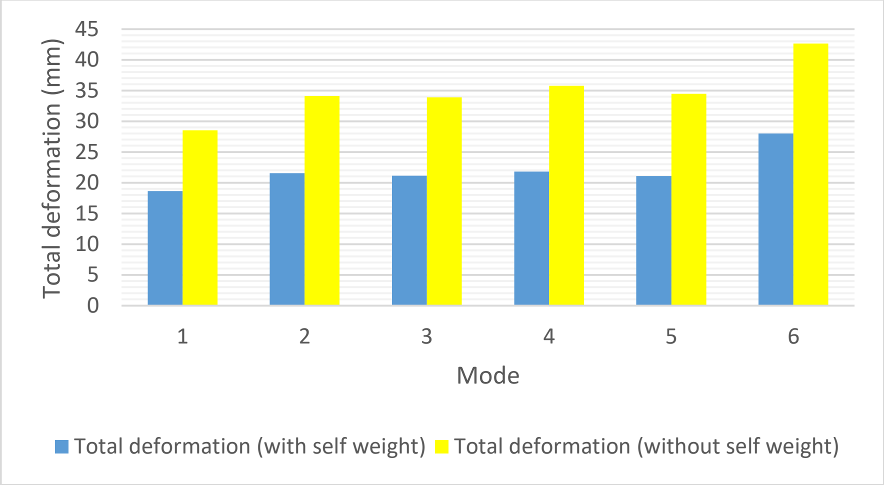
2	8.94	34.11		11.29	35.77	
3	9.31	33.88		12.24	34.48	
4	11.29	35.77		14.48	45.77	
5	12.23	34.48		16.9	52.88	

6	13.49	42.63		17.41	64.27	

4.6.6 Comparison between Result of Modal Analysis with self-weight and without Self-Weight from Variation 2

Figure 4.24 shows the comparison between results of total deformation with self-weight and without self-weight from variation 1. Pipe without self-weight experienced higher total deformation than pipe with self-weight. This is because there is lower load in pipe without self-weight than pipe with self-weight. Discussion has been stated in section 4.3.7. Average range percentage difference of total deformation between pipe with and without self-weight is 59%-62% while 54%-62% in variation 1 and 60%-63% in original model.

(a)



(b)

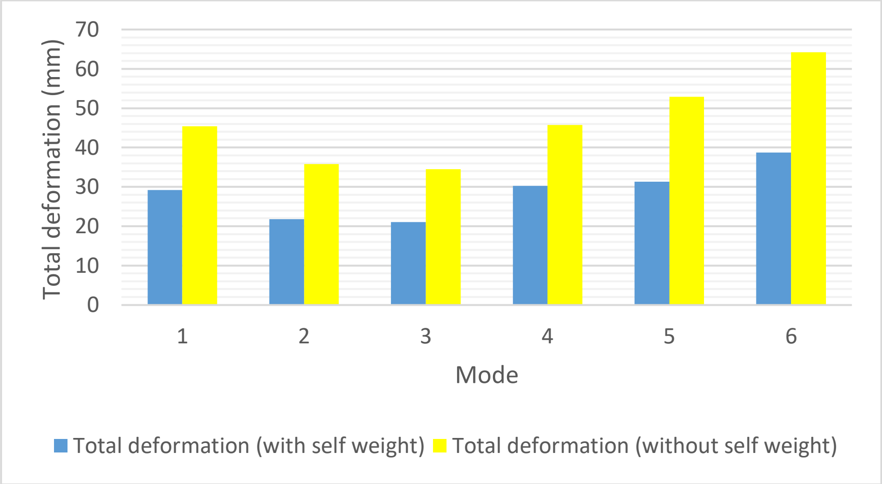


FIGURE 4.24: Graph of Comparison between results of total deformation with self-weight and without self-weight from variation 2. (a) Allows in X and Y direction; (b) constraint in X and Y direction.

Figure 4.25 shows the comparison between results of natural frequency with self-weight and without self-weight from original version. Pipe with self-weight has lower natural frequency than pipe without self-weight. The range of percentage different of natural frequency between pipe with self-weight and without self-weight is in between 59%-60%. It is lower when compare to the range from variation 1 version. While 61%-75% for variation 1 and 60% for original model.

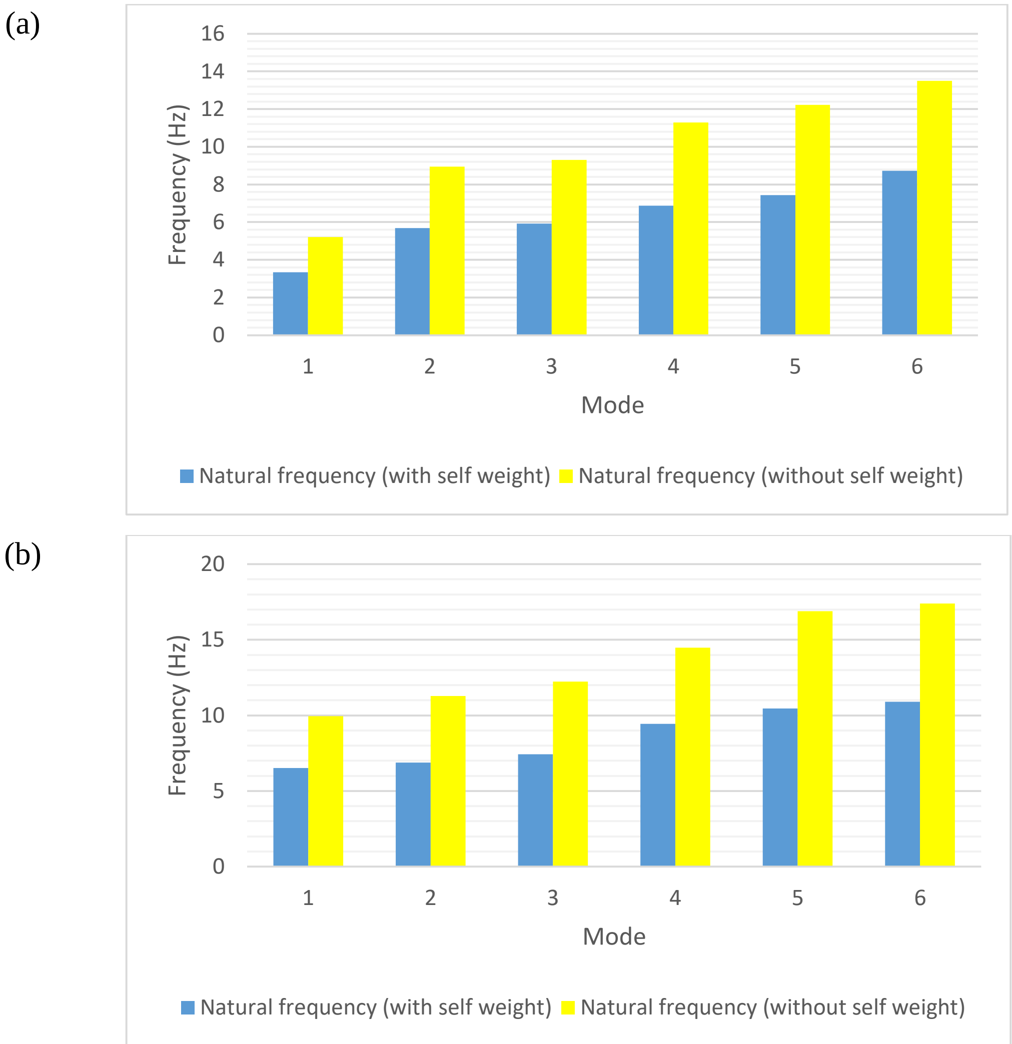


FIGURE 4.25: Graph of Comparison between results of natural frequency with self-weight and without self-weight from variation 2. . (a) Allows in X and Y direction; (b) constraint in X and Y direction.

4.7 Model for Variation 3

Based on previous result, hotspot area being detected by considering the most critical part of the pipe. Additional number of supports that have been introduced in variation 2 is kept to be there. Figure 4.26 shows the number, location and pipe of supports that been introduced for variation 3 model. Additional of 6 saddle plates and 1 trunnion is set to be in this model. Total additional pipe supports are 11 saddle plates and 3 trunnion. The location of additional pipe support is shown in Figure 4.26.

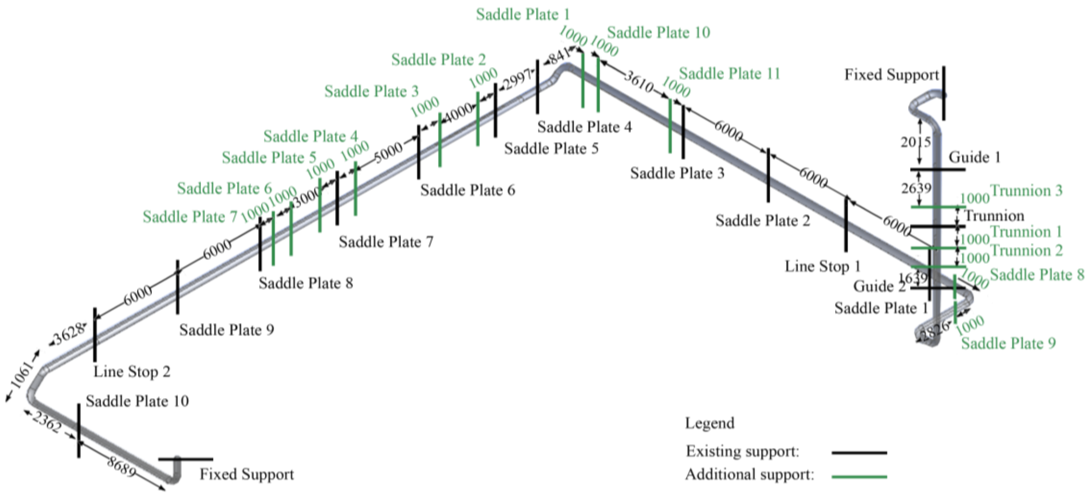


FIGURE 4.26: Pipe support distribution for variation 3

4.7.1 Static Structural with Self-Weight from Variation 3

Based on Figure 4.27, points that exceed allowable stress limit are point 1, point 3, point 5, point 6 and point 7. In variation 2, stress value for point 3 is below allowable stress limit. However, in this variation of number of pipe supports, the stress value for point 3 increase from 12.35MPa to 293.04MPa. After comparing with variation 2, stress value produced at point 1 is decrease from 227.12MPa to 196,01MPa, point 5 increase from 182.17MPa to 226.85MPa, point 6 from 275.19MPa to 527.76MPa while point 7 produced 232.33MPa increase by 20.03MPa compared to variation 2. Stress value at points 4 and 7 still exceed the allowable stress limit because of the set up in boundary condition. Stress value at point 3 suddenly exceed allowable stress because of it is over constraint because of the additional of Saddle Plate 9.

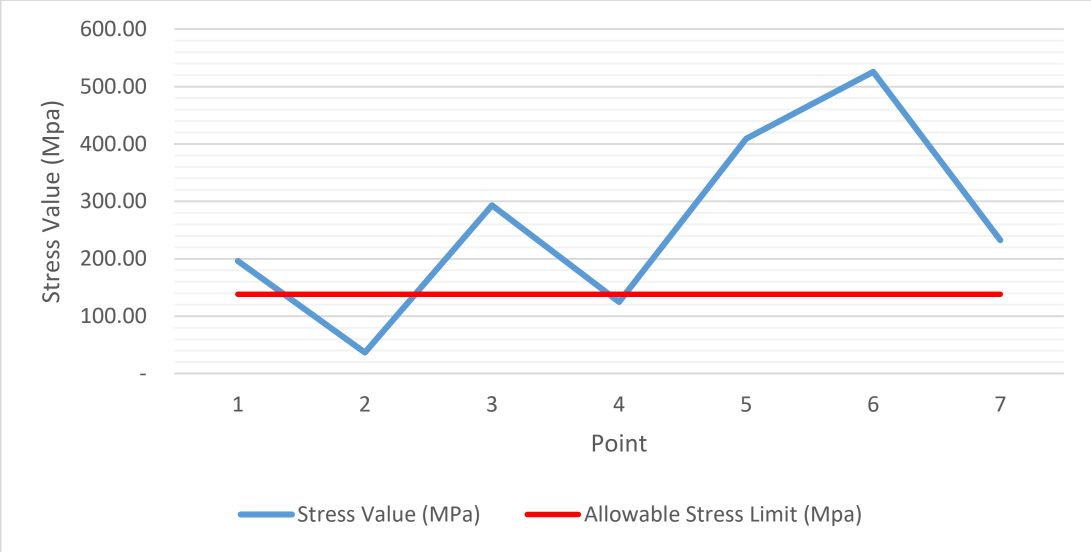


FIGURE 4.27: Graph of Stress value for variation 3 with self-weight

Figure 4.28 shows total deformation of the pipe for variation 3 with self-weight. The maximum total deformation is marked by red colour changed its location after saddle plate 1 is add in the pipe system. Value for maximum total deformation has improved from 42.20mm to 35.67mm after comparing with previous variation. This is because of the additional of the saddle plate. Maximum total deformation is improved by 15.47%.

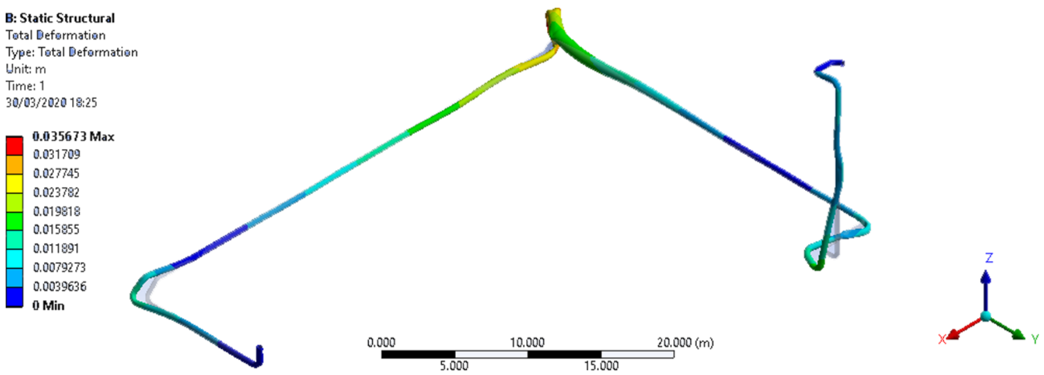


FIGURE 4.28: Total deformation of pipe for variation 3 with self-weight

4.7.2 Static Structural without Self-Weight from Variation 3

Figure 4.29 shows the result of stress value for original condition without self-weight. The result shows that the stress value at point 1, point3, point 5, point 6 and point 7 exceed the allowable stress limit. The reason behind this has been discussed in section 4.7.1.

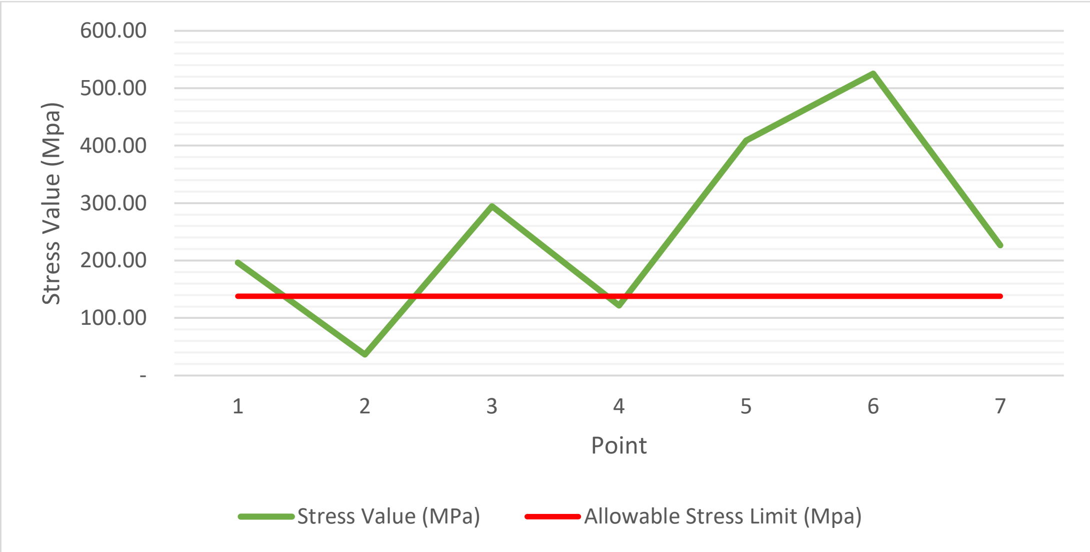


FIGURE 4.29: Graph of Stress value for variation 3 without self-weight

Figure 4.30 shows total deformation of pipe without self-weight. The maximum value of total deformation is improved from 42.17mm for variation 2 to 35.67mm. The additional of number of supports improved the maximum value of total deformation as it support the pipe and reduce the gap between supports.

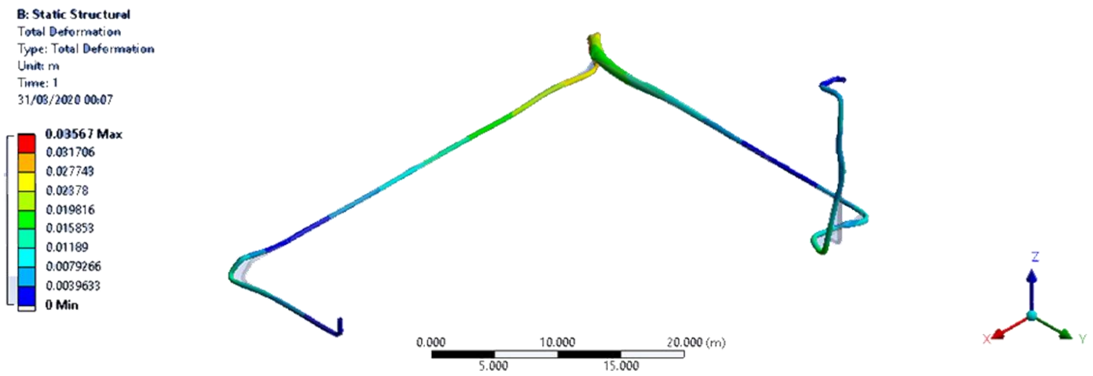


FIGURE 4.30: Total deformation of variation 3 pipe without self-weight

4.7.3 Comparison between Result of Static Structural with self-weight and without Self-Weight from Variation 3

Figure 4.23 shows the comparison between results of stress value with self-weight and without self-weight. There is only small difference between the results mentioned. However, stress value without self-weight is smaller than stress value with self-weight. The average percentage difference of stress value between these two conditions is 0.65%. In variation 2 it has 4.68% average percentage difference where else in variation 1, it has 15.3% of average percentage value. In addition, difference of maximum total deformation between both conditions is 0mm equivalent to 0%. High amount of support number make system become more rigid.

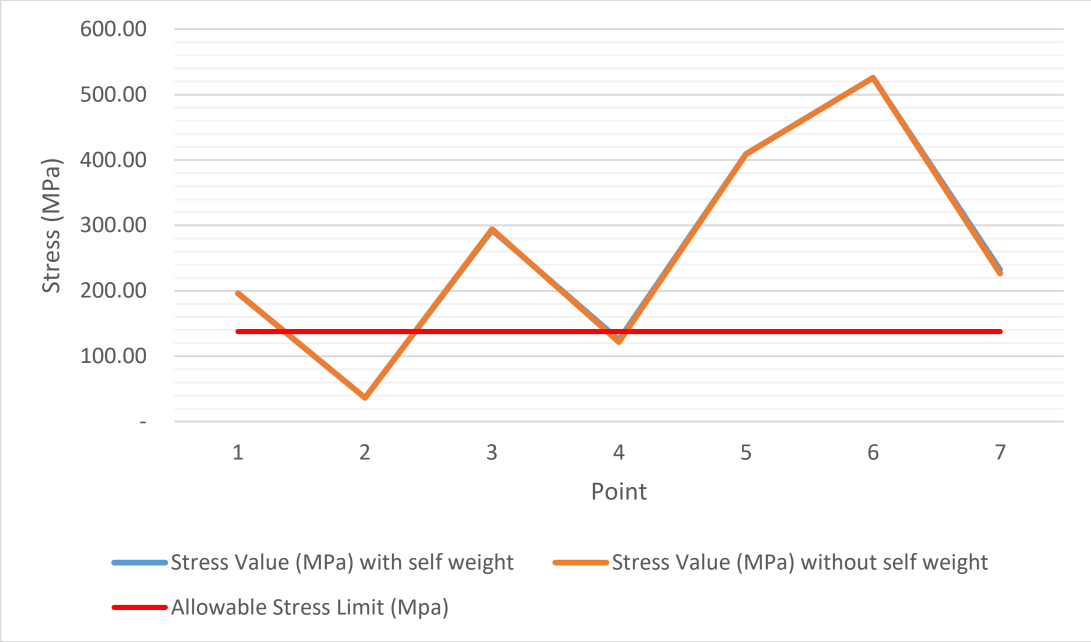
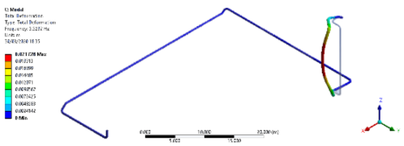
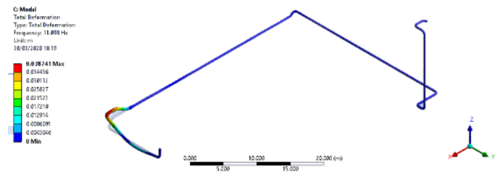
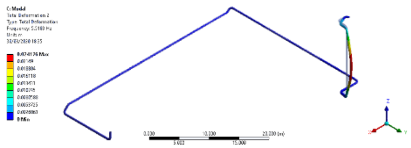
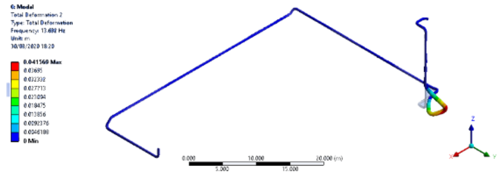
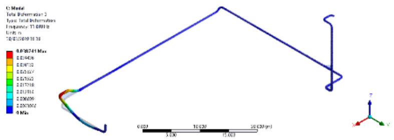
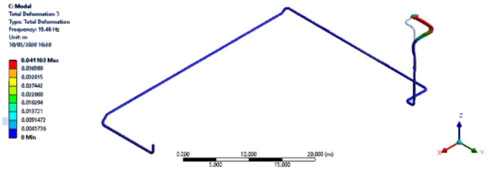


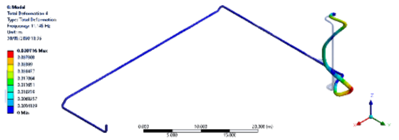
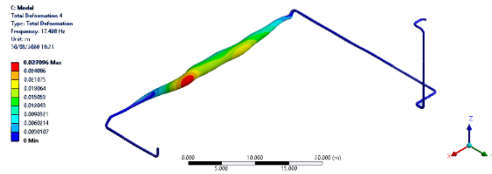
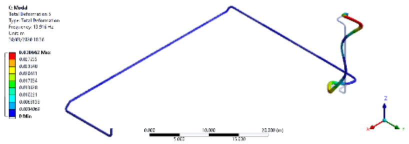
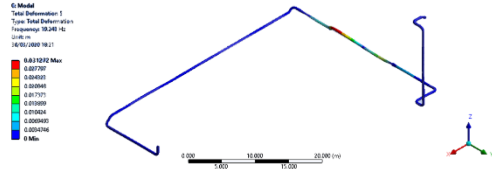
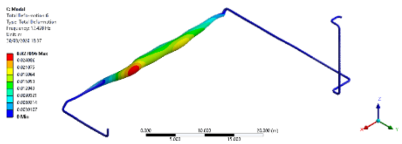
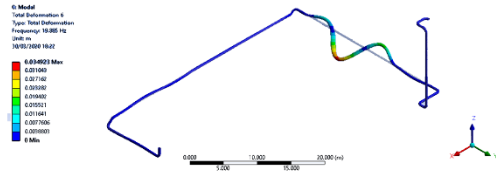
FIGURE 4.31: Graph of Comparison between results of stress value with self-weight and without self-weight from variation 3

4.7.4 Modal Analysis with Self-Weight from Variation 3

Table 4.8 shows result of modal analysis with self-weight follows the variation 3 boundary condition. Minimum frequency of vibration in guide that allows X and Y axis is 3.33Hz. It occurs in vertical orientation of pipe. However the minimum frequency of vibration in guide that fix X and Y direction is 11.0Hz. Instead of occurred at vertical piping, it occurs at different location which happened because of pipe support distribution. Natural frequency from mode 3 to mode 6 for guide that constraint X and Y axis is within safe operating limit which is must be more than 15Hz. Range of maximum total deformation is in between 30.72mm to 41.57mm.

TABLE 4.8: Result of modal analysis with self-weight from variation 3 condition

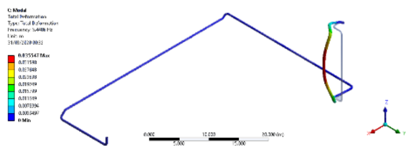
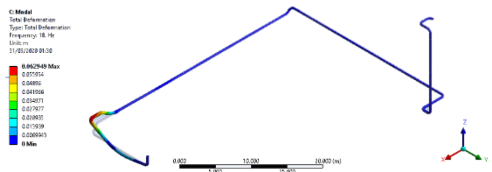
Mode	Guide (Allow: X and Y direction)			Guide (Constraint: X and Y direction)		
Mode	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape
1	3.33	21.73		11.10	38.74	
2	5.52	24.18		13.63	41.57	
3	11.10	38.74		15.46	41.16	

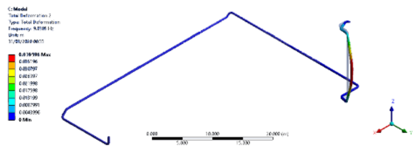
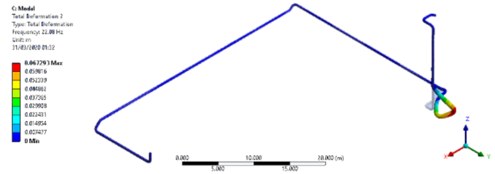
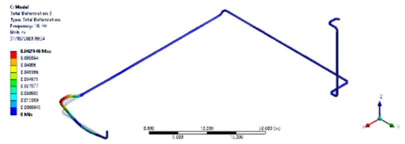
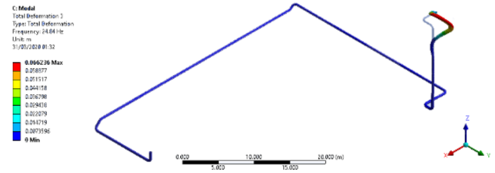
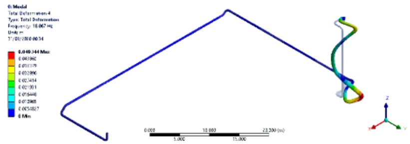
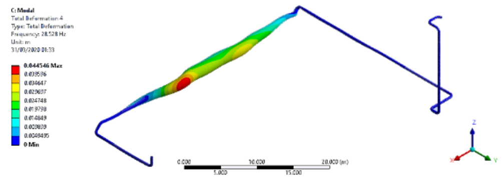
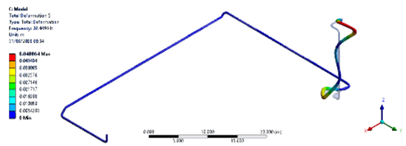
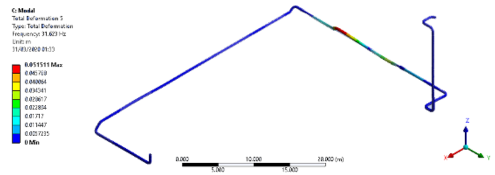
4	11.15	30.72		17.42	27.10	
5	13.92	30.66		19.24	31.27	
6	17.43	27.10		19.90	34.92	

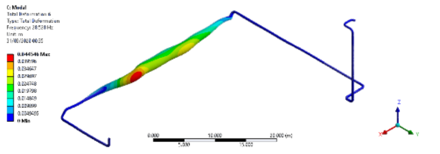
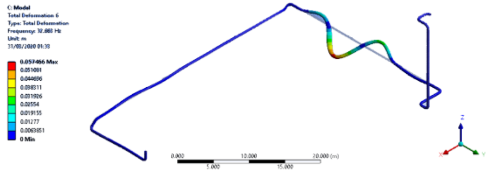
4.7.5. Modal Analysis without Self-Weight from Variation 3

Table 4.9 shows result of modal analysis without self-weight from variation 3 version. It shows that value of maximum deformation is in between 49.34mm to 67.29mm. It increase by 3.02mm equivalent to 4.5% compared to variation 2. The lowest natural frequency of vibration for guide that allows X and Y direction is 5.45Hz. Improved by 0.24Hz equivalent to 4.40% compared to variation 2. It occurs at vertical pipe orientation. While lowest natural frequency of vibration for guide that constraint in X and Y direction is 18Hz. It has been improved by 8.06Hz equivalent to 44.78% compared to variation 2.

TABLE 4.9: Result of modal analysis without self-weight from variation 3

	Guide (Allow: X and Y direction)			Guide (Constraint: X and Y direction)		
Mode	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape
1	5.45	35.55		18.00	62.95	

2	9.04	39.60		22.08	67.29	
3	18	62.95		24.84	66.24	
4	18.07	49.34		28.53	44.55	
5	22.45	48.86		31.62	51.52	

6	28.53	44.55		32.66	57.47	

4.7.6 Comparison between Result of Modal Analysis with self-weight and without Self-Weight from Variation 3

Figure 4.32 shows the comparison between results of total deformation with self-weight and without self-weight from variation 3. Pipe without self-weight experienced higher total deformation than pipe with self-weight. This is because there is lower load in pipe without self-weight than pipe with self-weight. Discussion has been stated in section 4.3.7. Average range percentage difference of total deformation between pipe with and without self-weight is in between 62% to 63%. The average range percentage difference of total deformation for variation 2 is in between 59%-62% while 54%-62% in variation 1 and 60%-63% in original model.

(a)

(b)

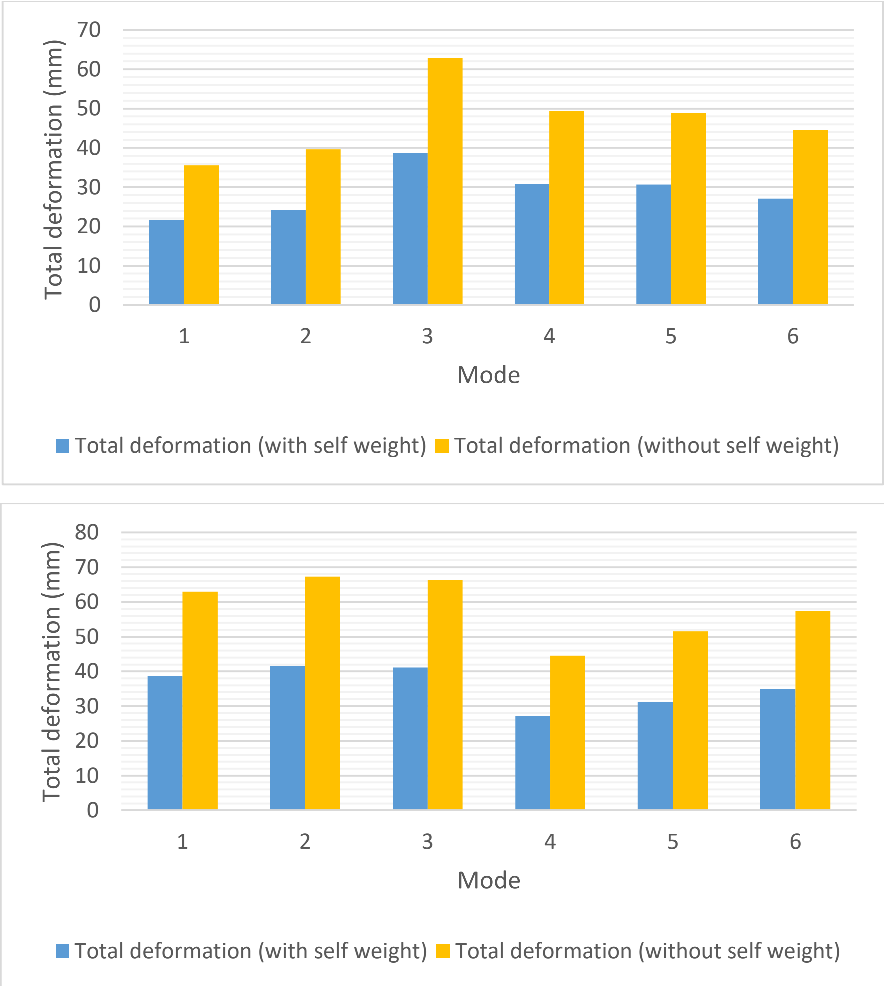


FIGURE 4.32: Graph of Comparison between results of total deformation with self-weight and without self-weight from variation 3. (a) Allows in X and Y direction; (b) constraint in X and Y direction.

Figure 4.33 shows the comparison between results of natural frequency with self-weight and without self-weight from original version. Pipe with self-weight has lower natural frequency than pipe without self-weight. The range of average percentage different of natural frequency between pipe with self-weight and without self-weight is in between 62.80%-62.85%. While for variation 2 is in between 59%-60%, 61%-75% for variation 1 and 60% for original model. The range of average percentage difference for this model is decreased**.**

(a)

(b)

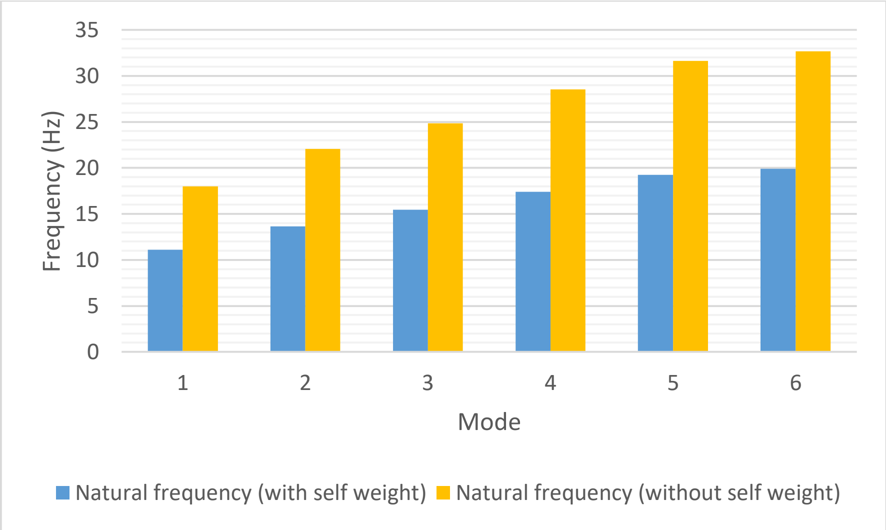


FIGURE 4.33: Graph of Comparison between results of natural frequency with self-weight and without self-weight from variation 3. (a) Allows in X and Y direction; (b) constraint in X and Y direction.

4.8 Model for Variation 4

Based on previous result, hotspot area being detected by considering the most critical part of the pipe. Additional number of supports that have been introduced in variation 3 has been modified as improvement need to be done. Figure 4.34 shows the number, location and pipe of supports that been introduced for variation 4 model. Additional saddle plate 1, saddle plate 9 and additional trunnion 1 is been removed from the system. This is because from previous result, this area create high stress value in the pipe. Additional of 1 saddle plates and 1 guide is set to be in this model. Total additional pipe supports are 11 saddle plates and 2 trunnion and 1 guide. The location of additional pipe support is shown in Figure 4.34.

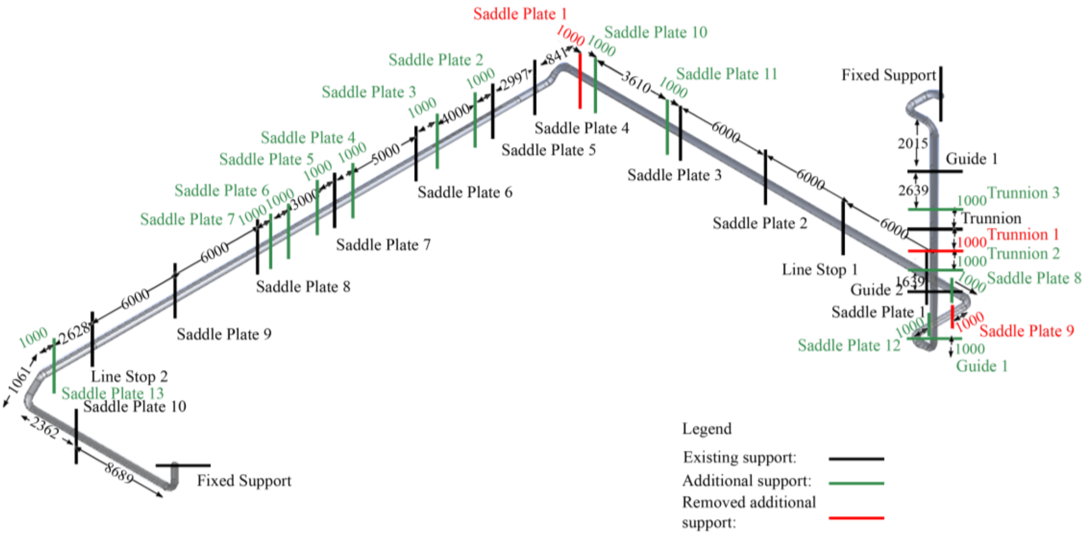


FIGURE 4.34: Pipe support distribution for variation 4

4.8.1 Static Structural with Self-Weight from Variation 4

Based on Figure 4.35, points that exceed allowable stress limit are point 1, point 5, point 6 and point 7. In variation 3, stress value for point 3 is above allowable stress limit. However, in this variation of number of pipe supports, the stress value for point 3 decrease from 293.04MPa to 33.71MPa. Improvement by 88.5%. This is after additional saddle plate 9 is removed. After comparing with variation 3, stress value produced at point 1 is decrease from 196.01MPa to 180.20MPa, point 5 increase from 226.85MPa to 404.94MPa, point 6 decrease from 527.76MPa to 522.91MP while point 7 produced 214.54MPa decrease by 17.79MPa compared to variation 3. Stress value at point 7 still exceed the allowable stress limit because of the set up in boundary condition.

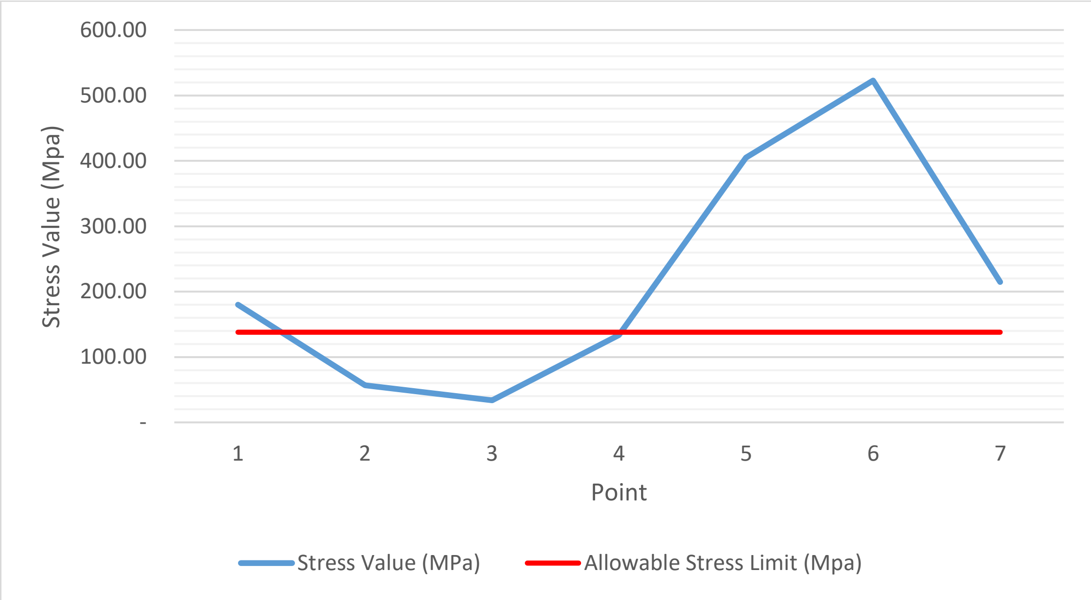


FIGURE 4.35: Graph of Stress value for variation 4 with self-weight

Figure 4.36 shows total deformation of the pipe for variation 3 with self-weight. Value for maximum total deformation has improved from 35.67mm to 35.6mm after comparing with previous variation. This is because of the additional of the saddle plate. Maximum total deformation is improved by 0.2%.

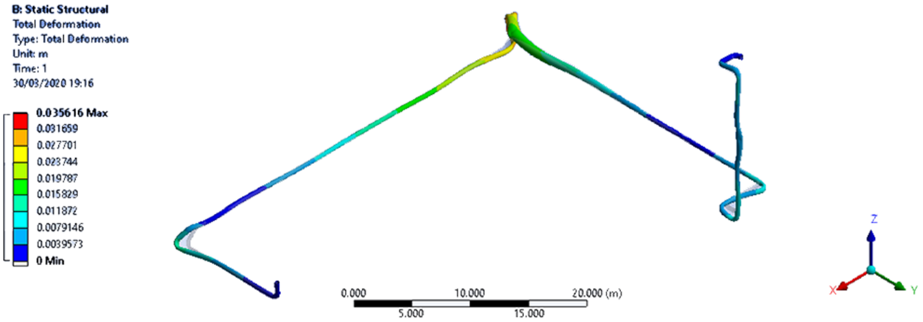


FIGURE 4.36: Total deformation of pipe for variation 4 with self-weight

4.8.2 Static Structural without Self-Weight from Variation 4

Figure 4.37 shows the result of stress value for original condition without self-weight. The result shows that the stress value at point 1, point 5, point 6 and point 7 exceed the allowable stress limit. The reason behind this has been discussed in section 4.8.1.

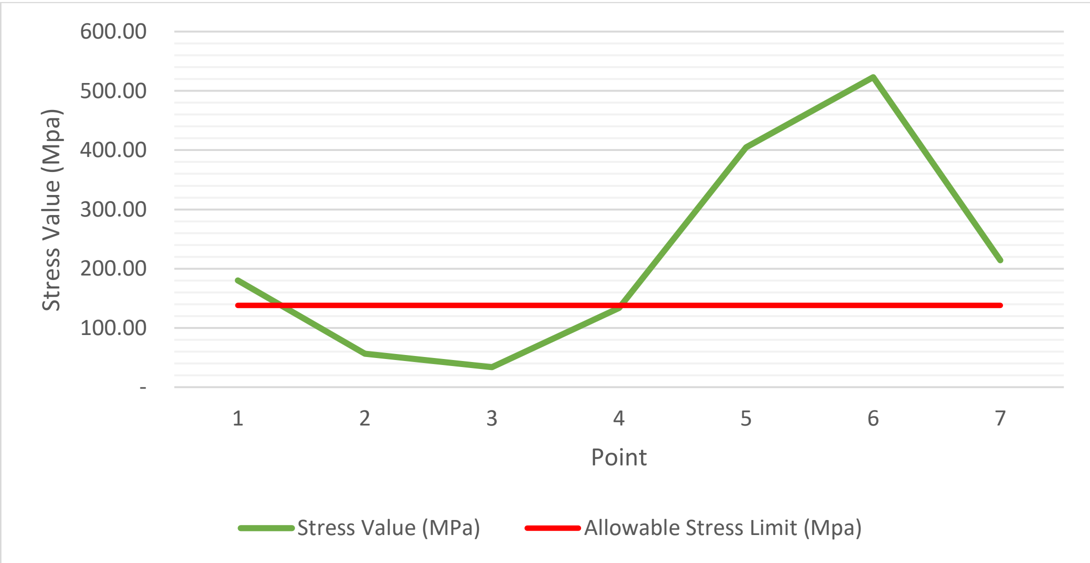


FIGURE 4.37: Graph of Stress value for variation 4 without self-weight

Figure 4.38 shows total deformation of pipe without self-weight. The maximum value of total deformation is improved from 35.67mm to 35.62. The additional of number of supports improved the maximum value of total deformation as it support the pipe and reduce the gap between supports.

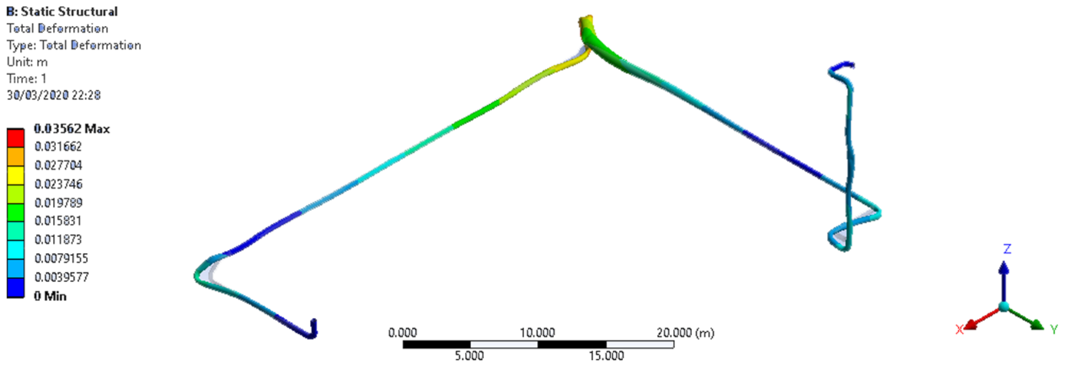


FIGURE 4.38: Total deformation of variation 4 pipe without self-weight

4.8.3 Comparison between Result of Static Structural with self-weight and without Self-Weight from Variation 4

Figure 4.39 shows the comparison between results of stress value with self-weight and without self-weight. There is only small difference between the results mentioned. However, stress value without self-weight is smaller than stress value with self-weight. The average percentage difference of stress value between these two conditions is 0.02%. In variation3, it has 0.65% of average percentage difference. While variation 2 has 4.68% average percentage difference where else in variation 1, it has 15.3% of average percentage value. In addition, difference of maximum total deformation between both conditions is 0mm equivalent to 0%. High amount of support number make system become more rigid.

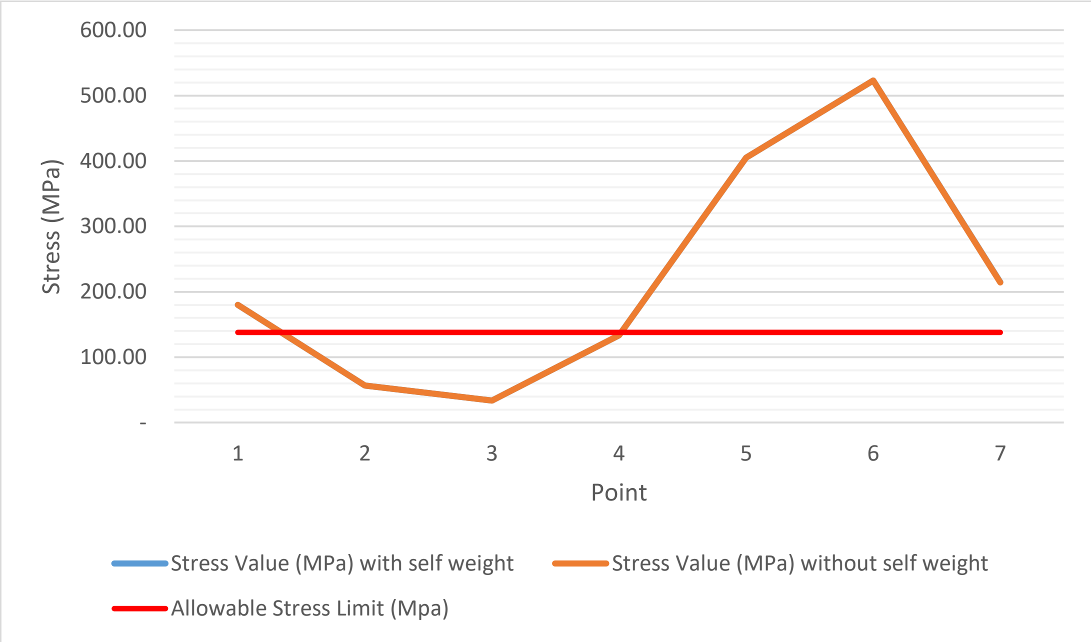
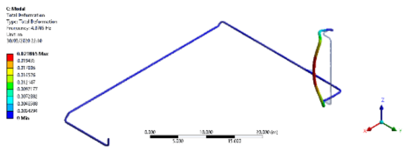
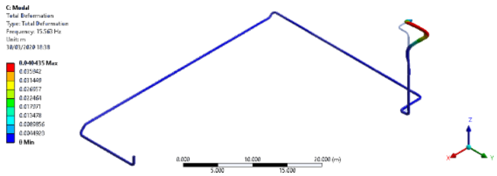
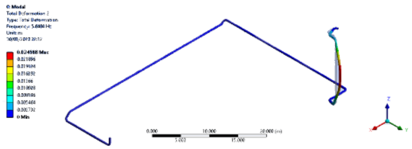
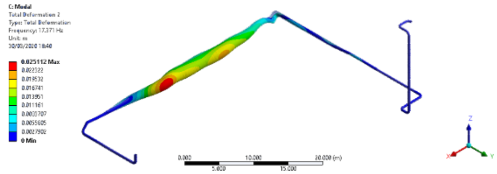
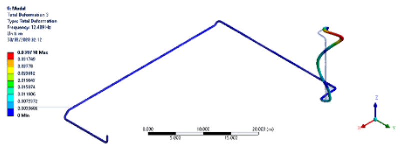
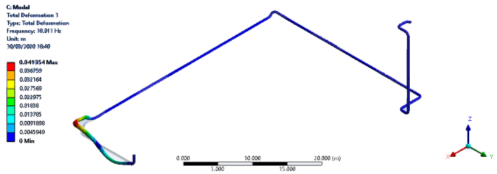


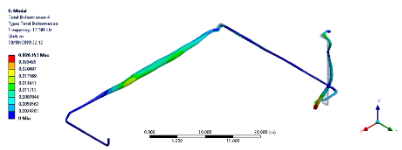
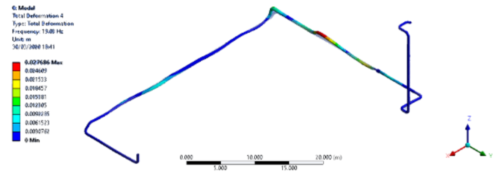
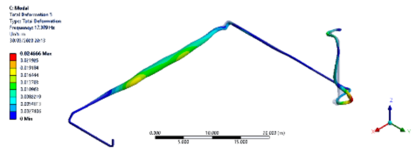
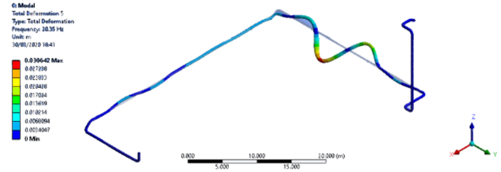
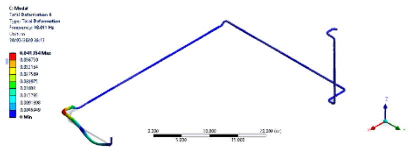
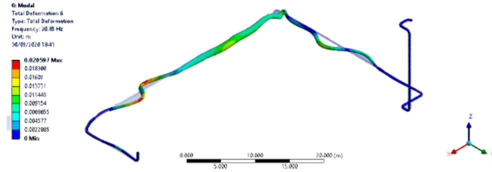
FIGURE 4.39: Graph of Comparison between results of stress value with self-weight and without self-weight from variation 4

4.8.4 Modal Analysis with Self-Weight from Variation 4

Table 4.10 shows result of modal analysis with self-weight follows the variation 4 boundary condition. Minimum frequency of vibration in guide that allows X and Y axis is 4.07Hz. It occurs in vertical orientation of pipe. However the minimum frequency of vibration in guide that fix X and Y direction is 15.56Hz. Instead of occurred at vertical piping, it occurs at different location which happened because of pipe support distribution and design of the pipe. If follows the condition of boundary condition of guide that constraint in X and Y axis, the system is safe to be operated as natural frequency of mode 1 already more than 15Hz. Range of maximum total deformation is in between 35.72mm to 40.44mm.

TABLE 4.10: Result of modal analysis with self-weight from variation 4 condition

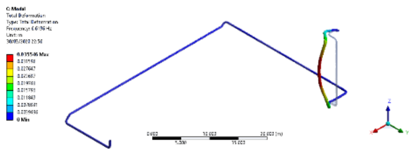
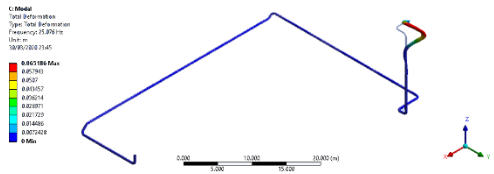
Guide (Allow: X and Y direction)				Guide (Constraint: X and Y direction)		
Mode	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape
1	4.07	21.87		15.56	40.44	
2	5.64	24.59		17.37	25.11	
3	12.49	35.72		18.01	41.35	

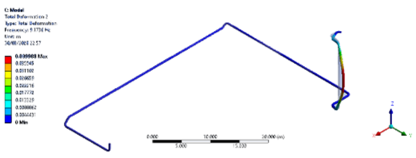
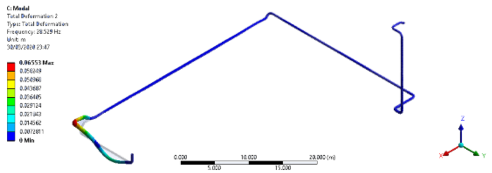
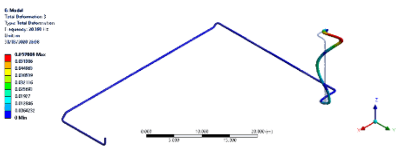
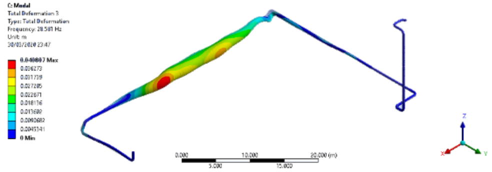
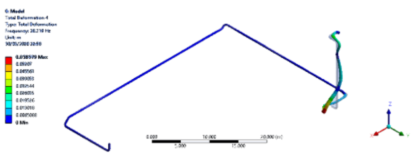
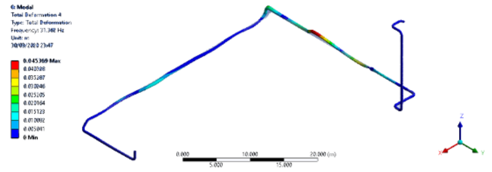
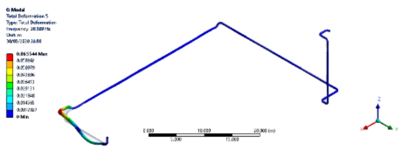
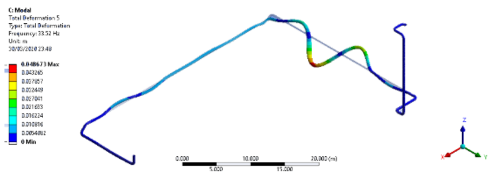
4	17.36	26.35		19.03	27.69	
5	17.38	24.67		20.35	30.64	
6	18.01	41.35		20.85	20.60	

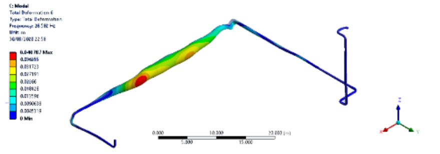
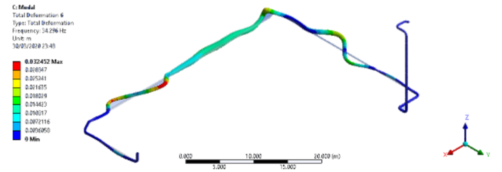
4.8.5. Modal Analysis without Self-Weight from Variation 4

Table 4.11 shows result of modal analysis without self-weight from variation 4 version. It shows that value of maximum deformation is in between 65.54mm to 65.53mm. It decreased by 1.76mm equivalent to 2.62% compared to variation 3. The lowest natural frequency of vibration for guide that allows X and Y direction is 6.625Hz. Improved by 1.17Hz equivalent to 0.22% compared to variation 3. It occurs at vertical pipe orientation. While lowest natural frequency of vibration for guide that constraint in X and Y direction is 25.08Hz. It has been improved by 7.08Hz equivalent to 39.3% compared to variation 3.

TABLE 4.11: Result of modal analysis without self-weight from variation 4

Guide (Allow: X and Y direction)				Guide (Constraint: X and Y direction)		
Mode	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape	Frequency (Hz)	Maximum Deformation (mm)	Mode Shape
1	6.62	35.55		25.08	65.20	

2	9.17	40.00		28.53	65.53	
3	20.19	57.81		28.58	40.81	
4	28.22	58.58		31.36	45.37	
5	28.53	65.54		33.52	48.67	

6	28.58	40.79		34.30	35.45	

4.8.6 Comparison between Result of Modal Analysis with self-weight and without Self-Weight from Variation 4

Figure 4.40 shows the comparison between results of total deformation with self-weight and without self-weight from variation 4. Pipe without self-weight experienced higher total deformation than pipe with self-weight. This is because there is lower load in pipe without self-weight than pipe with self-weight. Discussion has been stated in section 4.3.7. Average range percentage difference of total deformation between pipe with and without self-weight is in between 70% to 79%. The average range percentage difference of total deformation for variation 3 is in between 62% to 63%, while variation 2 is in between 59%-62% while 54%-62% in variation 1 and 60%-63% in original model.

(a)

(b)

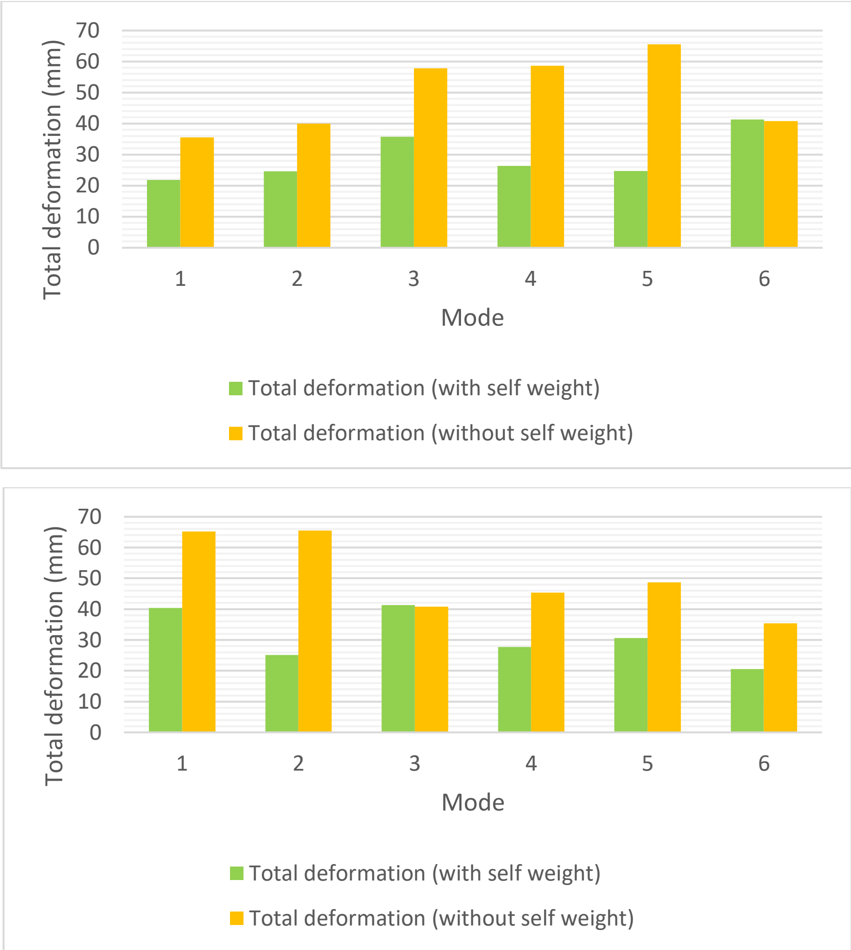


FIGURE 4.40: Graph of Comparison between results of total deformation with self-weight and without self-weight from variation 4. (a) Allows in X and Y direction; (b) constraint in X and Y direction.

Figure 4.41 shows the comparison between results of natural frequency with self-weight and without self-weight from original version. Pipe with self-weight has lower natural frequency than pipe without self-weight. The range of average percentage different of natural frequency between pipe with self-weight and without self-weight is in between 62%-63%. While for variation 3 is in between 62.80%-62.85%, 59%-60% for variation 2, 61%-75% for variation 1 and 60% for original model. The range of average percentage difference for this model is decreased respect to the increasing number of supports.

(a)

(b)

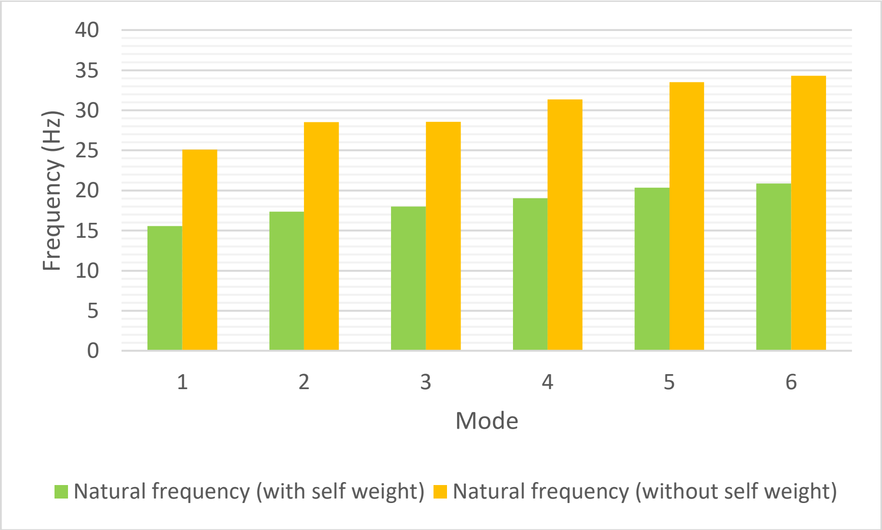


FIGURE 4.41: Graph of Comparison between results of natural frequency with self-weight and without self-weight from variation 4. (a) Allows in X and Y direction; (b) constraint in X and Y direction.

CHAPTER 5
CONCLUSION AND RECOMMENDATIONS

5.1 Conclusion

The significance of number of supports to pipe stress, total deformation and vibrational behaviour are being studied in this project. Various numbers of pipe supports are used to study the effect to the pipe system. Although an increase in the number of supports may lower the vibration level to an acceptable range and increases the natural frequency of the piping systems to be above 15Hz as recommended by ASME 2016, the large number of supports applied unfortunately, induces a larger stress onto the pipe because of the constraints. An optimum number of support needs to be introduced to pipe system to solve stress and vibration issue. In this study, vibration issue was managed to be improve by 81.62% considered solved. However, their stress value at critical points still exceeded the allowable stress limit by 74%. This shows that the optimizing of natural frequency and pipe stress of the piping could not be achieved through the variations introduced in the addition of supports.

5.2 Recommendations

For future development, it is recommended to study the effects of types of supports on pipe stress and vibration issue of a piping system. The number of supports applied to the system may be reduced if type of support used is suitable to the pipe system. Since this study considered the pipe flow as an ideal flow which is a single phase flow, the piping flow can be regarded in the future as a two-phase flow which involves gas and liquid.

REFERENCES

[1] J. C. Wachel, “Piping Vibration Analysis,” pp. 119-134, 1990.

[2] A. M. &. J. Hare, “Offshore hydrocarbon releases 2001-2008,” Health and Safety Laboratory, Derbyshire , 2008.

[3] Zachwieja, Janusz, “Stress analysis of vibrating pipelines,” in AIP, Bydgoszcz, 2017.

[4] Finnveden, S.; R., J. Pinnington, “A velocity method for estimating dynamic strain and stress in pipes,” *Journal of Sound and Vibration*, vol. 1, no. 229, pp. 147-182, 2000.

[5] R. D. Blevins, “Flow-induced vibration in nuclear reactors,” Progress in Nuclear Energy, vol. 4, pp. 25-49, 1979.

[6] E. Udoetok, “Internal Fluid Flow Induced Vibration of Pipes,” *Mechanical Design and Vibration*, vol. 6, no. 1, pp. 1-8, 2018.

[7] S. Kaneko, T. Nakamura, F. Inada, M. Kato, K. Ishihara, T. Nishihara and A. L. Mikael , “Vibrations Induced by Internal Fluid Flow,” in *Flow-induced Vibrations*, Elsevier Ltd., 2014, pp. 157-195.

[8] e. a. Shigehiko K, “Vibrations Induced by Internal Fluid Flow,” in *Flow-Induced Vibrations*, Elsevier Ltd., 2014, pp. 157-193.

[9] H., Dai L. ; L., Wang; Q., Qian; Q., Ni, “Vortex-induced vibrations of pipes conveying fluid,” *Journal of Fluids and Structures*, no. 39, pp. 322-334, 2013.

[10] Guo, H. Y, Lou,M, “Effect of Internal Flow on Vortex-Induced Vibration of Risers,” *Fluids and Structures*, no. 14, p. 496–504, 2008.

[11] Zhang, T.; Y., O. Zhang; H. , Ouyang, “Structural vibration and fluid-borne noise induced by turbulent flow,” *Pressure Vessels and Piping*, pp. 1-12, 2014.

[12] Eisinger, F. & Sullivan, R. & Feenstra, P. & Weaver, D.., “Acoustic Vibration in a Stack Induced by Pipe Bends,” *Pressure Vessel Technology*, vol. 2, no. 125, 2003.

[13] S. P.P.Modi, “Pressure Losses and Flow Maldistribution in Ducts with Sharp Bends,” *Chemical Engineering Research and Design*, vol. 82, no. 2, pp. 321-331, 2004.

[14] S. Kannappan, Introduction to Pipe Stress Analysis, ABI Enterprises, Incorporated, 2008.

[15] Engelkemier, Monte; Minn, Wayzata; Cargill, PMP, “How to perform a pipe stress analysis,” Stanley Consultants, 2017.

[16] J. C. Wachel, “Piping vibration and stress,” pp. 1-9, 1980.

[17] Rahul Jain, Surjya K. Pal, Shiv B. Singh, “Numerical Modelling Methodologies for Friction Stir Welding Process,” *Computational Methods and Production Engineering*, pp. 125-169, 2017

[18] W. T. S. L. W. X. Jiang X, “Stress Analysis of Unburied Gas Pipelines Laid in High Slopes,” *Nat Gas Oil*, vol. 31, no. 6, p. 26–30, 2013.

[19] M. F. Dr. D.P. Vakharia, “Determination of Maximum Span between Pipe,” *Recent Trends in Engineering*, vol. 1, 2009.

[20] Greer, B., Harper, C., Rahnama, R., Improved Thermal Piping Analysis for Reciprocating Compressor Piping Systems, Austin: BETA Machinery, 2012.

[21] Iyad A l-Zaharnah, “Introduction and scope of work,” *Thermal Stresses in Pipes*, pp. 1-5, 2002.

[22] Engineers, The American Society of Mechanical, *ASME*, 2017.

[23] ASTM A106 / A106M-19a, “Standard Specification for Seamless Carbon Steel Pipe for High-Temperature Service,” *ASTM International*, West Conshohocken, PA, 2019.

[24] M.B Standing, “Density of Crude Oils Saturated with Natural Gas,” *Society of Petroleum Engineers*, 1942

[25] C. Miki and N. Oguchi, “Deformation and Fracture Properties of Steel Pipe Bend with Internal Pressure Subjected to In-Plane Bending,” 2000.

[26] M. R. A1-Abed, C. C. Baniotopoulos,C. D. Bisbos & P. D., “Saddle-Supported Pipelines: Computation of the pressure distribution on pipe saddle interface,” vol. 44, no. 1-2, pp. 159-178, 1997

[27] American Society of Mechanical Enginners, “Process Piping: ASME Code for Pressure Piping,” *B31: an American National Standard, ASME*, 2016.

[28] Kulkarni, Siddharth Suhas; Chapman, Craig ; Hanifa, Shah;, “Computational Fluid Dynamics (CFD) Mesh Indepedency Study of a Straight Blade Horizontal Axis Tidal Turbine,” pp. 1-17, 2016.

[29] Ni, Qiao; Yangyang , Luo; Mingwu, Li;, “Natural Frequency and Stability Analysis of a Pipe Conveying Fluid with Axially Moving Supports Immersed in Fluid,” *Sound and Vibration*, vol. 403, pp. 173-189, 2017.

[30] Z. Tang and Z. Benmou , “The Effect of Mass Ratio and Spring Stiffness on Flow-Induced Vibration of a Square Cylinder at Different Incidences Angles,” *Ocean Engineering*, vol. 198, pp. 1-12, 2020.

[31] Group Technical Solutions, “Pipe Stress and Vibration Analysis for PCIHBV,” *PETRONAS Carigali Iraq Holding B.V.*, Iraq, 2015.