THE ANALYSIS OF FLUID STRUCTURE INTERACTION FOR SUBMERSIBLE WATER-HYDRAULIC MANIPULATOR USING COMPUTATIONAL FLUID DYNAMICS

MUHAMMAD ZUL FIKRI BIN SUHAIMI

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

© Universiti Teknikal Malaysia Melaka

SUPERVISOR DECLARATION

"I hereby declare that I have read this thesis and in my opinion this thesis is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering (Thermal-Fluids)"

Signiture : ....

Supervisor : DR. AHMAD ANAS BIN YUSOF

Date : 29 JUNE 2015

© Universiti Teknikal Malaysia Melaka

THE ANALYSIS OF FLUID STRUCTURE INTERACTION FOR SUBMERSIBLE WATER-HYDRAULIC MANIPULATOR USING COMPUTATIONAL FLUID DYNAMICS

MUHAMMAD ZUL FIKRI BIN SUHAIMI

This thesis is submitted in partial fulfilment of the requirement for the degree of Bachelor Mechanical Engineering with Honours (Thermal-Fluid)

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

JUNE 2015

© Universiti Teknikal Malaysia Melaka

DECLARATION

"I hereby declared that the work in this thesis is my own except for summaries quotation which have been duly acknowledged"

Signature	: ………………………..
Author	: MUHAMMAD ZUL FIKRI BIN SUHAIMI
Date	: 29 JUNE 2015

© Universiti Teknikal Malaysia Melaka

This report is dedicated to my family. Thank you for your continuous support during my vital educational years. Without their patience, understanding and most of all love, the completion of this work would not have been possible.

To my beloved parents,

Sharifah Binti Mohd Suhaimin@Abdul Rahman

&

Suhaimi Bin Yusoff

My siblings,

Muhammad Amiruddin Bin Suhaimi

Mohammad Faizal Bin Suhaimi

Siti Nur Hafizah Binti Suhaimi

Muhammad Hazim Bin Suhaimi

Siti Nur Ain Aini Binti Suhaimi

© Universiti Teknikal Malaysia Melaka

ACKNOWLEDGEMENT

"In The Name Of Allah, The Merciful, The Beneficent"

Glory to Allah S.W.T. the most gracious and most merciful. All the worship belongs to only Allah. We seek refuge with Allah from the wickedness within evil and until I have done the project. I also praised to Allah S.W.T for giving us courage, time, and knowledge in doing this report for Bachelor Degree Project.

Alhamdulillah, the report can be submitted on the due date. The successful of this project is because of the encouragement and support of many people. I wish to express my gratitude to my supervisor, Dr. Ahmad Anas Bin Yusof who was abundantly helpful and offered invaluable assistance, support and guidance and also deepest appreciation to co-supervisor, Engr. Mohd Noor Asril Bin Saadun. Without continued support and interest on the project.

Last but not least, I would like to thanks to all my friends who always been here to give a motivation. I would also like to convey thanks to the Faculty of Mechanical Engineering for providing the financial and laboratory facilities.

© Universiti Teknikal Malaysia Melaka

ABSTRACT

The advance in technology have brought the robotic industries into a new level. Starting by human divers to the uses of robotic for underwater cases, have brought upon the human exploration underwater and closer to nature. These technologies undergoes several changes from simple shape like torpedo of Autonomous Underwater Vehicle (AUV) to the Remotely Operated Vehicles (ROV) that appeared since 1900's been used for wide range of task today. This report will focus on one important part from the ROV which is the manipulator arm. The manipulator arm is analyzed using the Computer-Aided Engineering (CAE) tool which is the Computational Fluid Dynamics (CFD). This CFD analysis will be applied to the submersible water-hydraulic manipulator. Prototype from previous studies is taken and drawn as the model for analysis in the CFD. The model simulation will be designed in a water region with laminar flow condition considering the inlet velocity is 0.1 m/s. The velocity and pressure distribution contour is obtained considering initial, extend and blunt body of the model. The boundary for the model is different for each cases so that simulation can be done. Number of nodes that generated from each cases is different and been determined by altering certain value in the mesh detail.

© Universiti Teknikal Malaysia Melaka

ABSTRAK

Kemajuan dalam teknologi telah membawa industri robotik ke tahap yang baru. Bermula dengan penyelam manusia sehingga penggunaan robot bagi kes-kes di bawah air, telah membawa kepada penerokaan manusia di bawah air dan lebih dekat dengan alam semula jadi. Teknologi ini mengalami beberapa perubahan dari bentuk yang mudah seperti torpedo salah satu dari Kenderaan Dalam Air Autonomi (AUV) kepada Kenderaan Bawah Air Kawalan Operasi (ROV) yang telah muncul sejak tahun 1900-an digunakan untuk pelbagai tugas hari ini. Laporan ini memberi tumpuan kepada satu bahagian penting dari ROV iaitu lengan penggerak. Lengan penggerak dianalisis menggunakan salah satu alat Kejuruteraan Berbantu Komputer (CAE) iaitu Pengkomputeran Dinamik Bendalir (CFD). Analisis CFD akan digunakan untuk hidraulik air lengan penggerak yang boleh ditenggelamkan. Prototaip daripada kajian sebelum ini diambil dan disediakan sebagai model untuk analisis dalam CFD. Simulasi model akan direka di sebuah kawasan air dengan keadaan aliran laminar dan halaju masuk ialah 0.1 m/s. Kontur pembahagian taburan halaju dan tekanan diperolehi dengan mempertimbangkan kedudukan model iaitu kedudukan awal, pemanjangan dan tumpul. Had atau sempadan untuk model adalah berbeza bagi setiap keadaan supaya simulasi boleh dijalankan. Nilai nod yang dihasilkan bagi setiap keadaan berbeza dan ditentukan dengan merubah nilai tertentu pada bahagian perincian mesh.

© Universiti Teknikal Malaysia Melaka

TABLE OF CONTENTS

© Universiti Teknikal Malaysia Melaka

© Universiti Teknikal Malaysia Melaka

© Universiti Teknikal Malaysia Melaka

LIST OF FIGURES

© Universiti Teknikal Malaysia Melaka

© Universiti Teknikal Malaysia Melaka

LIST OF TABLES

© Universiti Teknikal Malaysia Melaka

LIST OF ABBREVIATION

ANSYS	=	Analysis System
CAD	=	Computer Aided Drawing
CAE	=	Computer-Aided Engineering
CFD	=	Computational Fluid Dynamics
DOF	=	Degree Of Freedom
ROV	=	Remotely Operated Vehicle
TOME	=	Task Oriented Manipulability Ellipsoid (TOME)
TOMM	=	Task-Oriented Manipulability Measure (TOMM)

© Universiti Teknikal Malaysia Melaka

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

According Vehicle teleoperation first appeared in the early 1900’s, but it was not until the 1970’s that systems became widely used [1]. Back then, the underwater ROV is mainly used in oil industry while manned submersible vehicle is the preferred tools for underwater scientific studies. As the technology developed, its functions have expended to be used in both industries and scientific studies. ROV is defined as the underwater robot that allows the vehicle’s operator to remain in contented environment while the piloted robot performs the work underwater.

The manipulator arm design has to be optimized to increase its working efficiency. Computer-Aided Engineering (CAE) is the most suitable technology to perform optimization. CAE is the use of information technology to support engineers in tasks such as analysis, simulation, design, manufacture, planning, diagnosis, and repair. Software tools that have been developed to support these activities are considered as CAE tools. CAE tools are being used to analyze the robustness and performance of components and assemblies. The term encompasses simulation, validation, and optimization of products and manufacturing tools.

To perform optimization on the manipulator arm, CAE tool that required is CFD approach. According to Versteeg and Malalasekera [2], computational fluid dynamic or CFD, is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by mean of computer based

© Universiti Teknikal Malaysia Melaka

simulation. This analysis technique is widely used that range of industrial and non-industrial application areas.

This project studies about design and analysis of the submersible hydraulic manipulator which shows the capability of the manipulator arm design and its features according to the analysis that will be performed. This computer based simulation is important because it can simulate the fluid that pass through the manipulator arm at different angle. Furthermore, simulation and analysis of the fluid flow in term of velocity and pressure distribution can be implemented into this project to detect any excessive pressure build that may happen to the manipulator arm that analytically based from underwater characteristic.

1.2 PROBLEM STATEMENT

Underwater robots play a vital role not only for underwater research but also for underwater tasks such as construction and investigation. A part from that, the manipulator arm is a major component for underwater intervention and manipulation. This component is most likely dealing with the interaction of the surrounding fluid which can cause stress to the components. There is a high tendency that the fluid interaction happens to the manipulator arm may cause damage to the component and not functioning properly during underwater tasks. So, the manipulator arm will be analyze in term of performance and its interaction underwater and detect the crucial part of the manipulator that will have critical stresses during underwater task by Computational Fluid Dynamics (CFD) analysis.

© Universiti Teknikal Malaysia Melaka

1.3 PROJECT OBJECTIVE

The objective of this project are as follow:

i. To evaluate water powered underwater manipulator pressure distribution using CFD.

1.4 PROJECT SCOPE

The scope of the project is used to show the limitation of the research of the project. The scopes of the project are as follows:

i. Using robotic arm as the submersible water-hydraulic manipulator for evaluating underwater manipulator performance in air medium.
ii. Using robotic arm as the submersible water-hydraulic manipulator for evaluating underwater manipulator performance in water vapor medium.
iii. Using robotic arm as the submersible water-hydraulic manipulator for evaluating underwater manipulator performance in water liquid medium.

© Universiti Teknikal Malaysia Melaka

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter will discuss about the research literature. Some concept of a project is described. This because an understanding of the work will assist in preparing the project end of this year. The background study or literature review come from various resources such as:

1. Books
2. Journals

But in some cases, resources like online article, video and images also contribute to the better understanding and concrete theory on this project. Its main purpose is to acquire knowledge and ideas about topics that have been issued and knows the strength and weaknesses of a study of the literature.

© Universiti Teknikal Malaysia Melaka

2.2 COMPUTATIONAL FLUID DYNAMICS

2.2.1 ANSYS Fluent

ANSYS Fluent software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants [3]. For the project, this software is been used generally to do analysis for the prototype model that been proposed.

2.2.2 CFD Pre-processing

i. Geometry

In obtaining the fluid region from the data computed in the CFD, the geometry must be applied to the software. The geometry of the target analysis model can be draw in the Design Modeler in the sub module of Geometry in the toolbox. The analysis model also can be imported from any type of CAD drawing such SolidWorks, CATIA or AutoCAD which is called the geometry interfaces. This feature allows direct interfaces to the CAD system. So, the CAD geometry can be directly used without translating the geometry files to other intermediate geometry formats. The changes in geometry are the ones that mainly relate to possible modifications on the geometrical aspects of the bodies that interact inside the simulation. The main possibilities that used for the geometry feature in the project are:

a) Import external geometry from CAD file, SolidWorks.
b) Enclosing the model bodies using Enclosure.
c) Addition-Subtraction of bodies using Boolean.

© Universiti Teknikal Malaysia Melaka

ii. Mesh

A mesh divides a geometry into many elements. These are used by the CFD solver to construct control volumes. Mesh generation is very crucial in simulation. The element or cell that been divided in the meshing will determine the accuracy of the simulation. This can be determined by the shape of the meshing by the mesh control and detail meshing generation. The cell that generate by the meshing, may result in the time required to run the solver or solution but more precise and accurate result. The feature in the Meshing used for the experiment are:

a) Mesh sizing.
b) Shape modification of the elements.
c) Named Selections.

iii. Physics and Solver Setting

Before continuing with the simulation, the properties of the material which is the model geometry and the surrounding of the model need to be defined. It can be done by undergoes Setup sub module in the Fluent. Usually, the model geometry is set in solid properties while the surrounding is set as fluid. This changes is related to the possible modifications the user might want to do over the physical constrains of the simulation. The possibilities that used for the project are:

(a) Fluid properties.
(b) Boundary conditions.
(c) Time step of the simulation.

2.2.3 CFD Solver

After setting constrain in the Setup, the simulation is been compute by running the calculation based on the number of iteration that been set by the user. The calculation run until convergence. Convergence is reached when

a) The solution variables change from one iteration to the next are negligible.

© Universiti Teknikal Malaysia Melaka

b) Overall property conservation is achieved.
c) Quantities of interest have reached steady values.

2.2.4 CFD Post Processing

Post processing is where the users can selected data for display. Massive amount of information is given to the user which can access after solving an scenario. This allow users to examine the results to review solution and extract useful data for differ cases. The tools in showing the result of the compute solution are visualization and numerical reporting tools. The visualization tool can display the overall flow pattern occur on the model. The numerical reporting tools can be used to calculate the quantitative result such as forces and moments happens which been display in the solver. The report also can display the average heat transfer coefficient, surface and volume integrated quantities. From this post processing, the user can do changes according to the cases that not satisfy the result needed by update the model in the pre-processing step and compute the solution until converge again [4] [5].

2.3 ANALYSIS CONSIDERING JOINT

Researchers Shibata, M. et al. [6] have done research on the joint of the manipulator arm called slave arm by developing a joint mechanism to stimulate underwater robot arm with a small-scale body and high waterproofing properties composed of combinations of rigid and flexible members with three degree of freedom (3DOF). Leaf spring is been utilized as the flexible joint in the mechanism in avoiding the use of gear and eliminating sliding parts from the joint. From the research, it have been found that the range of the moving angles of the joint was dependent on the characteristics of the flexible joint. The range of moving angles of the joint mechanism also dependent on the contraction rate of actuator, with higher contraction rates resulting in larger angles.

© Universiti Teknikal Malaysia Melaka

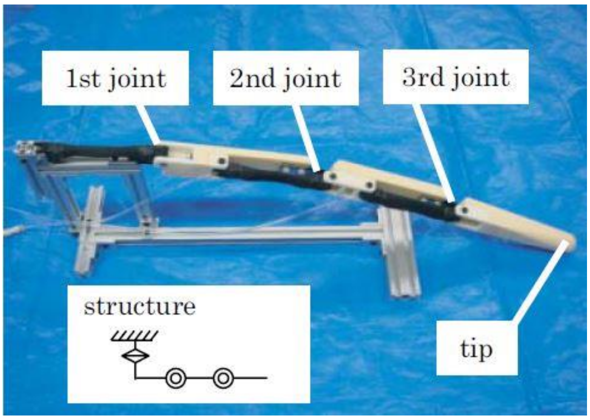

Figure 2.1: Slave arm including a flexible joint driven by water pressure[6]

Another research on the joint is by the Jun, B.H. et al. [7] considering task oriented manipulability analysis of the arm by setting the joint configuration based on the Denavit-Hartenberg parameters task-oriented manipulability measure (TOMM) and task oriented manipulability ellipsoid (TOME). The task-oriented manipulability measure is utilized to find optimal posture in off-line and to control the tele-operated manipulator in on-line.

So, in analyzing the manipulator arm, the joint configuration or angle joint must be considered for working of manipulator arm during underwater task. It is crucial not only to operate the manipulator arm in different angle but also to consider all the DOF in the simulation. Different in angle joint may varies the fluid flow through the model.

2.4 ANALYSIS FLUID FLOW CONDITION

In predicting the impact or interaction happen to the model, analysis is need to be done. The existing CAE software such as Computational Fluid Dynamic (CFD) may allow the analysis as contain and provide dynamics solution that can be trusted. The result that needed can be obtained from the software after undergoes several steps of analysis from the modelling to solution. In analyzing using CFD some research is been done on the parameter that need to be included in the project.

© Universiti Teknikal Malaysia Melaka

In 2012, Joung, T.H et al. [8] have done research on underwater disk robot (UDR) which designed as a disk shaped vehicle in order to minimize the effect of external disturbances such as currents and waves. In the research, CFD software, ANSYS-CFX is been used to evaluate the design by resistance, propulsion and motion test. These test required to predict the drag force propulsion and motion performance of the UDR. For the analysis on the shape of UDR body, the condition is set in a cylindrical shape water tank as the domain. The length of the tank was made long enough to observed the wake from the UDR body such Figure 2.2.

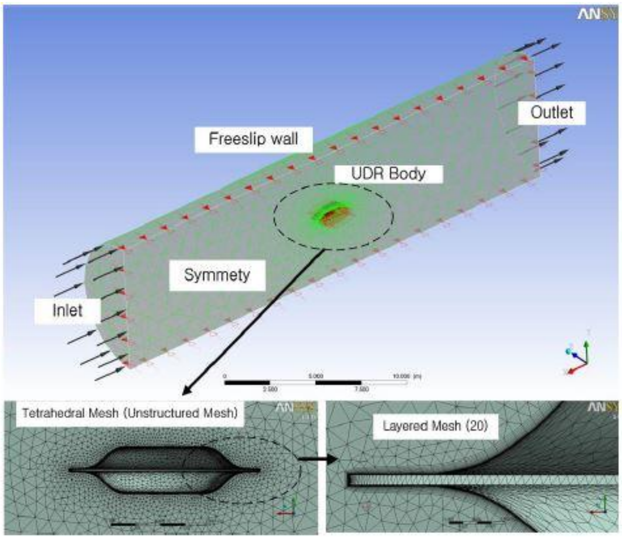

Figure 2.2: Review of the pre-processing and meshing results for the CFD analysis[8].

The design speed of the UDR in CFD model is at 3.5 knots@ 1.8004 m/s. The total drag force on the UDR body, composed of the friction drag force and the form drag force, was obtained by CFD analysis.

The research was continued in 2014 [9] with the same model which include new study in the effect of meshing on the converge study of the model performed at the same design speed which is 3.5 knots. The results are displayed in Table 2.2. It was observed that once the number of cells for the case is reached the size of the reference, the variation in the total drag force (R_T) was small considering the significant changes made to the simulation. The reference mesh size for the case was therefore considered to be of sufficient accuracy.

© Universiti Teknikal Malaysia Melaka