CHAPTER 1

INTRODUCTION

Refrigerant piping system is employed in air-conditioners to transfer heat from air inside of a room to the outside air. The most basic requirement of refrigerant piping stress is to ensure adequate flexibility in the piping system for absorbing the thermal expansion or contraction of the pipe. If the piping system does not have enough flexibility to absorb the expansion or contraction, the force and stress generated can be large enough to ruin the piping and the connecting equipment. Therefore, it is important to ensure that the piping is free from high stress resulting from the thermal expansion or contraction during operation to prevent product short life cycle and failure.

In order to achieve distinctive and economical piping system design, the design parameters affecting the piping stress demands extensive research and studies. There are limited experimental investigation has been carried out on air conditioning piping system from the literature. However, much more work based on computational finite element analysis can be found.

There are numerous parameters affecting the piping stress for instance pipe wall thickness, pipe length, temperature difference between suction temperature and ambient temperature, number of bend, internal pressure of pipe, material of pipe and so forth. Current work reinforces in the study of pipe wall thickness, temperature difference between suction temperature and ambient temperature, and length of pipe.

The common problem faced by piping designer in designing a piping system with good flexibility is to have an overpriced piping system. A good understanding of the effects of pipe wall thickness and pipe length on thermal stress is required in order to design a cost-effective piping system.

1.1 Research Problem Statement

For split air-conditioning system whereby the indoor unit is separated with the outdoor unit, the indoor and outdoor unit is designed to operate in an allowable ambient temperature envelope for good reliability. The difference of ambient temperature in which the indoor and outdoor unit operated affects the temperature of refrigerant. The temperature difference between the suction temperature and ambient temperature induces the thermal expansion or contraction. Current work focuses on the thermal contraction of the suction pipe when the units operate in four different conditions.

In addition, feasibility study has been made to investigate the potential of using finite element analysis in pipe stress analysis. The advantages of using finite element analysis is to reduce testing time and cost. Besides that, finite element analysis can be used to display the high stress concentration location on the pipe.

1.2 Objectives of Research

This research is carried out in order to achieve the following objectives:

a) To study the effect of pipe wall thickness on pipe stress by experiment.

b) To study the effect of temperature difference on pipe between suction temperature and ambient temperature by experiment.

c) To study the effect of pipe length toward piping stress by experiment.

d) To study the effect of pipe length on stress by use of finite element analysis

1.3 Scope of the Study

The current study is intended to cover:

a) Parametric study of the effect of pipe wall thickness on pipe stress

b) Parametric study of the temperature difference between suction temperature and ambient temperature on piping stress for an air-conditioning unit operates within a specified ambient temperature envelope for reliability assurance.

c) Parametric study of the effect of pipe length on pipe stress

d) Validation result of empirical testing and simulation approach for the pipe stress length.

CHAPTER 2

LITERATURE REVIEW

2.1 Parametric Study of Piping Stress

Piping systems are used to carry liquids and gases being transport which under pressure and thermal loads. Piping used in pumps, heat exchangers, valves and some vessels. Piping design parameter affected by diameter of the pipe, length of pipe, number of bend, pipe configuration, material, support stiffness, material of pipe, temperature of pipe and internal pressure.

Parameter study in finite element is crucial to be studied to examine specific issues occur on the pipe. The basic concept of the finite element method is to seek the solution of an intricate problem by replacing it by a simpler object. This method will be able to find approximate solution rather than exact solution as the actual problem is replaced by a simple object. This method can be refined to the approximate solution by spending more computational effort.

Failure of pipe occurs when the material resistance of the pipe unable to withstand the applied local stress or strain. There are various failure modes such as yielding of pipe due to excessive plastic deformation. Pipe body will undergo plastic deformation under slip action of grains. Material will re-crystallize after slippage. Creep will happen as material yielding continues without increasing any load and cause pipe failure. Piping stress can be sensed and amplified the signal by transducer and result can be captured in oscilloscope. Kannappan (1986) has written in his book that for nonferrous metals, such as copper, aluminium and plastic do not show yield point and differ with stress-strain curve. The "point" where the stress cause permanent strain of 0.2% with the load removed is regarded as the yield point of such material called as "yield strength".

2.1.1 Thermal Expansion Force and Stress

The main requirement in piping design is to provide adequate flexibility for absorbing the thermal expansion of the pipe. A pipe is expands or contracts, as the pipe temperature changes from the installation condition to the operating condition. Generally, the term for both expansion and contraction are called thermal expansion. The pipe has the potential to generating enormous force and stress in the system when a pipe is expanded. On the other hand, the expansion can be absorbed without creating excessive force or stress if the piping is flexible enough. Providing the proper flexibility is one of the major tasks in the design of piping system.

Piping is used to transfer a certain amount of fluid from one point to another. It is noted that the shorter the pipe is used the lesser the capital expenditure is required. The long pipe may also generate excessive pressure drop making it incompatible for the proper operation. However, the direct shortest layout generally is not acceptable for absorbing the thermal expansion.

Figure 2.1 shows what will happen when a straight pipe is directly connected from one point to another. First, consider that only one end is connected and the other end is loose. The loose end will expands an amount equal to

$\Delta = \alpha\,(T_2 - T_1)\,L = eL$ ……………………………………………………Equation 2.1

However, since the other end is not loose, this expansion is to be absorbed by the piping. This is equivalent to squeezing the pipe to move the end back a Δ distance. This is equivalent to squeezing the pipe to move the end back a Δ distance. This amount of squeezing creates a stress of the magnitude

$s = E\,(\Delta/L) = Ee$ …………………………………………………………..Equation 2.2

The force required to squeeze this amount is

$F = As = AEe$ …………………………………………………...……..Equation 2.3

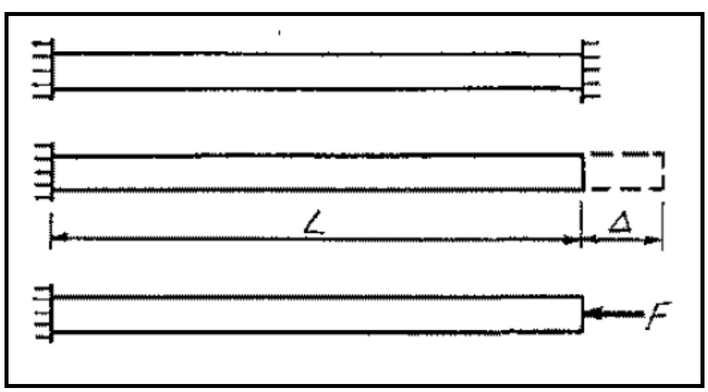

Figure 2.1: Thermal expansion force

Where,

Δ = thermal expansion (in)

α = Thermal expansion rate (in/in-°F)

T_2 = End temperature (°F)

T_1 = Starting temperature (°F)

L =pipe length, in

e = expansion rate, in/in

S = axial stress, psi

E = modulus of elasticity, psi

A =pipe cross section area, in^2

F = axial force, lbs

2.1.1.1 Method of Providing Flexibility

Piping flexibility is provided in many different ways. The turns and offsets of the pipe are needed to provide some flexibility to them. Additional flexibility can be provided by adding expansion loops or expansion joints. The stress can be reduced by a loop installed as shown in Figure 2.2 or by an expansion joint as shown in Figure 2.3.

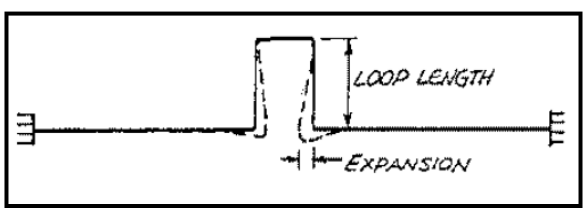

Figure 2.2: Piping loop

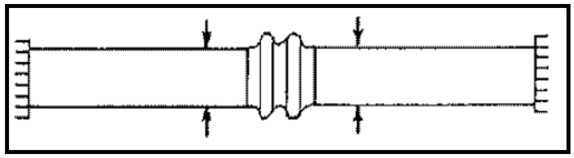

Figure 2.3: Expansion joint

Peng (2011) study has depicted the reason to provide some pipe perpendicular to the direction of expansion loop. As the pipe expands it bends the loop leg first before transmitting any load to the anchor. The longer the loop leg the lesser the force will be created. The force created is inversely proportional to the cube of the loop length and the stress generated is roughly inversely proportional to the cube of the loop length. Expansion joints are more sophisticated than the pipe loops which are just extra lengths of the same piping.

The expansion stress can be estimated by the guided cantilever approach. The method can be explained using the L-bend given in Figure 2.4 as an example. In free constraint condition, the system is not constrained the points B and C will move to B' and C' respectively due to thermal expansion. The end point C moves d_x and d_y respectively in X- and Y- directions, but no internal force or stress will be generated. However, in the actual case the ends of the piping are always constrained as shown in Figure 2.4 b). This is equivalent in moving the free expanded end C' back to the original C forcing the point B to move to B'. The d_x is the expansion from leg AB, and d_y from leg CB. The deformation of each leg can be assumed to follow the guided cantilever shape. This is conservative because the end rotation is ignored. The force and stress of each leg can now be estimated by the guided cantilever formula.

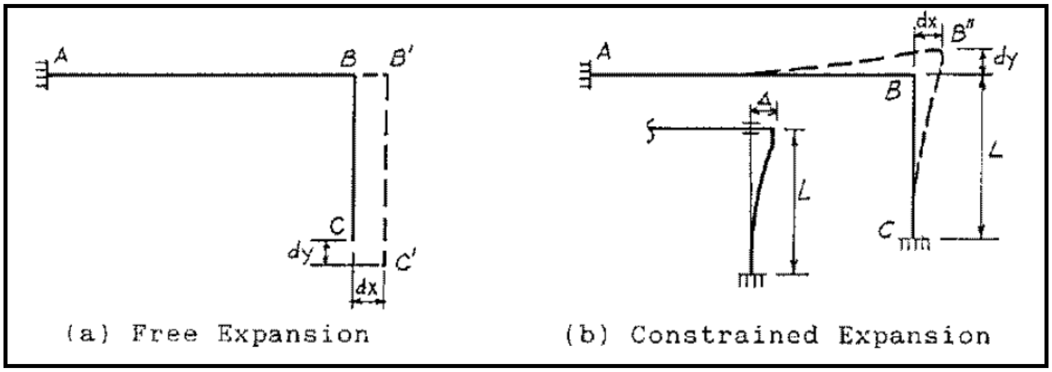

Figure 2.4: Free and constrained expansion

The leg AB is a guided cantilever subject to d_y displacement and leg CB a guided cantilever subject to d_x displacement respectively.

From the basic beam theory, the moment and displacement relation of a guided cantilever is

$$M = \frac{6EI}{L^2}\Delta \quad , \quad F = \frac{2M}{L} \qquad \text{...Equation 2.4}$$

For thin wall pipes, above equation can be further reduced. By using $I=\pi r^3 t$ and $S=M/(\pi r^2 t)$, the above equation becomes

$$S = \frac{6Er}{L^2}\Delta \; = \frac{ED\Delta}{48l^2} \qquad \text{...Equation 2.5}$$

Where,

S = thermal expansion stress, psi

E = modulus of elasticity, psi

r = mean radius of the pipe, in

Δ = total expansion to be absorbed, in

L = length of the leg perpendicular to the thermal expansion, in

l = length in feet unit, ft

D = outside diameter of the pipe, in

Al-Zaharnah et al.(2001) also stated in his study that the stress ratio is reduces at some location on the pipe wall as the location moves forwards to the pipe end as the axial distance increases.

2.1.2 Wall Thickness and Thermal Expansion Stress

In Peng (2011) study has also mentioned that because expansion stress is calculated by dividing the moment, M with the section modulus, Z, engineers might be wrongly tempted to increase the wall thickness to reduce the expansion stress. An increase in wall thickness increases the section modulus, but also proportionally increases the moment of inertia. The section modulus is defined as Z=I/r, which is directly proportional to the moment of inertia. Therefore, the first consequence of increasing the wall thickness is an increase in bending moment under a given thermal expansion. This increased moment divided by the proportionally increased section modulus ends up with the same stress as before, prior to the increase of the thickness. The thicker wall thickness does not reduce the thermal expansion stress. It only unfavorably increases the forces and moments in the pipe and at the connecting equipment. Therefore, as far as thermal expansion is concerned, the thinner the wall thickness the better it will be for the system.

2.1.3 Pipe Temperature

In Fonseca et al.(2005) study the pipe may not be only subjected to mechanical loads but also operate in several of thermal load. The severe temperature gradient, the higher thermal stress produced. Structural of piping exposed to temperatures rise will consequent of thermal expansion to pipe length. Bieniussa (1999) stated a change in temperature is affecting density of material. As the temperature increases, the density will decrease. The upper area of the cross-section of the pipe have greater temperature compare to lower area which affected by difference density. The thermal stratification caused cyclical stress applied on piping. Stress index method is used to access cyclical stress for piping and this testing is permitted by the framework of the "Component Specific Analysis of Mechanical Behavior", the nuclear codes and standards, e.g. KTA standard KTA 3201.2. In additional, Al-zaharnah et al.(2000) has presented low Prandtl number and low thermal conductivity ratios result in low radial effective stresses. Higher radial stress is generated in the inlet plane of the pipe compare to that corresponding to other plane in the axial direction.

To improve the oscillated stress on the pipe, Kandil (1995) has suggested the inner surface should heat gradually up to the operating temperature. From the study outcome, the value of maximum effective stress at inner surface may decrease to about 50 or 60 percent for the normalized heating time. However, Al-Zaharnah et al. (2001) suggested flow pulsation in the circular pipe can be heated externally. The effective stress rises rapidly at the initial stage and stress level becomes steady as the heating progresses. In Chattopadhyay (2009) have concluded that mixing of cold and hot fluids under certain low flow condition will cause failures in piping due to thermal stratification in nuclear piping system. The stratified temperature fluid profiles increase circumferential metal temperature gradient through the pipe and leading to high stresses caused fatigue damage.

2.1.4 Internal pressure

Piping is subjected to a uniform pressure loading when piping is pressurized. The pressure loading produce stresses in the direction normal to the pipe wall, parallel to the pipe axis and create stress tangential to the cross section circle.

In Lubis et al. (2004) has pointed out that the pipe will become unsecure if internal pressure increase in pipe. However, internal pressure will be reduced if the smooth pipe enhances the flexibility with bending. Moreover, this condition will not affect the critical constant much hence it is not considered in ours study.

Middleton et al. (1996) has reviewed that the integrity of material operate under condition will cause degradation processes occur. In order to ensure the pipe able to operate in designed operating condition of stress, temperature and environment; the piping design need to have a suitable engineering safety factor. In Middleton et al. also mentioned that in British Standard BS 1113 for tubes, pipe work and header are not required close dimension and clearance; but, the design stress is acquire from the mean rupture stress at 10^5h as below:

$$\text{Design Stress} = \frac{\text{Stress to Rupture in } 10^5 \text{ h}}{\text{Safety factor}} \quad \text{.......................................Equation 2.6}$$

Plus an increase in thickness such as a bending allowance. Furthermore, Lin et al. (2004) used general sampling computation method of stress intensity factor (SIF) to assess the in-service pressure piping. The assessment is based on code BS7910, R6 and SAPV-99 to consider uncertainties in operating loading, flow stress, flaw size and material fracture toughness.

2.2 Finite Element Analysis (FEA) Introduction

Finite element analysis (FEA) is a tool for numerical solution extensively used in structural, fluid flow analysis and thermal areas. Complicated design can be modeled with relatively ease, with the introduction of CAD system. Understanding of the basic theory, modeling techniques and computational aspects of the finite element analysis is important to be used in simulation before first prototype is built.

The finite element analysis was first brought by R.Courant who utilized the Ritz method of numerical analysis and thereafter Turner et al.(1956) pointed out triangle shape of finite element is described as three corner nodes can represent to the in-plane loaded plates in aircraft structure. Hence, this method has been recognized worldwide and widely used in various types of applied science and engineering problem.

Rao (2005) has written that in the finite element method, the model called as finite element is built up of many small, interconnected sub-regions. The element are interconnected at specified joint and called as nodes or nodal points. The nodes which lie on the element boundaries are considered to be connected with adjacent element. Since the actual variation of the field variable such as displacement, stress, temperature, pressure, or velocity is not known, thus, the variation is assumed to be a simple function inside finite element. The field equation for the whole continuum is written in the form of matrix equation, the new unknown to be the nodal values of the field variable is known. When the nodal values are known, the approximating functions define the field variable throughout the assemblage of element.

The original body or domain such as shapes, sizes, number and configurations of the element is being simulated need to as close as possible to not increase the computational effort for the solution. For simple analysis, one –dimensional element such as trusses or frame is connected have two nodes, one at each end. Two-dimensional and three-dimensional finite element is used in the analysis of shell and mass structures. Figure 2.5 is shown one, two and three dimensional finite element. The geometry of the body and the number of independent coordinates used to describe in the system.

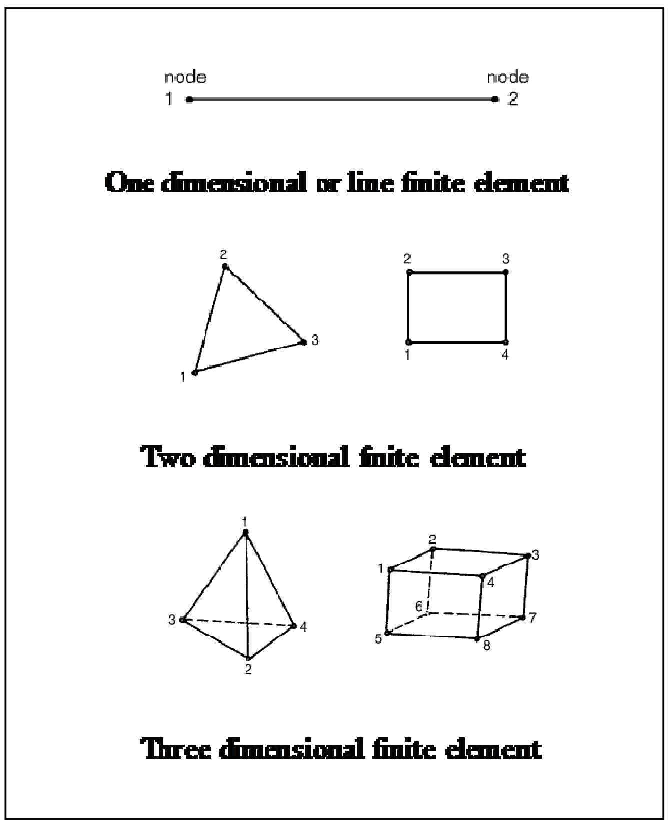

Figure 2.5: Types of finite element (Rao ,2005)

The ABAQUS is an effective tool in engineering simulation research. It provided general choice of solution in linear and non-linear, and static and dynamic types of finite element package. The analysis involves solid bodies to temperature, contact, impact load and other environment conditions. Generally, ABAQUS simulation saves time to accomplish complicated mathematic calculation. ABAQUS simulation can do verification on data errors before simulation start. In additional, the FEA can trace the errors location so that user can correct the error instantly before running the simulation. Nevertheless, Nakamura (1987) has commented the finite element analysis can based on the result of experiment which can used to evaluate simplified method to compare with more detailed finite element solution. Electric Power Research Institute has contributed partial funding for the development and resulted useful in structural integrity evaluation where mentioned by Hibbitt(1983). The finite element is independent because the assumed material model can be tested in various assumptions in simplified method on comparatively realistic problem.

CHAPTER 3

METHODOLOGY

3.1 Piping Stress Empirical Testing

This chapter outlines the choice of study points on suction pipe, the experimental setup of stress test and the selection of parameters for stress study.

3.1.1 Study of Stress Points of Refrigerant Piping System

Figure 3.1 indicates the basic cooling refrigerant cycle for air-conditioning unit. An air conditioner is a system designed to change the air temperature and humidity within an area and it is used for cooling as well as heating. The cooling refrigeration process undergoes heat absorption at evaporator and heat rejection at condenser. Suction pipe plays the role of transferring the refrigerant in gas form from evaporator to accumulator before entering the compressor. Suction pipe adjacent to the accumulator of the air-conditioning system is selected as study object. This pipe has high frequency of failure because it is located near to vibrating compressor.

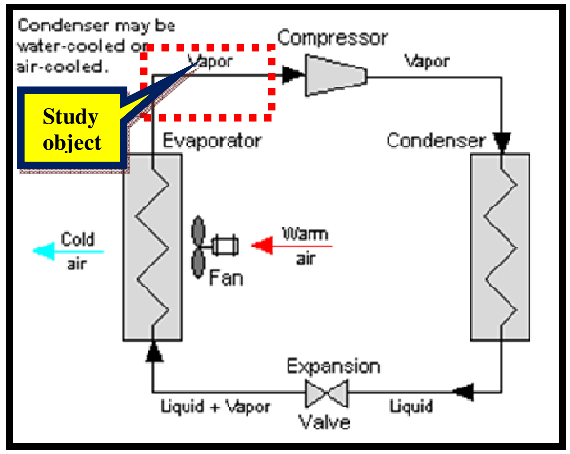

Figure 3.1: Typical air-conditioning system

At the beginning of the study, piping stress is measured for every bending point along the suction pipe by using strain gage. Due to expensive material cost for the strain gage, only two highest stress level points on the suction pipe are selected as study points as shown in the Figure 3.2 in the testing compared to others bending point. Figure 3.3 shows the position of suction pipe in the assembly unit. The suction pipe is attached from the valve to the accumulator.

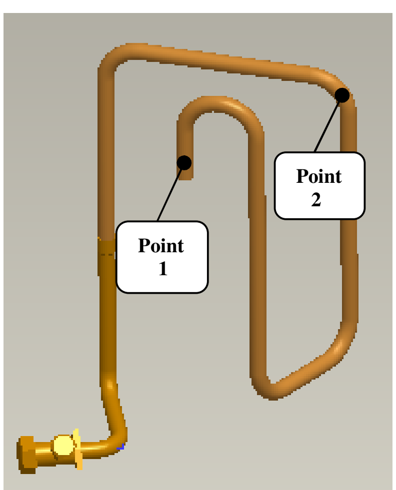

Figure 3.2: Two study points on suction pipe

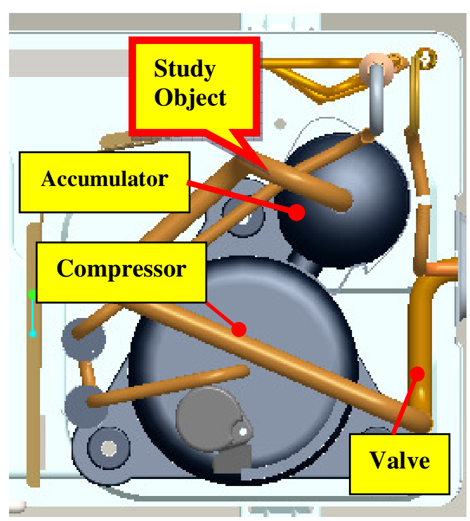

Figure 3.3: Top view of study unit

3.1.2 Experimental Equipment & Procedure

3.1.2.1 Piping Stress Test Equipment

The piping stress test is used to measure stress on pipe which the testing is conducted in psychometric test room. The strain gage bonded on pipe transfers the resistance change signal to Wheatstone bridge for strain measurement and then oscilloscope converts the resistance change to a voltage change. The output value is in voltage and it will be converted to stress in kg/mm^2 unit by calculation sheet. The equipments used in this piping stress test consist of oscilloscope, strain amplifier, strain gage, adhesive paper, strain gauge cement, polyethylene insulation and polystyrene sheet. Oscilloscope (Model DS06014A) is used for strain measurement as shown in Figure 3.4. The function of strain amplifier (DPM-712B) is to amplify the low level output signal as shown in Figure 3.5. Testing equipment is calibrated within every two months. Strain gage (model KFG-1-120-D16-16L1M2S) is used to measure strain of piping as shown in Figure 3.6. Sand paper is used to remove dirt from piping surface and to polish piping surface as shown in Figure 3.7. Strain gage cement is used to affix the strain gauge on the surface of the piping as shown in Figure 3.8. Polystyrene sheet as shown in Figure 3.9 is used as an accessory to hold and press the strain gage after applying adhesive and it is removed after the completion of adhesion. PE insulation as shown in Figure 3.10 is used to cover the strain gauge after the strain gage is applied with strain gage cement on the pipe surface.

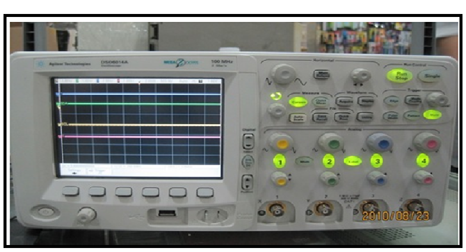

Figure 3.4: Strain Amplifier DPM-712B   Figure 3.5: Oscilloscope model DPM-712B

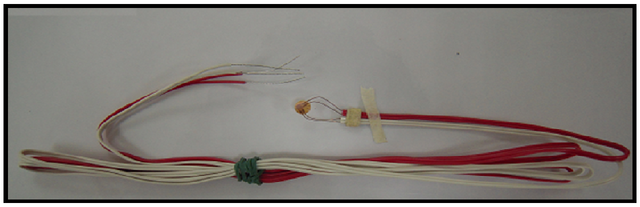

Figure 3.6: Strain gauge KFG-1-120-D16

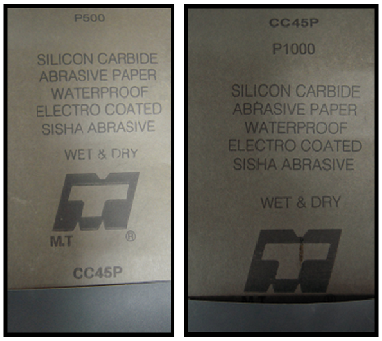

Figure 3.7: Abrasive paper in grit P500 (left side) and P1000 (right side)

Figure 3.8: Strain gauge cement

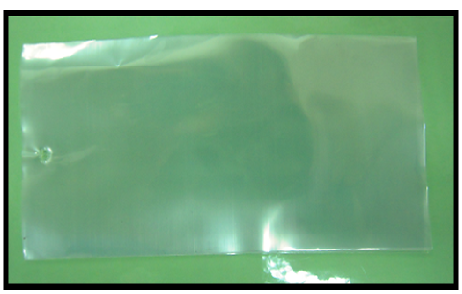

Figure 3.9: Polyethylene insulation

Figure 3.10: Polystyrene sheet

3.1.2.2 Piping Stress Test Procedure

1 Structure of piping is examined without obvious flaw, rust or discoloring. Strain gages location is identified.

2 The location of pipe is wiped with sandpaper and the pipe surface is degreased with alcohol.

3 A drop of adhesive (strain gage cement) is applied at the back of strain gage bonding site as shown in Figure 3.11.

4 The strain gage is covered with polyethylene sheet and it is pressed with thumb for approximately 1 minute as shown in Figure 3.12.

5 Polyethylene sheet is removed after the adhesive cures. Then, the strain gage is covered with polyethylene insulation as shown in Figure 3.13.

6 Two pairs of strain gage lead wires are connected to the Wheatstone bridge of the strain amplifier as shown in Figure 3.14.

7 The strain amplifier and oscilloscope are switched on. Strain amplifier is calibrated before running the testing.

8 "RUN/STOP" button on oscilloscope is pressed and the waveforms is captured and recorded. Finally, the reading from the oscilloscope is converted to stress in kg/mm^2 unit by calculation.

Figure 3.11: Strain gage cement applied at the back of strain gage bonding site.

Figure 3.12: Finger pressed on polystyrene sheet

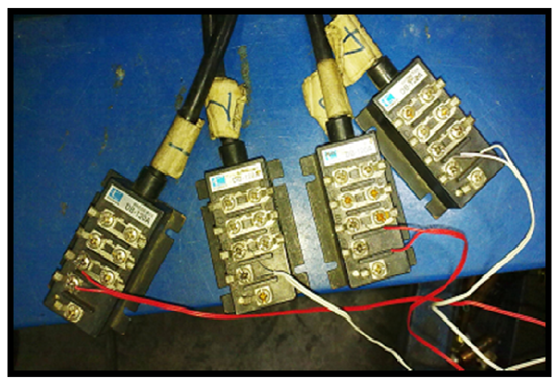

Figure 3.13: Strain gage is covered by polyethylene insulation

Figure 3.14: Lead wires of strain gage are connected to Wheatstone bridge.

3.1.3 Psychometric Test Setup

The equipment used in the psychometric testing laboratory is indoor and outdoor chamber, testing unit which are indoor and outdoor air-conditioner unit, resistance temperature detector (RTD) and pressure tapping. Indoor and outdoor chamber are used to control the testing operating ambient temperature by means of chiller, heater and humidifier. The control setting of chiller, air heater and humidifier is based on the required operating ambient temperature of the room chamber. RTD is used to measure the dry bulb and wet bulb temperature of the ambient temperature. Pressure tapping is to measure suction pressure of the suction pipe. In Figure 3.15 has shown the overview of psychometric laboratory system setup.

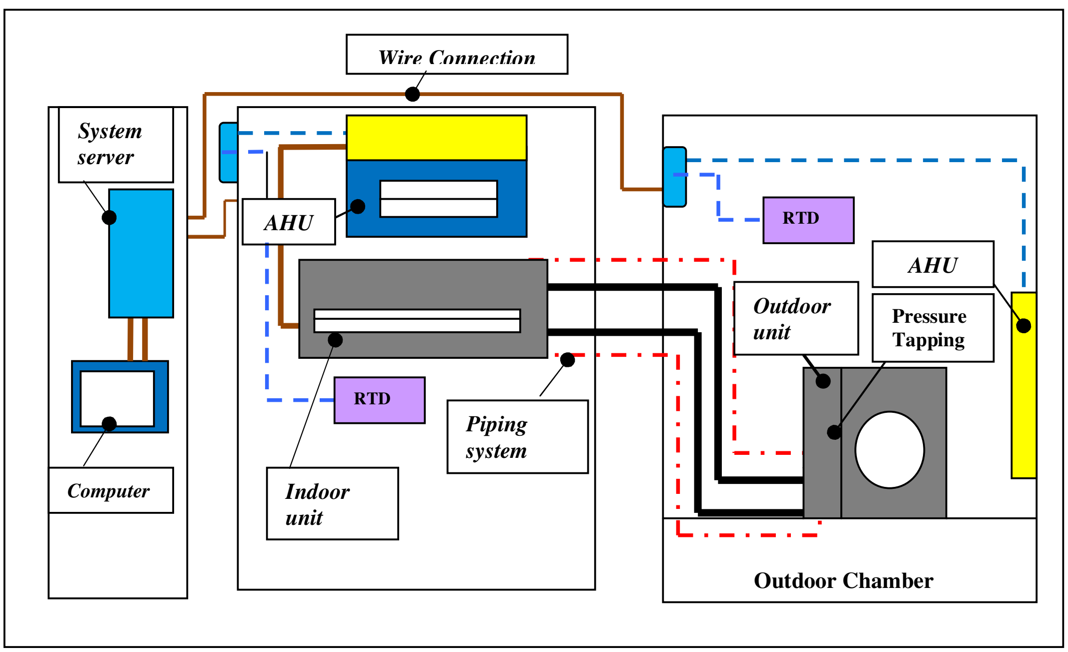

Figure 3.15: Overview of the Psychometric laboratory system

3.1.3.1 Psychometric Test Procedure

a) Power supply and computer system are switched on.

b) The air handling units (AHU) of indoor and outdoor are switched on and operating temperature of indoor dry bulb and wet bulb and outdoor dry bulb are set in the system.

c) The air-conditioner unit is switched on and the desired indoor temperature is set.

d) The pre-set ambient temperature is left to stabilize within two hours in indoor and outdoor chamber.

e) The temperature of dry bulb and wet bulb for indoor and outdoor is measured and control by resistance temperature detector (RTD).

f) Suction pressure is measured by pressure gauge meter as shown in Figure 3.16. The pressure tapping is connected perpendicular to the pipe wall as shown in Figure 3.17.

g) Suction temperature is measured during testing by use of thermocouples attached to the suction pipe.

h) Pipe stress is measured by piping stress equipment after the room chamber has reached the pre-set temperatures.

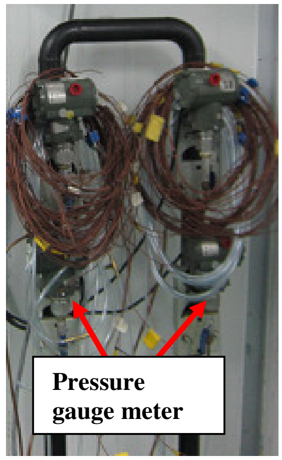

Figure 3.16: Pressure gauge meter used to measure pressure in pipe.

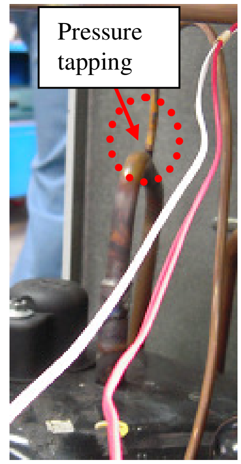

Figure 3.17: Pressure tapping on pipe

i. Parameters of Piping Stress to be Studied

In this present study, three parameters are selected which are pipe wall thickness, temperature difference between suction temperature and ambient temperature, and pipe length.

3.1.4.1 Pipe Wall Thickness

The outer diameter of the pipe is fixed at 15.88mm, and the inner diameter has two variables that are 13.48mm and 13.88mm. Therefore, the wall thicknesses in this current study are 1.0mm and 1.2mm. The test conditions to study the effect of pipe wall thickness on stress are listed in Table 3.1 below.

Table 3.1: Test condition to study the effect of pipe wall thickness on stress

Test no.	Wall thickness (mm)	Pipe length(mm)	Test Condition
1	1.0	1260	A
2	1.2	1260	A

3.1.4.2 Temperature between Pipe Suction Temperature and Ambient Temperature

Current work focuses on the effect of the temperature difference between pipe suction temperature and ambient temperature on the stress level of piping for air-conditioner is which operates in the air-conditioning allowable ambient temperature envelope A, B, C and D as shown in Figure 3.18. The temperature envelope is developed to ensure the product can operate safely in different climate. The suction temperature at the corresponding point A, B, C and D are measured during testing. The test matrix is displayed in table 3.2.

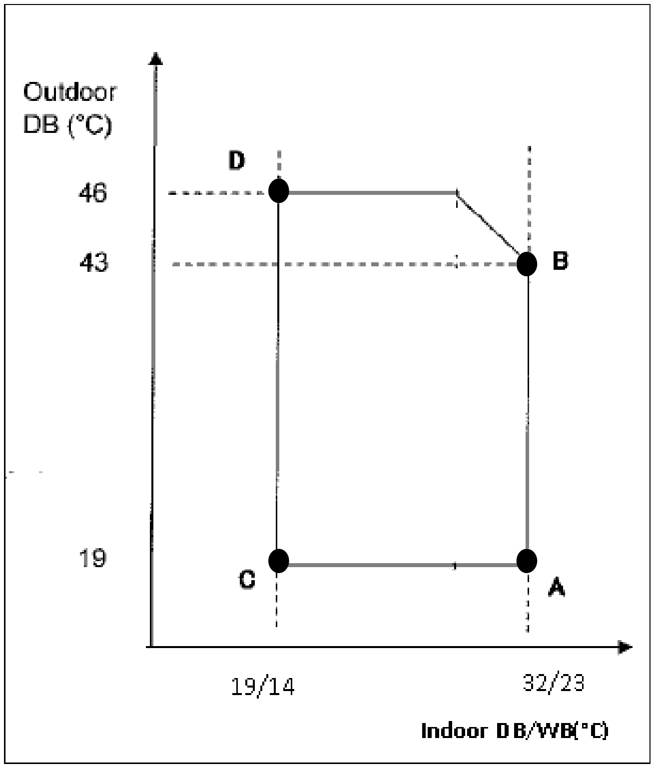

Figure 3.18: Temperature envelope for air-conditioner operation

Table 3.2: Test matrix to study the temperature difference between the ambient temperature and suction temperature on stress

Test no	Temperature envelop point(°C)	Pipe wall thickness (mm)	Pipe length(mm)
1	A	1.2	1260
2	B	1.2	1260
3	C	1.2	1260
4	D	1.2	1260

3.1.4.3 Pipe Length

The selected suction pipe lengths in this study are 1260mm and 986mm. The geometry of these pipes are shown in Figure 3.19 and Figure 3.20 respectively. The test conditions to study the effect of pipe length on pipe stress are listed in table 3.3.

Table 3.3: The test conditions to study the effect of pipe length on pipe stress

Test no	Pipe length(mm)	Wall thickness (mm)	Temperature envelop point(°C)
1	1260	1.2	A
2	986	1.2	A

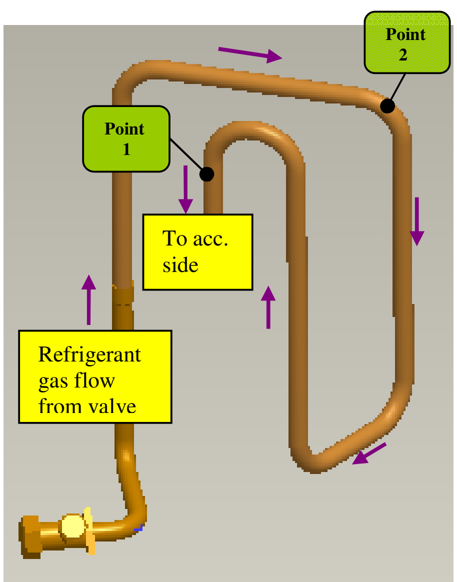

Figure 3.19: Pipe design with pipe length 1260mm.

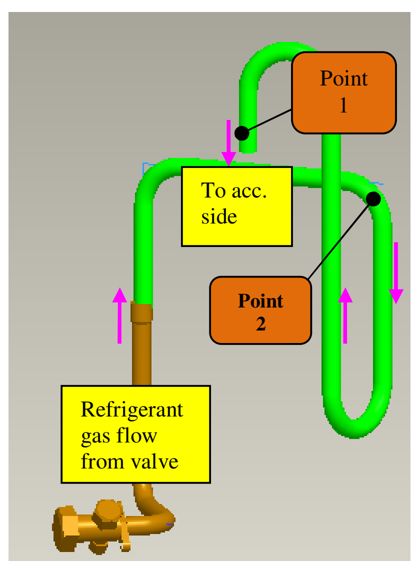

Figure 3.20: Pipe design with pipe length 986mm.

3.1.5 Overview of Experimental Methodology

The flow chart of the experimental methodology is presented in Figure 3.21 from suction pipe preparation to pipe stress measurement.

Start

↓

<u>Suction pipe preparation & piping assembly</u>

a. To prepare suction pipe as below:

1) With pipe length 1260mm and thickness 1.0mm
2) With pipe length 1260mm and thickness 1.2mm
3) With pipe length 986mm and thickness 1.2mm

b) Assemble each pipe in the outdoor unit for testing.

↓

<u>Bonding of strain gage on suction pipe</u>.

a. To decide on bonding position.
b. To clean the dust or rust on pipe by using sand paper.
c. To adhere the strain gage on pipe.
d. To press on the strain gage by finger using PS sheet and remove the sheet after the strain gage is securely bonded.
e. To use PE to cover the strain gage on pipe.

↓

<u>Laboratory setup</u>

a. Turn on the indoor and outdoor chamber system and set the operating ambient temperature according to the testing condition (for A, B, C and D).
b. Turn on the test unit and set the indoor temperature.

↓

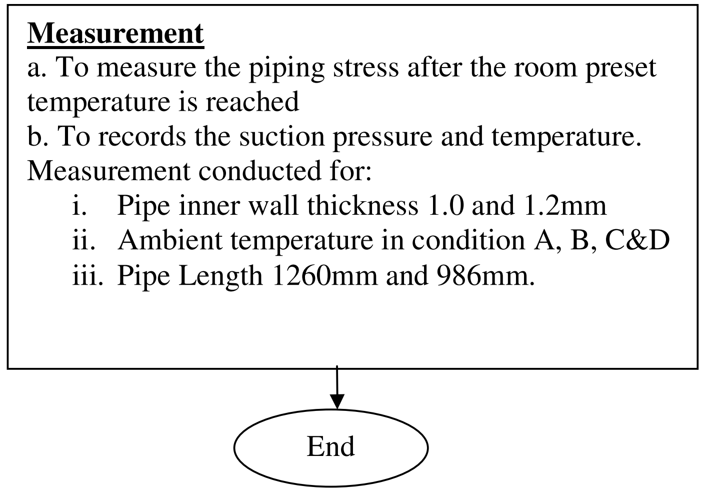

Figure 3.21: Flow chart for Piping Stress Test

3.2 Piping Stress Simulation

The inputs for the finite element modal analysis are excitation response from the accumulator and the stiffness of piping. These two inputs can be obtained by use of empirical method. The modal analysis is conducted by using ABAQUS software which is one of the well-known Finite Element (FE) analysis software.

3.2.1 Stiffness Testing

3.2.1.1 Stiffness Test Equipment

The objective of the stiffness test is to measure pipe stiffness by applying a load on the pipe and measuring the deflection of pipe. The measured value is then used to characterize the boundary conditions of both pipes end in the simulation. The equipments used in the piping stiffness test consist of push pull gauge (model ANF-100), stand support and ruler. The push pull gauge is used to apply required force on piping as shown in Figure 3.22. The stand support is used to hold the ruler as shown in Figure 3.23 and the ruler is used to measure the deflection of the piping.

Figure 3.22: Push pull gauge model ANF-100

Figure 3.23: Support stand

3.2.1.2 Stiffness Test Procedure

i. The ruler is held by the stand support and it is located in locations as shown in Figure 3.24 in order to measure the x, y, and z displacement when the ends of suction pipe is applied with a force of 30N.

ii. A 30N force is applied at both ends of suction pipe by using push pull gauge in x, y and z direction. The displacement in every direction is measured and recorded.

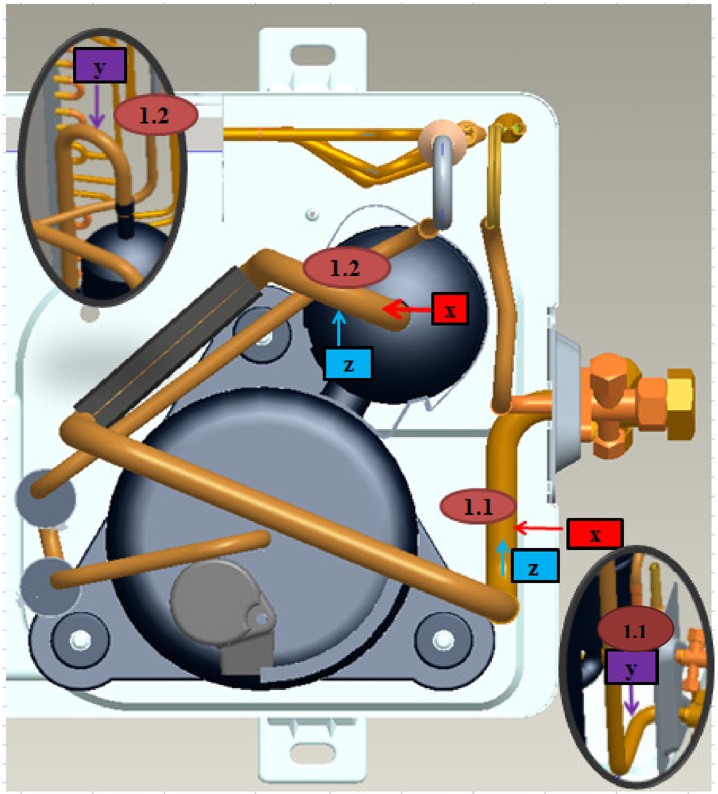

Figure 3.24: Force is applied at both ends of suction pipe in x, y and z direction.

3.2.1.3 Stiffness Results

The result of the stiffness test is displayed in table 3.4. The result shows that the highest stiffness is in y-direction and the lowest stiffness is in z-direction for both ends of suction pipe.

Table 3.4: Suction Pipe Stiffness

	Suction pipe end 1.1 (From pipe inlet – valve)			Suction pipe end 1.2 (From pipe outlet- accumulator)		
Direction	x	y	z	x	y	z
Displacement (mm)	0.60	0.50	1.0	2.50	0.50	6.0
Applied force (N)	30	30	30	30	30	30
Pipe stiffness (N/mm)	50	60	30	12	60	5

3.2.2 Excitation Response Test

3.2.2.1 Excitation Response Test Equipment

The excitation response test is used to measure the excitation response of the accumulator when the system is running. The measurement equipment consists of an exciter or source of vibration, accelerometer, NI signal acquisition analyzer and CPU with Lab View software. The exciter is used to apply the known input force to the structure (accumulator). Accelerometer functions as a sensor that receives the physical motion signal in x, y and z directions as shown in Figure 3.26. National Instruments NI PX1-1031 data acquisition analyzer is used to acquire and amplify the signals as shown in Figure 3.25. Lab View is an interface to interpret the measurement. The overview of the equipment setup is depicted in Figure 3.27.

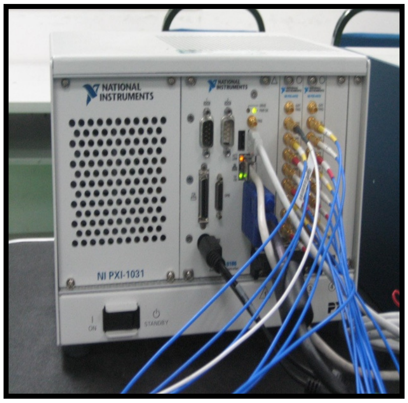

Figure 3.25: NI Data Acquisition Analyzer (PX1-1031)

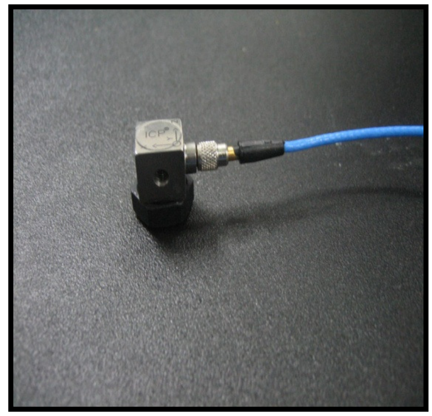

Figure 3.26: Accelerometer

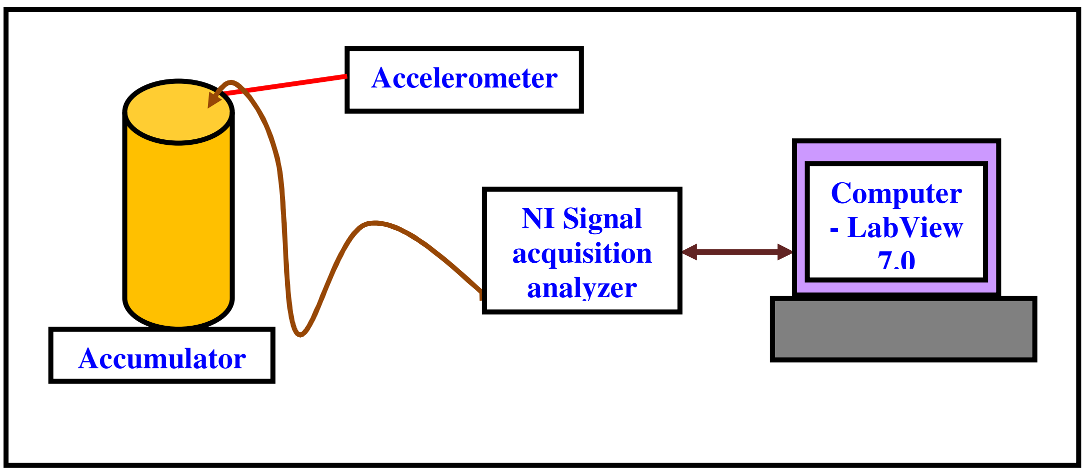

Figure 3.27: Overview of the equipment setup for excitation response test

3.2.2.2 Excitation Response Test Procedure

a) The accelerometer is positioned on top of accumulator as depicted in Figure 3.28. This location has the maximum excitation response because it is located in the direction of the refrigerant flow.

b) The CPU and NI Data Acquisition Analyzer are switched on.

c) The accelerometer is connected to NI Data Acquisition Analyzer.

d) The measured signal is then converted by LabView 7.0 into displacement in millimeter unit.

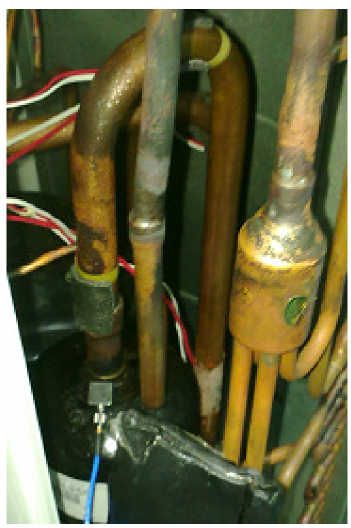

Figure 3.28: Accelerometers are attached on accumulator

3.2.2.3 Excitation Response Result

Figure 3.29 shows the excitation response imposed on the piping over time. This graph indicates that the high excitation response is in x-direction and y-direction whereas the excitation response in the z-direction is relatively low. This data is then used in simulation as amplitude.

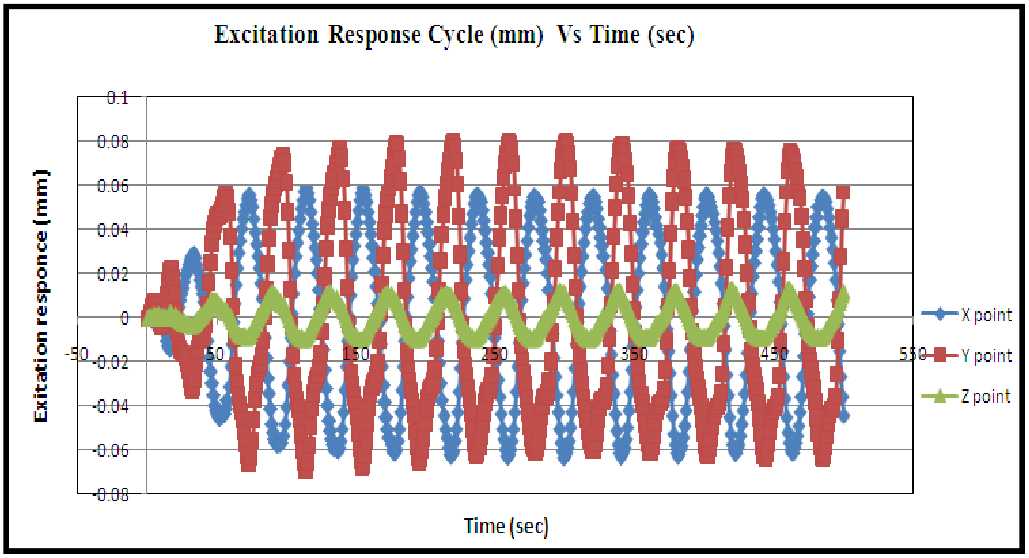

Figure 3.29: Excitation response cycle (in mm) Versus Time (in sec)

3.2.3 FINITE ELEMENT ANALYSIS

Finite element method enable to find the solution of a complicated problem by replacing it by simple approximation solution. In finite element analysis, the study object for a pipe used to find the stress distribution. Automatic mesh generation is used to involves the discretization of irregular domain into small, interconnected subregions called as finite element. The automatic mesh generation generates the location of nodes point and elements and provide element –node connectivity relationship.

After accomplished the meshing, the three-dimensional finite element (FE) model is performed by using ABAQUS software. The pipe material is set as copper phosphorous alloy (C1220T) and the material properties are defined as shown in Table 3.5.

The geometries are meshed with homogeneous shell section by using second order quadratic S8R and triangular STRI65 shell element as shown in Figure 3.30. S8R is uses as it is a common element in Abaqus and gain higher nodes and flexibility in shell modeling. STRI65 as a connection element to maintain the size of S8R in the meshing.

By assuming both ends of the pipe are elastic supports, the boundary condition at both ends of pipe can be represented as spring stiffness which can be measured by testing. The spring stiffness of pipe is defined at the ends of the pipe as depicted in Figure 3.31. All the defined springs are grounded to avoid free movement of the pipe at the both end. The setting of boundary condition for both end of pipe is shown in Figure 3.32. The translational DOFs at the pipe end connected to the accumulator are defined as k_x, k_y, k_z and the setting value is 12, 5 and 60N/mm respectively. The rotational DOFs at the pipe end connected to the accumulator is set as free of constraint because the rotating manner of the rotary compressor gives rotational effect to the pipe end connected to the accumulator. The translational DOFs at the pipe end connected to the valve are defined as kv_x, kv_y, kv_z and the setting value is 60, 50 and 30N/mm respectively. The rotational DOFs at the pipe end connected to the valve is set as fixed constraint because it is mounted to close-to-rigid valve. In this case, the pipe end connected to valve is restrained from moving in all rotational directions.

The tested excitation response data is used as dynamic loading in finite element analysis. The excitation response data is used to simulate the dynamic response of the pipe subjected to the excitation force from compressor.

At last, suction pressure of 51.4psig is applied to the inner wall of the pipe due to discharge pressure of refrigerant.

Table 3.5: Definition of material properties in finite element analysis

Description	Setting Value
Material	copper
Density(ton/mm^3)	8.946 x 10 -9
Mechanical elastic (MPa)	26,800 MPa
Poisson's ratio	0.343
Pipe thickness(mm)	1.2

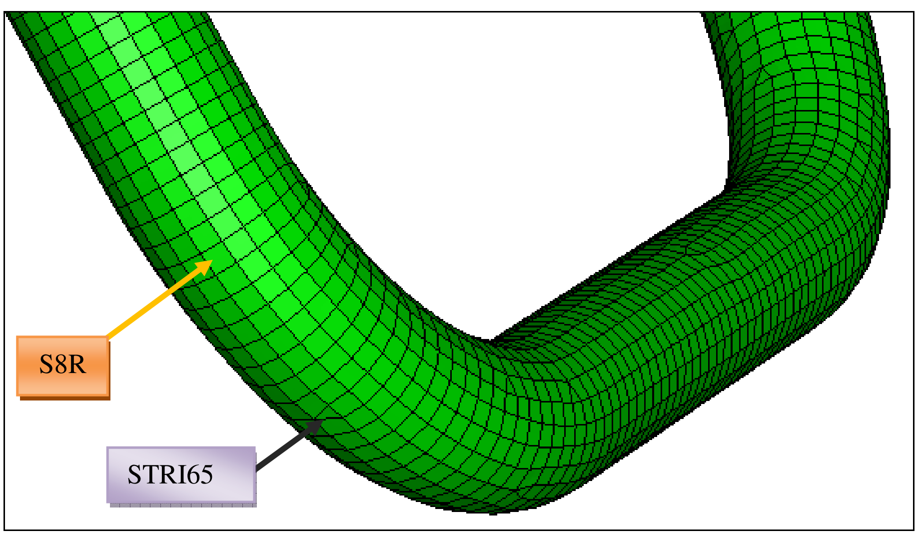

Figure 3.30: The meshes used are S8R and STRI65.

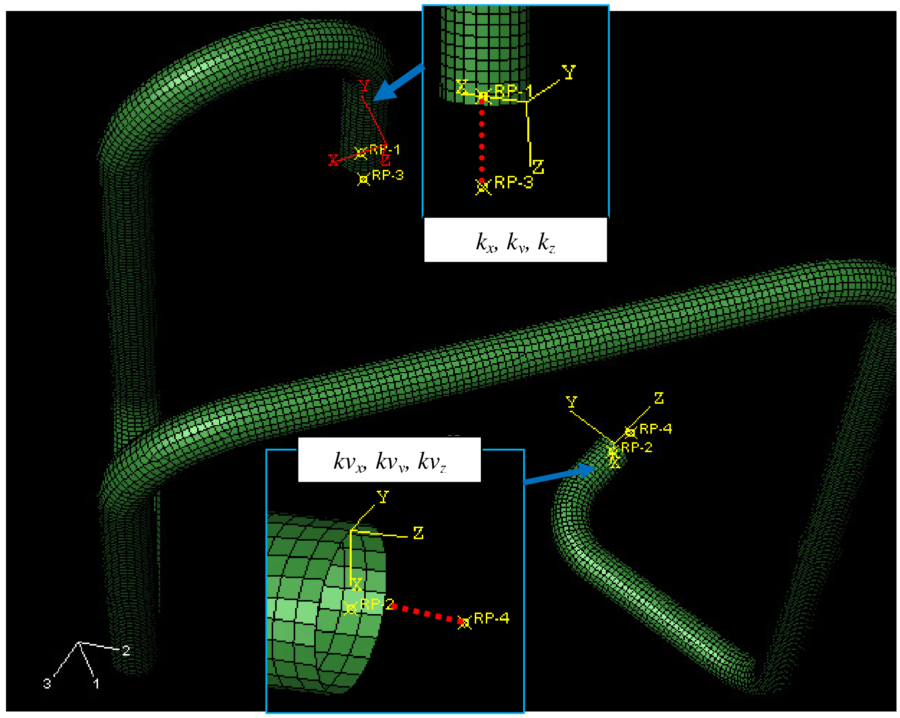

Figure 3.31: Translational DOF at both ends of pipe in simulation

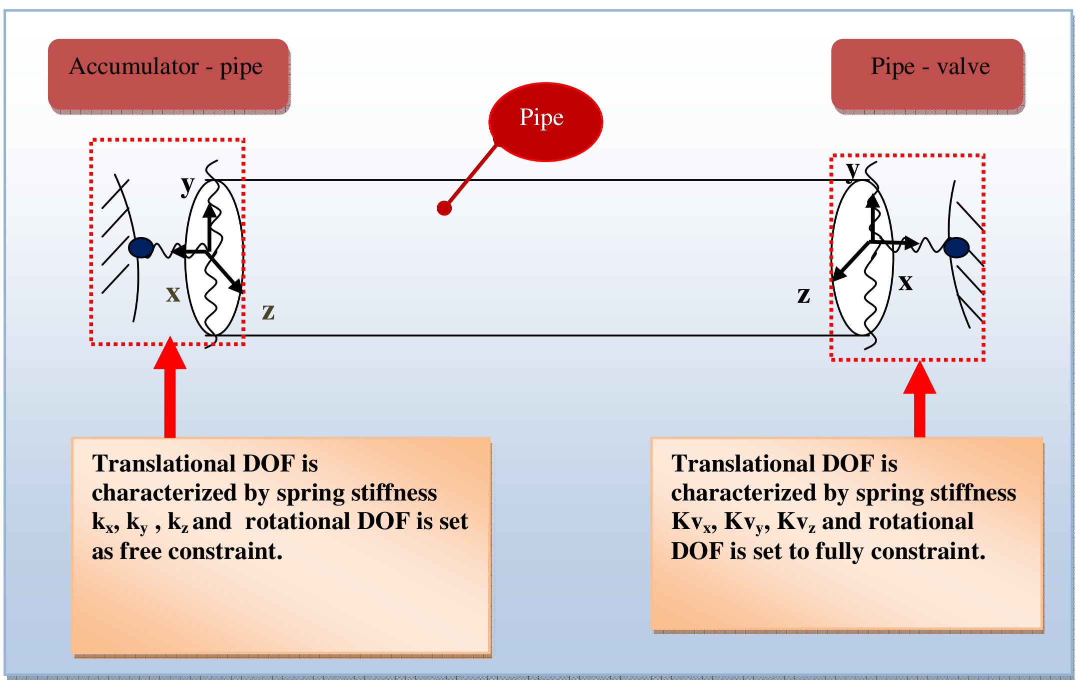

Figure 3.32: The setting of boundary condition at both ends of pipe.

3.2.4 Overview of the Finite Element Methodology

The flow chart of the finite element methodology is presented in Figure 3.33 starting from the stiffness test to the computational simulation.

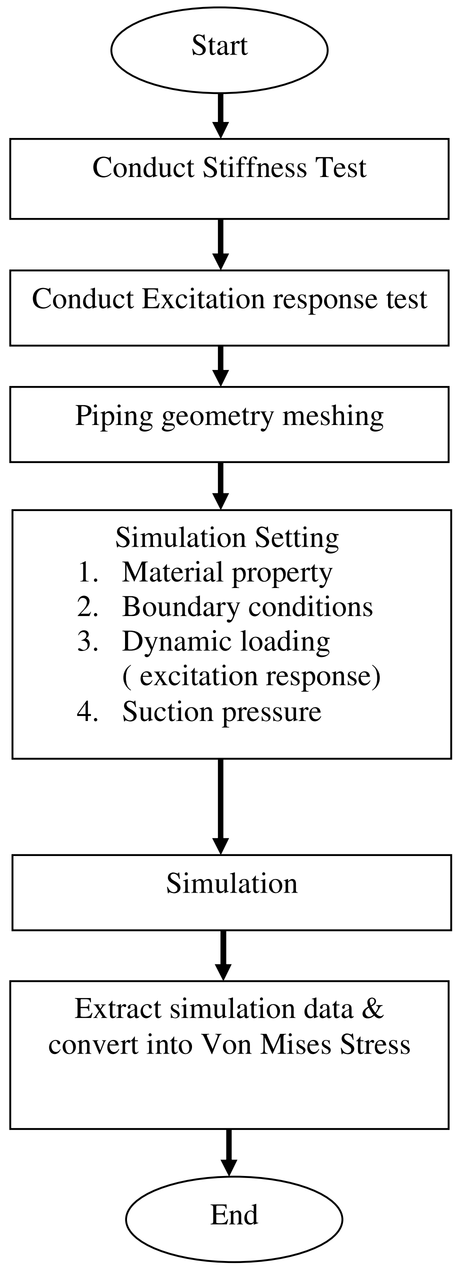

Figure 3.33: Flow chart of finite element methodology for pipe stress analysis

CHAPTER 4

RESULT AND DISCUSSION OF

EXPERIMENTAL TESTING AND SIMULATION

This chapter is consists of three parts. In part I present the empirical test result and discussion on the effect of pipe wall thickness, temperature difference between ambient temperature and suction temperature, and pipe length on pipe stress. In part II, the result and discussion regarding the comparison between simulation and testing output is presented. Lastly, prediction of the effect of pipe length of stress by finite element method is discussed.

4.1 Empirical Test Result and Discussion

4.1.1 Effect of Pipe Wall Thickness on Stress

The stress test result of test point 1 and 2 of two pipes with different wall thickness 1.0mm and 1.2mm is indicated in Table 4.1. The test results show that stress reduces with reduced wall thickness. The stress on stress test point 1 and point 2 reduces 10.8% and 13.6% respectively with the reduction of the wall thickness from 1.2mm to 1.0mm.

Table 4.1: Pipe stress test result of point 1 and point 2 for different wall thickness.

No.	Wall Thickness (mm)	Pipe Length (mm)	Suction Temperature (°C)	Point1 (MPa)	Point2 (MPa)
1	1.2	1260	-0.9	4.08	3.81
2	1.0	1260	-0.6	3.64	3.29

This phenomenon is explained by a simple piping system consists of two legs shown in Figure 4.1. For ease of understanding, the thermal contraction is assumed to occur in leg AB only. The vertical leg CB is a cantilever subject to thermal contraction Δ, and the contraction stress is calculated by dividing the moment, M with the section modulus, Z. The section modulus is defined as Z=I/r, which is directly proportional to the moment of inertia.

An increase in wall thickness increases the section modulus but also proportionally increases the bending moment, and this ends up with the same thermal stress as before, prior to the increase of the thickness. The thicker wall thickness does not reduce the thermal contraction stress, however; it would increase the moments and forces in the pipe. Although the thermal expansion stress is the same for leg CB with different wall thickness, the axial stress on the leg AB increases due to increased pulling force.

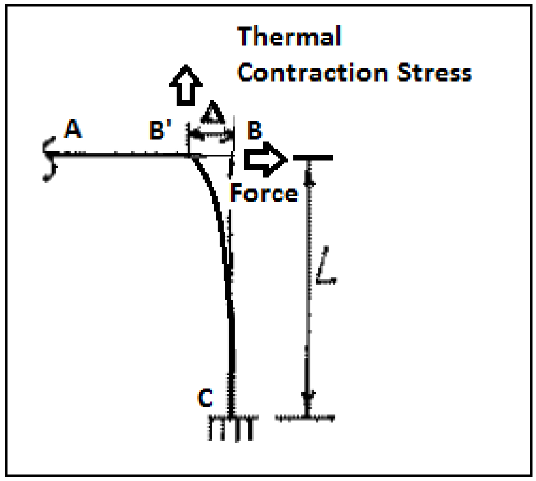

Figure 4.1: A simple pipe under contraction stress

4.1.2 Effect of Temperature Difference between Ambient Temperature and Pipe Suction Temperature on Stress

Table 4.2 indicates stress test result of test point 1 and point 2 for unit tested in condition A, B, C and D. The ambient temperature in which the indoor and outdoor unit operated would influence the suction pressure and suction temperature of refrigerant. Suction pressure and temperature are measured during testing in condition A, B, C, and D.

Table 4.2: Pipe stress test result of test point 1 and 2 for unit tested in condition A, B, C and D

		Condition A	Condition B	Condition C	Condition D
Indoor Dry bulb(°C)		32	32	19	19
Indoor Wet bulb(°C)		23	23	14	14
Outdoor Dry bulb(°C)		19	43	19	46
Discharge Pressure (psi)		199.3	342.3	181.4	340.0
Suction Pressure (psi)		55.1	65.2	43.8	51.2
Different Pressure.(psi)		144.2	277.1	137.6	288.8
Discharge Temp.(°C)		38.3	60.4	34.9	60.1
Suction Temp.(°C)		-0.9	3.14	-6.0	-3.0
Difference Temp.(°C)		39.2	57.26	40.9	63.1
Stress (in Von Mises ,MPa) Point 1		4.08	5.48	4.29	5.48
Stress (in Von Mises ,MPa) Point 2		3.81	3.98	3.81	4.08
Excitation of accumulator	x	0.124	0.192	0.124	0.209
	y	0.050	0.062	0.025	0.043
	z	0.113	0.250	0.133	0.254

The stress is the highest in condition D followed by condition B, C, and A. It is attributed to the fact that the thermal contraction is the greatest in condition D due to the highest temperature difference between the ambient temperature and suction temperature as governed by the equation 2.1 in section 2.1.1:

The thermal stress equation 2.5 as shown in section 2.1.1 that the thermal stress is directly proportional to the thermal contraction. Therefore, the consequence of higher temperature difference is an increase of thermal stress.

4.1.3 EFFECT OF PIPE LENGTH ON STRESS

The stress test results of two different pipe lengths which are 986mm and 1260mm tested in room ambient temperature with indoor dry bulb temperature 32°C and wet bulb temperature 23°C and outdoor dry bulb temperature 19°C are tabulated in table 4.3. Table 4.3 shows the pipe stress and pipe length for stress test point 1 and point 2. The result indicates that the piping stress reduces when the pipe length increases. It is for the reason that as the pipe length increases, it provides better flexibility for absorbing the thermal expansion of the pipe. The thermal stress is inversely proportional to the squared of pipe length based on equation 2.5 in section 2.1.1 Therefore, thermal stress reduces as the pipe length increases.

Table 4.3: Pipe stress result for different pipe length

	Original pipe length-1260mm	Revise pipe length-986mm	Difference (%)
Point 1 (in Von Mises , MPa)	4.47	6.43	43.8
Point 2 (in Von Mises ,MPa)	3.81	7.85	106

4.2 Comparison of Simulation and Experimental Result

Simulation is conducted in order to validate the simulated stress result with the empirical testing result for pipe thickness 1.2mm and pipe length 1260mm in test condition A. The simulated stress values are taken at locations as per stress test point 1 and 2 in experiment as shown in Figure 4.2. In Figure 4.2 has shown the simulation result for point 1 and point 2 are 3.74MPa and 4.38MPa respectively.

From the validation, the simulated stress result is within the range of 8 % to 15% in error compared to empirical data on test point 1 and 2 pipe. The error might be due to the setting of the boundary condition in the model. The setting of rotational stiffness at both pipes end might be one of the factors causing differences between the empirical testing result and simulated result. In the present FE model, the rotational stiffness is not defined based on testing input data due to limitation resources of testing equipment used to measure the rotational stiffness. Besides that, internal pressure is set to apply along the pipe wall in FE model instead of using fluid structure interaction methodology for simulation due to the limitation on the computational simulation tool and knowledge.

Table 4.4: Comparison of tested and simulated pipe stress result

	Tested (MPa)	Simulation (MPa)	Error (%)
Point 1	4.08	3.74	8.3
Point 2	3.81	4.38	15

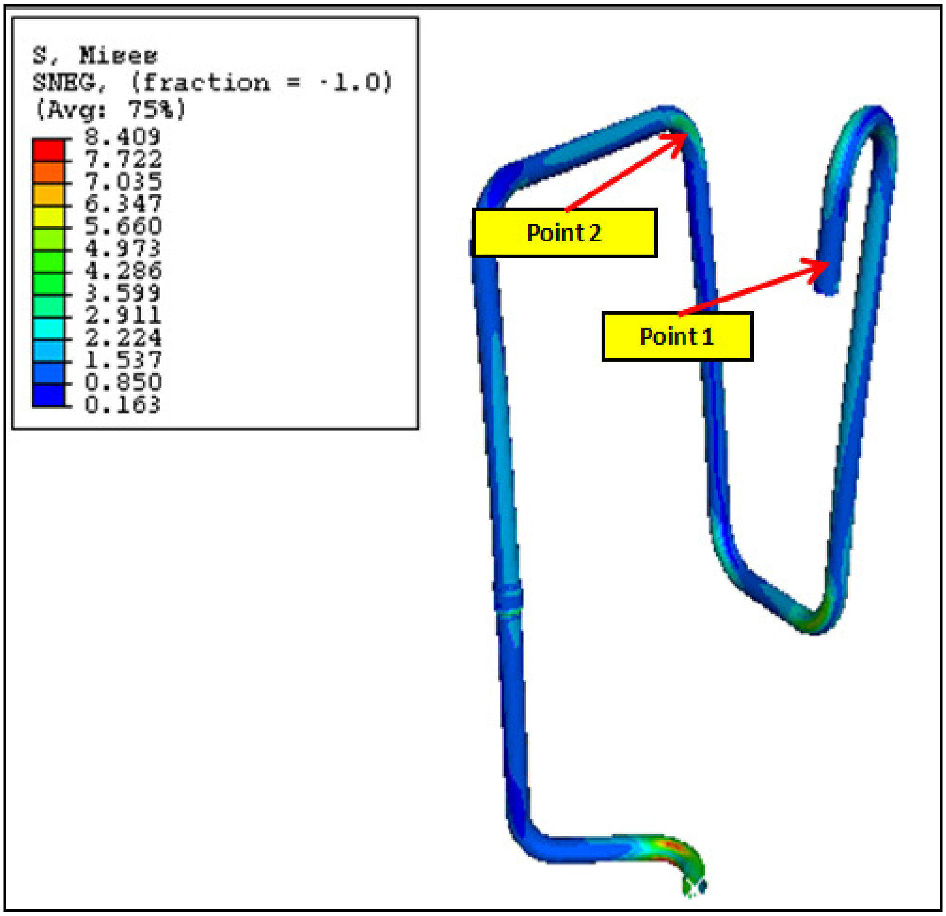

Figure 4.2: Location of test point 1 and point 2 on suction pipe with 1260.0mm length

4.3 Prediction of the Effect of Pipe Length on Stress by Finite Element Method

Finite element method is used to study the effect of pipe length on pipe stress. The study is conducted on five different pipe lengths which are 883mm, 1008mm, 1133mm, 1260mm and 1386mm. The boundary condition is set to be same as previous.

The simulated result is listed in Table 4.5 and the graph of pipe stress versus pipe length is shown in Figure 4.3. From the simulation, it can be observed that the pipe stress reduces with increasing pipe length. This trend is compatible with the test result and theory. Figure 4.4 is shown the pipe stress on shortest pipe length. Simulation shows that longer pipe which has highest distortion has lower stress due to higher flexibility.

Table 4.5: Simulated result at point 1 and 2 for different pipe length

Length of pipe(mm)	Simulated result (in Von Misses ,MPa)	
	Point 1	Point 2
1386	3.12	3.79
1260	3.74	4.38
1133	4.84	5.49
1008	5.63	6.02
883	6.44	6.86

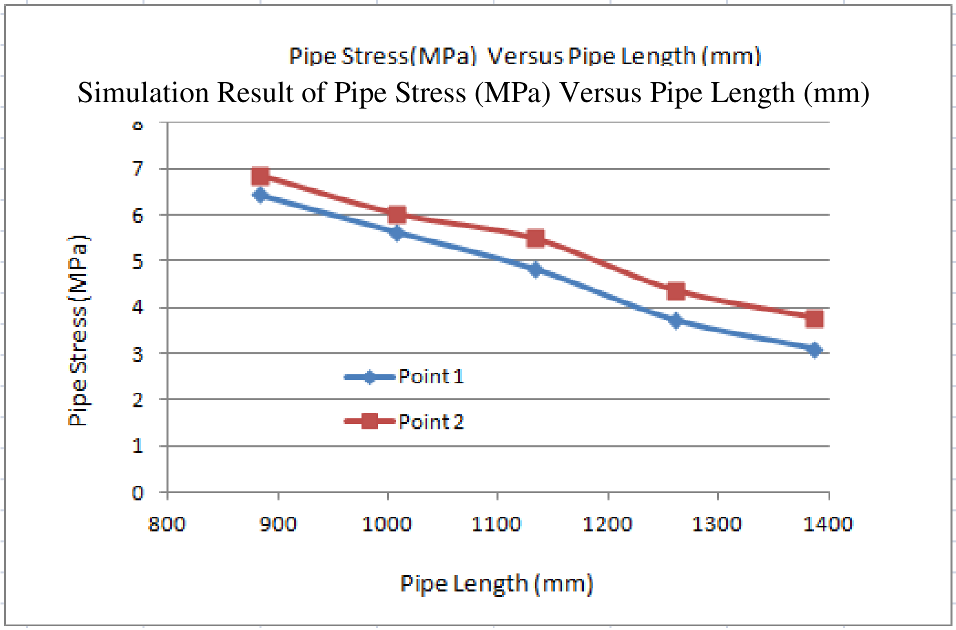

Figure 4.3: Effect of piping length on pipe stress

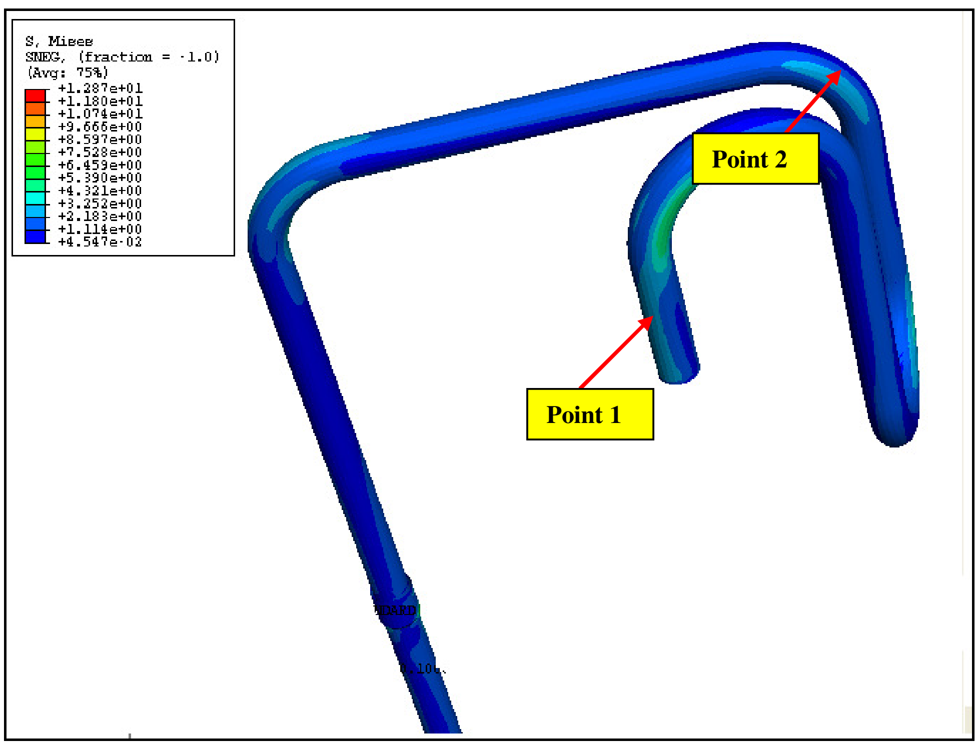

Figure 4.4: Location of test point 1 and point 2 on suction pipe on 883.0 mm pipe length

CHAPTER 5

CONCLUSION AND RECOMMENDATION

This chapter outlines the conclusion on the piping stress study and the recommendation to improve the piping stress study.

5.1 Conclusions

Some conclusions can be drawn from the results obtained from the experiments and simulation.

1) The thicker wall thickness does not reduce the thermal expansion or contraction stress. It only unfavorably increases the forces and moments in the pipe and consequently causes higher stress in adjacent pipe region.
2) An increase in the temperature difference between pipe suction temperature and ambient temperature increases the thermal contraction stress due to higher degree of thermal contraction.
3) Longer pipe length reduces the thermal contraction stress because it provides better flexibility for absorbing the thermal contraction.
4) The simulated stress result is below 16% in error compared to experimental test data. Although finite element analysis is unable to provide high accuracy on stress value; it still can be used to predict the effect of parameters on stress.

5) From the simulation, the pipe stress reduces with increasing pipe length. This result is compatible with test result and theory.
6) The parametric study on difference pipe wall thickness and pipe length enable to reduce the piping material used and decrease the material cost. For the parametric study of temperature difference on pipe between suction and ambient temperature can ensure the pipe to be operated in safety temperature envelope for good reliability.

5.2 Recommendations

For future study, the error of the simulation can be further reduced by using torsion formula to estimate the rotational stiffness along the axial pipe direction as an input to the simulation. The formula is given as follows:

$$\frac{T}{\theta} = \frac{GJ}{L}$$

Where

T/θ: Torque required inducing one degree of twist angle, i.e. rotational stiffness.

G: Shear modulus,

J: Polar moment of inertia

L: Length of pipe.

Fluid structure interaction (FSI) (Mackerle,1996) technique should be studied in simulation to investigate the effect of interaction of fluid and structure on pipe stress.

REFERENCES

Al-zaharnah,I.T., Hashmi,M.S. and Yilbas,B.(2000). “Conjugate heat transfer in fully developed laminar pipe flow and thermally induced stresses.”Journal of Computer Methods in Applied Mechanics and Engineering. Vol.190,p1091-1104.

Al-zaharnah,I.T., Hashmi,M.S. and Yilbas,B.(2001). “Pulsating flow in circular pipes – the analysis of thermal stresses.” Journal of Pressure Vessels and Piping. Vol.78, p567- 579.

Al-zaharnah,I.T., Hashmi,M.S. and Yilbas,B.(2001). “Thermal stresses in thick-walled pipes subjected to fully developed laminar flow.” Journal of Materials Processing Technologiy. Vol.118, p50- 57.

Bieniussa, K.W., Reck,H. and Yilbas, B.(1999). “Piping specific analysis due to thermal stratification.” Journal of Nuclear Engineering and design. Vol.190,p239-249.

Chattopadhyay, S. (2009). “ Structural evaluation of piping system subjected to thermal stratification.” Journal of Nuclear Engineering and Design. Vol.239, p2236 -2241.

Fonseca,E.M.M., de Melo, F.J.M.Q. and Oliveira, C.A.M. (2005). “The thermal and Mechanical behavior of structural steel piping systems.” Journal of Pressure Vessels and Piping. Vol.82, p145 -153.

Hibbitt,H.D.(1984). “Abaqus/EPGEN-A general purpose finite element code with emphasis on nonlinear application.” Journal of Nuclear Engineering and Design.Vol.77,p271-297.

Kandil,A., El-Kady, A.A. and El- Kafrawy, A. (1995). “Transient thermal stress analysis of thick-walled cyclinders.” Vol.37, p721 -732.

Kannappan,S. (1986). Introduction to pipe stress analysis. (1st Edition). United States of America: John Wiley & Sons.

Lin,Y.C., Xie, Y.J., Wang, X.H. and Luo, H. (2004). “Risk analysis of in-service pressure piping containing defects.” Journal of Pressure Vessels and Piping. Vol. 81, p13-20.

Lubis, A. and Boyle,T.J.(2004). “The pressure reduction effect in smooth piping elbows- revisited.” Journal of Pressure Vessels and Piping. Vol. 81, p119-125.

Mackerle,J.(1996). “Finite element in the analysis of pressure vessels and piping- a bibliography(1976-1996).” Journal of Pressure Vessels and Piping. Vol.69,p279-339.

Middleton, C.J., Timmins,R. and Townsend, R.D. (1996). “The integrity of materials in high temperature components; Performance and life assessment.” Vol.95,p33-57.

Nakamura,K. and Boyle, J.T. (1986). “The assessment of elastic follow-up in high temperature piping systems –Some example problems.” Journal of Pressure Vessels & Piping. Vol.29, p243-273.

Peng,L.C.(2011). “ Quick check on piping Flexibility.” Journal of Pressure Vessels & Piping. Vol.90,p101-109.

Rao,S.S. (2005). The finite element method in engineering. (4th Ed.). United Kingdom: Ed.Elsevier Butterworth –Heinemann.

Turner, M.J., Clough, R.W., Martin, H.C. and Topp, L.J. (1956). “ Stiffness and Deflection analysis of complex structures.” Journal of Aero Science. Vol.23, p805- 823.