CFD MODELING OF FLOW INDUCED VIBRATION MECHANISM IN SEA WATER PIPING ON A REGASIFICATION TERMINAL PLATFORM

AHMAD BIN MORTAZA

A project report submitted in partial fulfilment of
the requirements for the award of the degree of
Masters of Engineering (Mechanical)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

JUNE 2015

Dedicated to Ayah & Mak,

Professor Dato' Dr. Mortaza Mohamed & Datin Siti Aini Suriati Abdul Samad

ACKNOWLEDGEMENT

To my dear wife, Hasliza binti Alias, first and foremost, deserves my utmost gratitude for continually aiding me and pushing me across the finish line. Her love and care have given me strength to make all this possible – And to all our children, the source of my inspiration ~ Iman Hadfina, Iman Hasanah, Iman Humaira & Muhammad…

I wish to express my utmost sincere appreciation and full respect to my supervisor, Associate Professor Dr. Kahar bin Osman, for all the encouragement, sacrifice, guidance, critics and friendship. Without his continuous support and wisdom, this project report would not have been the same as presented here.

Ahmad bin Mortaza

ABSTRACT

Vibration is a common problem in water piping system, particularly when it involves multiple elbows. One of the sources of vibration is the turbulent flow in the system. The objective of this project paper is to investigate the mechanisms that leads to excessive vibration in the sea water discharge piping experienced on floating LNG platform. The model is developed to simulate the actual conditions experienced at the platform. The simulations were done using selected turbulent models, because the Reynolds's Number at any condition within the operating envelope is more than 4000. Upon completion of the simulations, focus on excitation at the wall is critical to be done. The major mechanism of vibration is pressure difference within the fluid flow. Thus, pressure and wall shear stress acting along the pipe wall need to be analyzed to pin point whether there is any mechanism at work which will induce vibration. The results show large pressure difference was observed between the inner and outer elbows thus prompting high inclination to vibration. It was observed that higher fluctuations of velocity and pressure near the wall as the Reynolds's number increases. The amplitudes of the fluctuations for both pressure and velocity were seen to be close to linear in nature.

ABSTRAK

Getaran adalah masalah biasa dalam sistem paip air, terutamanya apabila ia melibatkan pelbagai siku. Salah satu sumber getaran adalah aliran gelora di dalam sistem. Objektif kertas kerja ini adalah untuk menyiasat mekanisma yang membawa kepada gegaran berlebihan pada sistem perpaipan air laut di atas platform GAC terapung. Rekabentuk ini dibangunkan untuk membuat simulasi kepada keadaan sebenar yang dihadapi di atas platform. Simulasi ini dibuat dengan menggunakan model gelora, kerana nombor Reynolds's pada apa jua keadaan di dalam rangkuman operasi adalah lebih dari 4000. Selepas tamat simulasi, focus akan terarah kepada getaran pada dinding paip. Mekanisma getaran adalah perubahan tekanan di dalam aliran bendalir. Oleh yang demikian tekanan dan tekanan ricih yang bertindak pada dinding paip hendaklah dianalisa untuk menentukan sama ada terdapat mekanisma yang boleh menjadi punca getaran. Keputusan menunjukkan perbezaan tekanan besar diperhatikan antara siku dalaman dan luaran. Keadaan ini mendorong kecenderungan tinggi untuk getaran. Diperhatikan bahawa halaju dan tekanan berhampiran dinding berubah dengan lebih tinggi dengan peningkatan nombor Reynolds. Perubahan pada amplitude untuk kedua-dua tekanan dan halaju dilihat berkeadaan hampir linear.

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURES

LIST OF SYMBOLS

D	-	Diameter
r	-	Radius Of Curvature
g	-	Gravity = 9.81 m/s
ε	-	Turbulent Dissipation Rate
k	-	Turbulent Kinetic Energy,
l	-	Turbulent Length
υ	-	Dynamic Viscosity
P	-	Pressure
h	-	Height
N_R	-	Reynolds Number
V, U,v	-	Velocity
L	-	Characteristic Length
A	-	Cross Sectional Area
ρ_{sw}	-	Density

CHAPTER 1

INTRODUCTION

Oil, gas and energy are an essential driving force for any modern economy. The domestic oil and gas industry had contributed significantly to the growth of Malaysian economy, particularly in peninsula Malaysia. Natural Gas is a natural resource that eventually will deplete. There is a high likelihood that 1 to 2 percent per year on average decline is expected in the coming decades if significant efforts in upstream exploration, development and production were not being made. Due to this fact, and the growing demand for natural gas by the growing industrialization in the peninsula, resulted in the instability of the supply and demand from the national gas supply grid.

In order for Malaysia to sustain and increase the supply for the demand of natural gas, Malaysia regasification terminal project was officially endorsed on June 10th, 2010 under the 10th Malaysia Plan, under the oil, gas & energy NKEA. The main reason of commissioning the regasification terminal is to enable Malaysia to import Liquefied Natural Gas (LNG). This will boost natural gas supply in the national grid, and avoid reoccurrence of gas curtailment due to process upset in gas processing plants.

Malaysia is one of the largest global producer and exporter of LNG. LNG, as its name suggests, is a process where natural gas obtained from gas producing field, being transmitted to an LNG plant, where the natural gas in its gaseous state will be liquefied through thermodynamics processes, and exported via LNG vessels globally.

In 2013, Malaysia comes second after Qatar as the largest supplier of LNG in the world. Traditionally, these resources had been exported by Malaysia to countries far from gas producing regions, like Japan, Korea and Taiwan, contributing to the national GDP. In the near future LNG supply from local processing plant, can be used as leverage to accommodate Malaysian gas and energy demand.

1.1 Background of The Research

The objective of liquefaction of natural gas is only to aid in storage and transport. LNG takes approximately about $1/600^{th}$ the volume of natural gas in the gaseous state. Imported LNG arrived in a cryogenic state at approximately –160 °C, requires the process of regasification to its gaseous state before being transmitted into the natural gas grid for the use as fuel for power generation and city gas in peninsula Malaysia.

There are a number of LNG regasification technologies in the world today. This technology can be employed on land based plant or on offshore facility. Most offshore facility in the world today is retrofit of large LNG carriers. In Malaysia, LNG regasification facility is built on a floating platform. This regasification terminal is the first of its kind.

The technology employed in the local LNG regasification facility is Intermediate Fluid Vaporizer (IFV) technology, where liquid propane and seawater is employed as thermal energy source. IFV uses an intermediate heat transfer fluid to degasify LNG. Propane is used because it's inherent low flash points that are ideal for heat transfer and does not require a larger heat transfer surface contact area. This meant that the design does not require heavier and larger regasification system. This allows the package to be built on a floating platform.

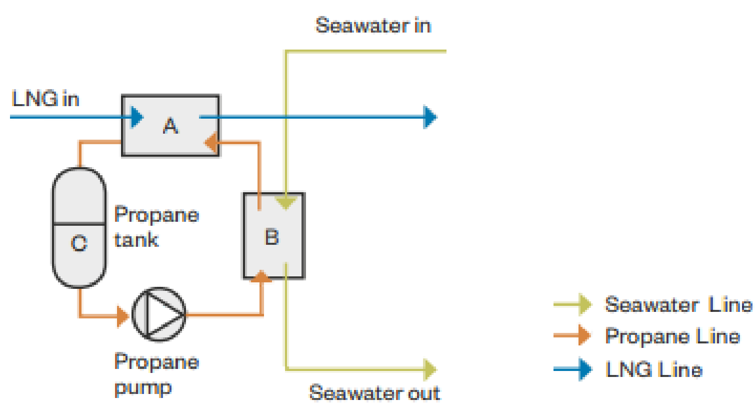

Figure 1.1 Layout of Intermediate Fluid Vaporizer (IFV) System

Figure 1.1 shows the layout of an IFV system, where A and B are LNG-Propane and Propane-Seawater heat exchangers respectively. Regasification of LNG using IFV system, consist of two stages. In the initial stage, LNG is heated by an intermediate fluid, which is propane in a heat exchanger, where the LNG absorbs the heat from the propane vapor through plate heat exchanger, and the LNG will convert to its gaseous state. Due to this heat loss, the propane will condense into its liquid vapor state.

In order to continuously maintain heating of propane, sea water is used as the thermal medium. The function of sea-water in this facility is only to heat up propane into its gaseous state. Propane in gas state is also used as the heating medium to bring the natural gas to ambient temperature required to send it out into the national grid for use.

In this case study, the area of concern is on the discharge piping of sea-water used for thermal expansion of propane. Upon operation, the regasification facility experiences unusual vibration in its sea-water discharge piping, reported particularly on the final bends. Vibration can occur due to flow turbulence within the piping system. In order to predict the mechanism which maybe the root cause of the reported occurrence, numerical approach will be applied on the discharged sea-water system.

1.2 Review of The Literature

It is a common occurrence, if a fluid flow through a piping system with multiple elbows at high flow rates will result in a condition called “hammering”. This can even be observed in household water system, where at times the sink faucet shakes violently when in use. This is due to the fact that turbulent flow in a piping system, which contains multiple elbows in its fitting, will result in velocity and pressure fluctuation throughout the system. The vibration, can be amplified enormously when dealing with large diameter, high flow rate piping system in the industry.

The pressure and velocity profile throughout a pipe is practically impossible to be measured physically. Further in the industry, where some point of pipes can be inaccessible by man (e.g. submerged, buried, etc.), further impairs our ability to determined and measure the required parameters. Thus, to indicate root cause of problem, mathematical approach can be engaged to indicate approximately the active parameters acting on any desired point within the system. This is a powerful tool for the industry to determine the root cause and fix the problem.

The use of navier-stokes equation to make approximation of parameters had been researched, study and applied widely. With the use of computational fluid dynamics (CFD), critical parameters can be calculated and the result can almost predict accurately the actual condition of the system.

Flow behavior in piping system using CFD, particularly in elbows has been studied by several researchers such as H.P. Rani et al., A. Ono et al. and T. Zhang et al.by means of CFD modeling and experimentation on test bench. The research is done exclusively for turbulent flow. The focus is done particularly in the measures of critical parameters variation at each point within the system, particularly at the region where fluid is entering and exiting the elbow. In all the research, it was observed that results obtained from CFD simulation is supports the result obtained from experimental data.

The study done by H.P. Rani et al. discussed the flow accelerated corrosion in 3D elbows based on actual field problem. The study was conducted on carbon steel piping, which bears relevance because similar piping material is used for consideration. Figure 1.2 shows the model and velocity distribution of the study. This research looks into flow through a three dimensional piping system with dual elbow at relatively close proximity fitting configuration. The study looks into the velocity profile of the second span vertical limb that plots velocity along the y-axis in the piping centerline. In this study, realizable k – ε model ANSYS Fluent® (2009) was used for evaluation. This model was selected because the study looks into the turbulence kinetic energy and dissipation rate, which can be determined by the stated model. It was found from the study that, the close proximity and high mass transfer coefficient rate will result in high intensity of vortex and fluctuation of velocity gradient and pressure. These conditions will increase flow accelerated corrosion. In order to minimize the effect of high mass transfer coefficient, it was proposed that vertical limb to be lengthened (i.e. increase the distance between elbows)[2]. The study shows, that in close proximity bends, under turbulent flow, fluctuation of velocity and pressure will occur along the system. Focus on fluctuation of velocity and pressure on the wall was not emphasized in this study.

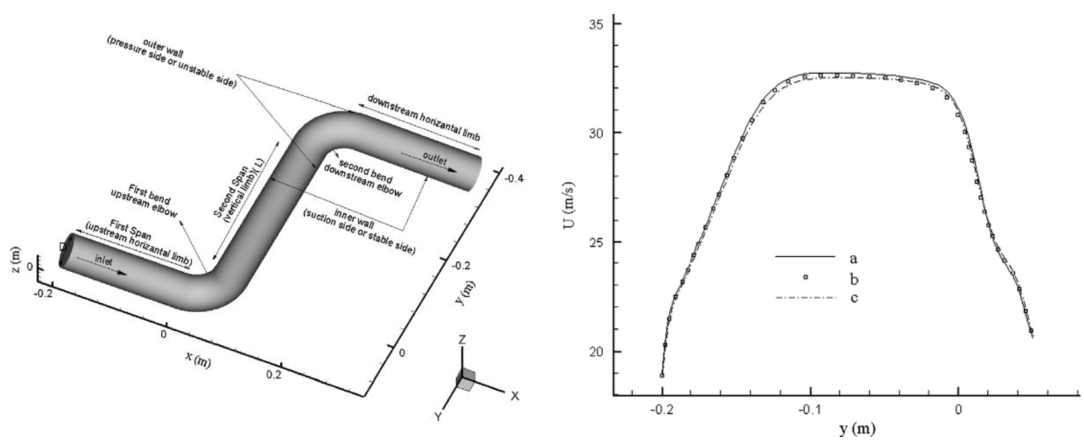

Figure 1.2 Computational Domains of Model and Velocity Magnitude Along y-axis[2]

The research done by A. Ono et al, discussed the influence of elbow curvature on flow structure at pipe elbow under high Reynolds number condition. The research includes experiment on a test loop that represents actual condition in the coolant piping design of Japan Sodium-cooled Fast Reactor (JSFR). The experiment is done on high inlet velocity of above 9 m/s, with Reynolds's Number of 4.2×10^7. The research focus on the determination of mechanism that contributes to the flow induced vibration, in an unsteady flow structure with emphasize on velocity profile. The experimentation involves the use of tracer particle method, where particle movement image was captured and mapped. The experiment was conducted on two sizes of resin elbow, which are short elbow and long elbow, where the ratio of curvature radius to the diameter, r/D, of the elbows are 1.0 and 1.5 respectively. It was found, from experiment that there is a region of low velocity in the inner side of the elbow. This pattern is clearly observable in a short elbow compared to the long elbow. From this experiment, the low velocity radically fluctuates in a 5 seconds time-frame, as shown in figure 1.3. Due to this phenomenon, vibration will occur in short elbow pipe induced by turbulent flow.

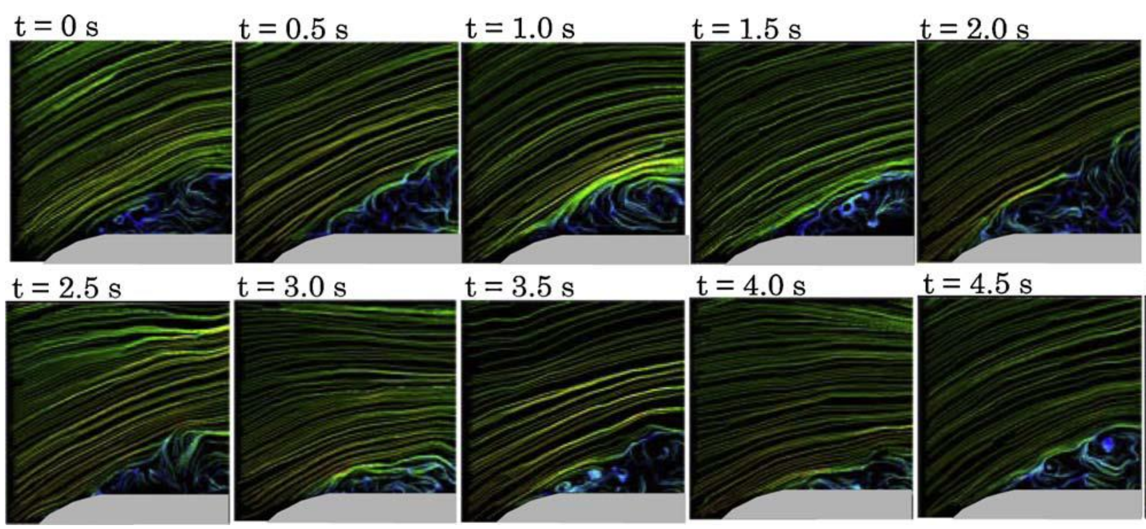

Figure 1.3 Velocity image with blue color is low velocity region[1]

T. Zhang et al. research on vibration induced by turbulent flow in 90° elbow. In this research, LES model was used to calculate velocity and pressure distribution throughout the elbow. The modeling contour result is as shown in figure 1.4, shows low pressure and low velocity region in the inner elbow. The finding supports the work by A. Ono et al. which suggest pressure difference phenomenon in pipe elbows due to turbulent flow. In this study, FSI was applied on the model to determine vibration characteristics of the flow. It was concluded that, mechanism of vibration is due to pressure and velocity difference topology within the turbulent model[8].

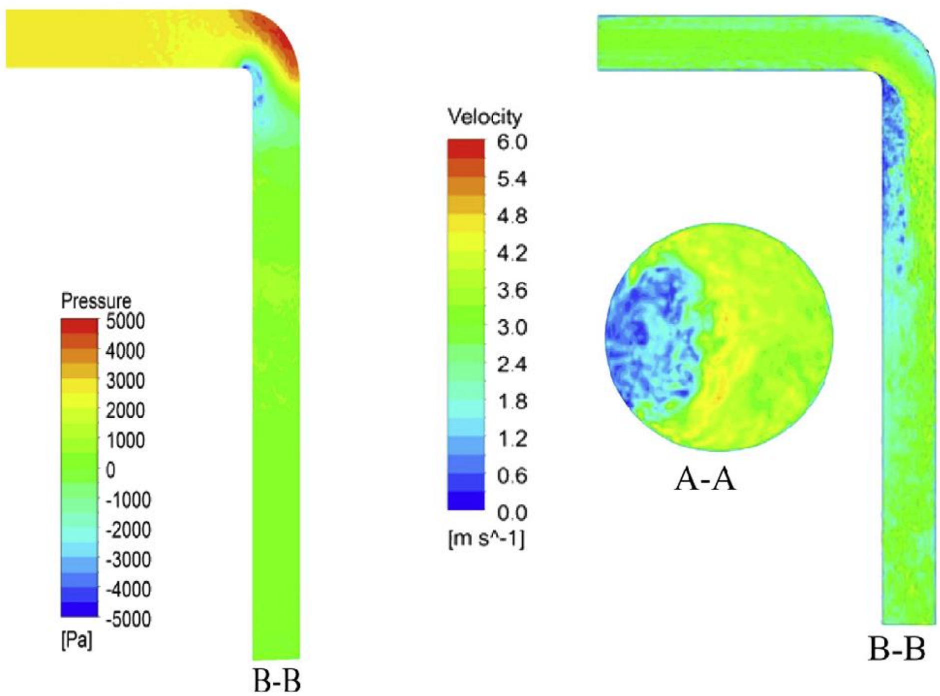

Figure 1.4 Cross section results by T Zhang et al. for Pressure and Velocity[8]

Turbulent flow results in fluid borne acoustic power in a 90° elbow's turbulent flow as found by S. A. Hambric et al. This research presents the findings on both fluid dynamic and vibration behavior of an elbow. The model is a long elbow with r/D value of 1.5. It simulates high velocity and Reynolds's Number flow. To model the flow behavior, the two equation SST k – ωmodelwas selected. The study looks into the wall pressure and the wall shear stress. Velocity distribution was not considered in this research. It was found that there is large pressure difference between inner and outer elbow due to the turbulent flow through the model. From the resulted shear stress contour, obtained it was seen that high shear stress concentrated at the inner wall of the elbow, while low values of shear stress observed concentrated at the outer elbow. Based on the obtained CFD model result combined with hydro-acoustic modeling program, the statistical energy analysis was obtained. It was suggested that, as the fluid flows through an elbow, strong pressure gradient developed which generate secondary flow field downstream of the elbow. This pressure fluctuation will excite the piping walls and emanate noise in the piping. Based on this analysis, it was concluded that pressure and wall shear stress are the mechanism that induces acoustic energy, leading to vibration[7].

HidemasaYamano et al. evaluate flow induced vibration methodology for large diameter piping elbow. Similarly, the study is on turbulent flow, but with different flow rate (i.e. the author classify by rated power, MWe). Each rated power has different velocity and Reynolds's Number, and these values will be higher with higher rated power classification. Pressure is examined by means of power spectrum densities (PSD), where it has a correlation[3].

YuzuruEguchi et al. research on high Reynolds's Number flow for short elbow pipe. This study favors LES, where flow behavior was modeled using Smagorinsky-lilly, WALE and DNS and was compared. It was demonstrated that higher separation region produces higher pressure fluctuations. But, the focus on the study is more concentrated towards the boundary layer, where turbulent fluid flowresults in varying separation region. Nonetheless, in this study it also shows turbulent flow in pipe elbow produces pressure fluctuation[10].

While there are many research on turbulent flow through pipe has extensively done, none have attemptedmodel of multiple bend piping system, and study the behavior of the flow.

1.3 Statement of Hypothesis

The primary objective of this study is to focus on identifying parameter variation using numerical approximation using Navier-Stokes equation and describe discharge flow behavior of sea-water used as a thermal energy source in an LNG regasification facility, with known fluid volume, inlet velocity and pressure at the outlet.

Turbulent flow in a system generates flow fields that result in velocity and pressure fluctuation throughout the system. Based on numerous researches, these variations will result enough excitation that can generate energy to emanate flexing on pipe surface.

Comparison will be made between instances of typical, high and low operation demand. Based on the primary objective, it is to be determined the critical parameters that affect the pipe walls, particularly on the bends.

1.4 Problem Statement and Limitation

Noise and vibration in piping elbow, is always associated with turbulence flow generated by pressure drop. Generally in the industry solid mild steel pipe is used to transmit process fluids. Fluid flow behavior inside a solid pipe or duct is difficult to measure and quantify which will always become a stumbling block in a Root Cause Analysis processes.

In the industry, vibration is a common norm in any system, which is managed to avoid catastrophic failures. There is a set standard of critical limits introduced to govern allowable vibration condition. If these limits were exceeded, review in the system's design need to be undertaken, repair and rerouting of piping, bracing of piping structures, and installation of fittings need to be done, which at most of the circumstances will be time consuming and costly.

At times due to design constraints, any piping system can have complicated routing. In this facility, a simple sea-water discharge piping from the header to the underwater sea level, have eight 90° tight elbows fabricated relatively in close proximity. Due to these conditions, generation of secondary flow field is expected to be compounded by the numbers of elbows. This is consistent with what was reported by the facility of observable alarming vibration condition and unusual gravel sounds within the piping elbows.

1.5 Scope of Research

Noise and vibration induced by developed turbulent flow in piping with elbow are studied by several researchers such as S.A. Hambric [7], HidemasaYamano [3]and J.L. Riverin [4]. Most of the research is on turbulent fluid flow in a single elbow and dual elbow, with water as working fluid. However, the studies of sea water flow, in multiple elbows in three directions have yet to be explored.

In order to determine root cause of unusual noise and vibration resulted from flow in a piping system, it is crucial that modeling of the piping system is done in order to determine flow behavior inside the piping system. Based on the modeling, the pressure and velocity distribution within the piping system can be numerically mapped with known boundary condition of the system. CAD model of the piping system based on figure 1.5 can be completed using any modeling software. In this research, modeling was done using SolidWorks™, and exported into Ansys™ Fluent. With the obtained model, meshing is done for mathematical iteration to indicate essential values on each node along the pipeline.

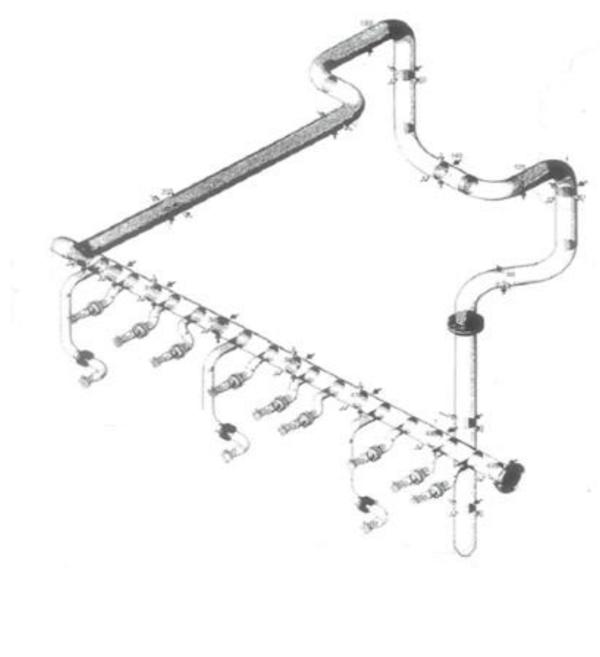

Figure 1.5 Model of Sea Water Heat Exchanger Discharge Piping

1.6 Theoretical Framework

The framework of this research is represented by figure 1.6. The critical steps will involve the development of mathematical and computational of a 3-D model for the fluid flow in a multiple elbow piping.

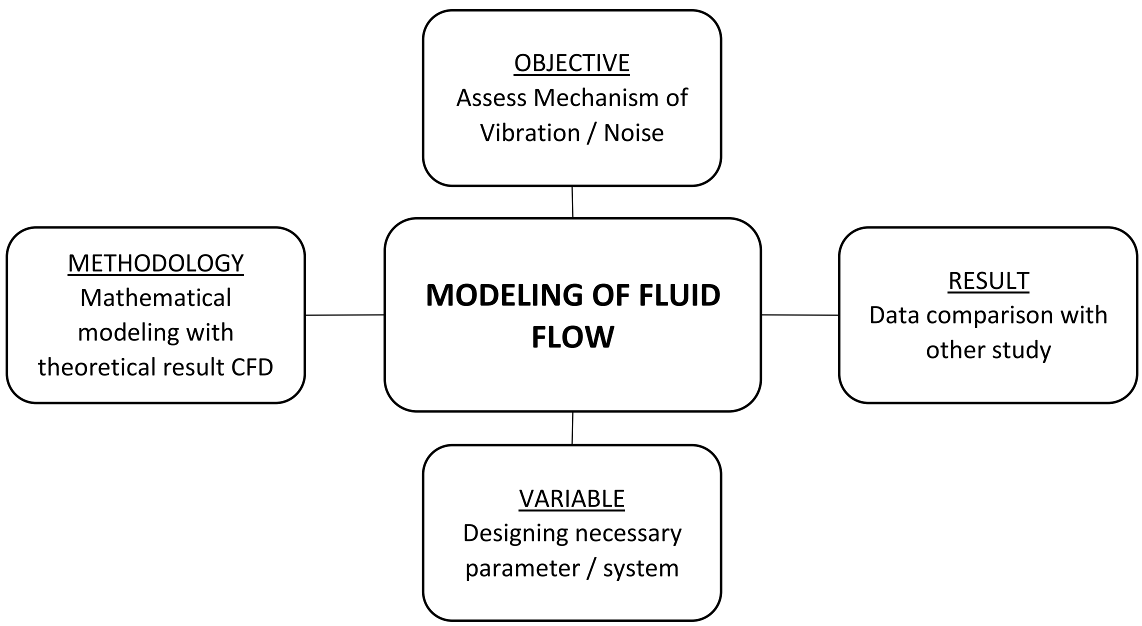

Figure 1.6 Theoretical Frameworks

REFERENCES

[1] A. Ono, N. Kimura, H. Kamide, A. Tobita (2011). Influence of elbow curvature on flow structure at elbow outlet under high Reynolds number condition. *Nuclear Engineering and Design 241*, 4409–4419

[2] H.P. Rani, T. Divya, R.R. Sahaya, VivekanandKain, D.K. Barua (2014). CFD study of flow accelerated corrosion in 3D elbows. *Annals of Nuclear Energy 69*, 344–351

[3] HidemasaYamano, Masa-aki Tanaka, Nobuyuki Kimura, Hiroyuki Ohshima, Hideki Kamide, Osamu Watanabe (2011) Development of flow-induced vibration evaluation methodology for large-diameter piping with elbow in Japan sodium-cooled fast reactor. *Nuclear Engineering and Design 241*, 4464–4475

[4] J.L. Riverin, E. de Langre, M.J. Pettigrew (2006) Fluctuating forces caused by internal two-phase flow on bends and tees. *Journal of Sound and Vibration 298*, 1088–1098

[5] Mostafa H. Sharqawy, John H. Lienhard V, Syed M. Zubair (2009) Thermophysical properties of seawater: a review of existing correlations and data.*Desalination and Water Treatment 16*, 354–380

[6] Pope, S. B., (2003).*Turbulent Flows*. Cambridge: Cambridge University Press

[7] S.A. Hambric, D.A.Boger, J.B.Fahnline, R.L.Campbell (2010) Structure- and fluid-borne acoustic power sources induced by turbulent flow in 90° piping elbows. *Journal ofFluidsandStructures26*, 121–147

[8] T. Zhang, Y.O. Zhang, H. Ouyang (2014). Structural vibration and fluid-borne noise induced by turbulent flow through a 90° piping elbow with/without a guide vane.*International Journal of Pressure Vessels and Piping*, 1 – 12

[9] Versteeg, H.K., and Malalasekera, W., (2007). *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, 2nd ed., Pearson Education Limited, London, England.

[10] YuzuruEguchi, Takahiro Murakami, Masaaki Tanaka, HidemasaYamano (2010) A finite element LES for high-Re flow in a short-elbow pipe with undisturbed inlet velocity. *Nuclear Engineering and Design 241*, 4368–4378