Hydrodynamic modelling and identification of the robotics research centre remotely operated vehicle mark II

Eng, You Hong

2009

Eng, Y. H. (2009). Hydrodynamic modelling and identification of the robotics research centre remotely operated vehicle mark II. Master's thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/41743>

<https://doi.org/10.32657/10356/41743>

HYDRODYNAMIC MODELLING AND IDENTIFICATION OF THE ROBOTICS RESEARCH CENTRE REMOTELY OPERATED VEHICLE MARK II

ENG YOU HONG

School of Mechanical and Aerospace Engineering

A thesis submitted to the Nanyang Technological University
in partial fulfillment of the requirement for the degree of
Master of Engineering

2009

ACKNOWLEDGEMENT

If not for the assistance and support of the following people, this work would have been much more difficult, if not impossible to accomplish:

In school, I would like to express my sincere gratitude and appreciation to my supervisor, A/P Michael Lau and co- supervisor, A/P Eicher Low. Their guidance and mentorship, and support at all levels are always being appreciated.

Special thanks to all laboratory technicians in Robotics Research Centre, Mr Lim Eng Cheng, Ms Agnes Tan, Mr You Kim San and Ms Toh Yen Mei for assistant in maintenance and setting up of the experiment equipments.

All the experimental works could not be carried out without a team of members. Hence, I would like to thank Cheng Siong, Gin Chee, Ti Liang, Minh Tuan, Si Feng Andre and Chun Yu. We really worked as a team and made things happen.

I also would like to thank my friends for providing mental support and share of tear and joy all along the way. And above all, my gratitude goes to my wife, Shu Lien.

TABLE OF CONTENTS

Acknowledgement..... i

Table of Contents ii

Summary.....v

List of Figures..... vi

List of Tablesx

List of Acronyms..... xi

Chapter 1. Introduction1.1

 1.1 Background.....1.1

 1.2 Research Platform.....1.3

 1.3 Objective.....1.4

 1.4 Organization of report.....1.4

Chapter 2. Literature Review2.5

 2.1 Dynamics Model.....2.5

 2.2 System Identification2.6

 2.2.1 Analytical.....2.8

 2.2.2 Semi-empirical.....2.8

 2.2.3 Computational Fluid Dynamics (CFD).....2.9

 2.2.4 Towed Tank Test (PMM)2.10

 2.2.5 On-board Sensor Identification.....2.11

 2.2.6 Free Decays Test.....2.12

 2.3 Research Gap2.13

Chapter 3. Added Mass3.14

 3.1 Introduction.....3.14

 3.2 WAMIT3.15

 3.3 Pre-Processing3.16

 3.3.1 Geometry Build Up.....3.16

 3.3.2 Meshing Domain-Panel Method3.17

 3.4 Boundary Conditions & Model Environment Set Up.....3.18

 3.4.1 MultiSurf.....3.19

 3.4.2 WAMIT3.19

 3.4.3 MATLAB.....3.21

 3.5 Result and Discussion.....3.21

 3.5.1 Case Study3.21

 3.5.2 Effect of changing depth of the submerged body3.24

 3.5.3 Effect of changing origin of the reference frame.....3.24

 3.5.4 Effect of changing orientation of the reference frame3.27

 3.5.5 Effect of change in the body hydrodynamics shapes.....3.28

 3.5.6 Effect of Scaling Object Size.....3.29

 3.5.7 Lower Order Method versus Higher Order Method3.30

 3.5.8 ROV II3.35

 3.6 Chapter Summary3.37

Chapter 4. Hydrodynamics Drag	4.38
4.1 Introduction.....	4.38
4.2 Pre-Processing	4.39
4.2.1 Physical Model	4.39
4.2.2 Mathematic Model.....	4.40
4.2.3 Geometrical Model	4.41
4.2.4 Meshing Domain.....	4.42
4.3 Boundary Condition & Model Environment Setup.....	4.43
4.3.1 Domain.....	4.43
4.3.2 Inlet	4.44
4.3.3 Outlet	4.46
4.3.4 Free Slip Wall Boundary	4.47
4.3.5 Non-Slip Wall Boundary	4.47
4.3.6 Symmetry Plane.....	4.48
4.3.7 Initial Value	4.49
4.3.8 CFX's Solver Control	4.49
4.4 Results & Discussion	4.50
4.4.1 Case Study: Sphere	4.50
4.4.2 Scaled Model of the ROV II	4.54
4.4.3 NTU ROV II	4.58
4.5 Chapter summary	4.61
Chapter 5. Free Decays Test	5.62
5.1 Introduction.....	5.62
5.2 Theory	5.63
5.2.1 Nomenclature.....	5.63
5.2.2 Dynamics Equation.....	5.64
5.2.3 Least Square Algorithm [27]	5.68
5.2.4 Similitude [43]	5.68
5.3 Experiment Procedures	5.71
5.3.1 Experiment Setup.....	5.71
5.4 Experiment Result & Discussion.....	5.73
5.4.1 Experiment in Air	5.73
5.4.2 Experiment in Water (Surge, Sway and Heave).....	5.78
5.4.3 Experiment in Water (Yaw).....	5.82
5.4.4 Comparison: (Experiment Vs CFD)	5.87
Chapter 6. Verification of the Hydrodynamics Parameters	6.91
6.1 Introduction.....	6.91
6.2 Error Sources	6.92
6.2.1 Unwanted Excitation in other degree of freedoms	6.92
6.2.2 Thrusters Modelling.....	6.92
6.2.3 Sensors Limitation	6.92
6.2.4 External Disturbance.....	6.93
6.3 Verification Methodology.....	6.94

6.3.1	Forces and Moments (Detail in Appendix F)	6.94
6.3.2	Dynamics Model.....	6.95
6.3.3	Simulated Response	6.96
6.3.4	Real Plant.....	6.97
6.3.5	Measured Response	6.97
6.3.6	Verification	6.98
6.4	Experiment Result & Discussion.....	6.98
6.4.1	Surge	6.99
6.4.2	Heave	6.101
6.4.3	Yaw	6.103
6.5	Conclusion	6.104
Chapter 7	Conclusion and Future Work	7.105
7.1	Conclusion	7.105
7.2	Future work.....	7.106
7.2.1	Navigation.....	7.106
7.2.2	Control	7.106
References		Ri
Appendix A:	Properties of Hydrodynamics Added Mass	Ai
Appendix B:	Potential Flow Theory.....	Bi
Appendix C:	Physic of Hydrodynamics Drag	Ci
Appendix D:	Viscous flow: Navier-Stokes Equation	Di
Appendix E:	Typical Stages in CFD.....	Ei
Appendix F:	Thrusters Identification.....	Fi

SUMMARY

Dynamics model is essential and critical for a successful navigation and control system design of an underwater vehicle. The main difficulty in dynamics modelling of an underwater vehicle is due to the hydrodynamics forces, in which added mass coefficients and drag coefficients need to be determined. In this project, two approaches - computational fluid dynamics (CFD) approach and experimental approach have been used to find the hydrodynamics coefficients for the RRC ROV II, a remotely operated underwater vehicle. Under the CFD approach, WAMIT and ANSYS CFX have been used to estimate added mass coefficients and drag coefficients respectively. After that, the coefficients were compared with the experimental result obtained from a novel free decay test of a scaled model of ROV II. The results show good agreement in prediction of both added mass and drag coefficients. The usage of experimental method and CFD method in parallel has suggested that the two methods complement each other and one's advantages could be used to mask another's weaknesses. The hydrodynamics coefficients found have been verified through experiments using physical ROV II. Therefore, the combined-approach developed in this report could be used to find the added mass and drag coefficients for other underwater vehicles.

LIST OF FIGURES

Figure 1.1 NTU RRC ROV II - The Twin-Barrel1.3

Figure 1.2 Work Flow of the Report.....1.4

Figure 2.1 Classification of Identification Method.....2.7

Figure 2.2 REMUS Vehicle Profile (Taken from [17]).....2.9

Figure 2.3 HUGIN 3000 AUV and CR-02 AUV [21].....2.9

Figure 2.4 (a) Planar Motion Mechanism (PMM) attached above the Towed Tank
(b) The PAP 104 Mine Disposal Vehicle was tested using the PMM
(Picture taken from [24]).....2.10

Figure 2.5 Experimental Set-up for Longitudinal Identification showing the UUV
attached by 4 Springs. Picture Taken from [29]2.12

Figure 3.1 Geometry of ROV II modeled in MultiSurf. Only half of the ROV II is
modelled to take advantage of the symmetry of the ROV II in XZ-
plane. (Save in .MS2 format)3.17

Figure 3.2 Representation of ROV II Geometry by Panels. In Lower-order Method,
Surfaces are represented by thousands of small surface with four edges. .3.17

Figure 3.3 Programs Flowchart in Simulation Setup.....3.18

Figure 3.4 POT File3.19

Figure 3.5 FRC File3.20

Figure 3.6 GDF File.....3.20

Figure 3.7 Sphere drawn in MultiSurf [Origin = (0,0,0) Radius, r = 1m Density,
 ρ]3.21

Figure 3.8 Cylinder drawn in MultiSurf [Origin = (0,0,0) Radius = 1m Length =
80m Density = ρ]3.22

Figure 3.9 Added Mass of a Sphere versus Depth of submerged.....3.24

Figure 3.10 Effects of shifting the Origin on the Added Mass.....3.25

Figure 3.11 Effect of changing Orientation of the Reference Frame on Added Mass
Matrix: (a) Hemisphere locate at Positive X-axis (b) Hemisphere locate
at Negative X-axis.....3.27

Figure 3.12 Different Shapes of Ballast and their Added Mass Matrix.....3.29

Figure 3.13 Process in preparing Geometry File for Lower-order Method.....	3.30
Figure 3.14 Cylinder Hull drawn using MultiSurf	3.32
Figure 3.15 Subdivision of one quadrant of a Cylinder into Patches and Panels.	3.32
Figure 3.16 ROV II drawn in MultiSurf.....	3.35
Figure 3.17 Convergence Test: Added Mass for ROV II	3.36
Figure 4.1 Lift and Drag	4.38
Figure 4.2 Flow Separation (Taken from [15]).....	4.39
Figure 4.3 Physical Model and Computer Simulation Model	4.39
Figure 4.4 ROV II Model in SolidWork.....	4.41
Figure 4.5 CFX-Mesh. The domain meshes consist of tetrahedral, prisms and pyramids.....	4.42
Figure 4.6 Meshing Domain	4.44
Figure 4.7 Inlet Settings.....	4.46
Figure 4.8 Outlet Settings	4.46
Figure 4.9 Free-Slip Boundary Condition	4.47
Figure 4.10 Non-Slip Boundary Condition.....	4.48
Figure 4.11 Symmetry Boundary Condition.....	4.49
Figure 4.12 Mesh Domain of the Sphere.....	4.51
Figure 4.13 Streamline Plot at $Re = 100$	4.51
Figure 4.14 Pressure coefficient around the Sphere and Wall Shear around the Sphere, $Re = 100$	4.52
Figure 4.15 Streamline Plot at $Re = 300$	4.53
Figure 4.16 Streamline Plot at $Re = 1.1 \times 10^6$	4.53
Figure 4.17 CAD Modeling of Scaled Model of ROV II and its Surface Mesh Plot.....	4.54
Figure 4.18 Cut Surface Mesh Plot at the Middle Plane of the Cylinder Hull	4.55
Figure 4.19 Streamline Plot of the Scaled Model	4.55
Figure 4.20 Iteration History of the Normal Force and Tangential Force acting on the Scaled Model.....	4.55
Figure 4.21 Drag Force for Scaled Model in Surge.....	4.56
Figure 4.22 Drag Coefficient versus Reynolds Number.....	4.56

Figure 4.23 Comparison of experiment and CFD result of Drag	4.57
Figure 4.24 Contribution of Drag due to different components of Scaled Model	4.58
Figure 4.25 Meshing Domain	4.58
Figure 4.26 Pressure Distribution on the ROV II Body.	4.59
Figure 4.27 Streamline and Vector Plot at the Centre Plane of the Cylinder Hull. Flow speed: 0.5m/s.....	4.59
Figure 4.28 Drag Force as the function of Velocity in Surge, Sway and Heave	4.60
Figure 4.29 Drag Torque s a Function of Square Angular Velocity.....	4.61
Figure 5.1 Pendulum Motion in Water	5.62
Figure 5.2 Free Body Diagram of the Pendulum swinging in the Water	5.64
Figure 5.3Transformation form earth fixed frame to body fixed frame	5.64
Figure 5.4 Earth-Fixed Reference Frame and the Body-Fixed Reference Frame	5.65
Figure 5.5 Experiment Setup for Pendulum Test	5.71
Figure 5.6 Series of Frames Cut from the Video during swinging in Air	5.73
Figure 5.7 X-Y Coordinates of the Marker and their Centre of Rotation.....	5.74
Figure 5.8 Time History of Angle Theta	5.74
Figure 5.9 Angular Velocity and Angular Acceleration derived from Polynomial Fitting	5.75
Figure 5.10 Data between the Intervals of 0.5 to 4.5 sec.....	5.76
Figure 5.11 Experimental versus Simulated Angle Theta	5.77
Figure 5.12 Image Sequence of the scaled Model when swinging in the Surge Direction.....	5.78
Figure 5.13 Experiment Data versus Simulated Data for Angle θ in Surge Direction..	5.78
Figure 5.14 Experiment Data versus Simulated Data for Angle θ in Sway Direction..	5.79
Figure 5.15 Experiment Data versus Simulated Data for Angle θ in Heave Direction.....	5.79
Figure 5.16 Drag Force versus Velocity for Surge, Sway and Heave	5.81
Figure 5.17 Free Body Diagram of Torsional Pendulum	5.82
Figure 5.18 Image Sequence of the Scaled Model when oscillated in the Yaw Direction.....	5.85

Figure 5.19 Experiment Data versus Simulated Data for Angle θ in Yaw Direction ...5.86

Figure 5.20 CAD model used in WAMIT and the Real Model used in Experiment.....5.87

Figure 5.21 Drag in Surge Direction.5.88

Figure 5.22 Drag in Sway Direction.5.89

Figure 5.23 Drag in Heave Direction.....5.89

Figure 5.24 Drag in Yaw Motion.....5.90

Figure 6.1 ROV with Umbilical Cable (Tether)6.93

Figure 6.2 Verification Flowchart6.94

Figure 6.3 Simulink Block Diagram for Decoupled Model6.96

Figure 6.4 Thrust for Surge Verification6.99

Figure 6.5 Experimental versus Simulated Position in a positive Surge Movement.....6.99

Figure 6.6 Experimental versus Simulated Speed in a positive Surge Movement6.100

Figure 6.7 Thrust for Heave Verification6.101

Figure 6.8 Experimental versus Simulated Depth in a Heave Movement.....6.101

Figure 6.9 Experimental versus Simulated Speed in a Heave Movement.....6.102

Figure 6.10 Torque for Yaw Verification.....6.103

Figure 6.11 Experimental versus Simulated Depth in a Yaw Movement.....6.103

Figure 6.12 Experimental versus Simulated Depth in a Heave Movement.....6.104

LIST OF TABLES

Table 3.1 Summary of the Files use in solving Added Mass Coefficients3.19

Table 3.2 Low-order Method for Sphere (see 3.5.7)3.22

Table 3.3 Higher-order Method for Sphere (see 3.5.7).....3.22

Table 3.4 Low-order Method for Cylinder3.23

Table 3.5 Higher-order Method for Cylinder3.23

Table 4.1 Initial condition of the flow domain4.49

Table 4.2 CFX’s Solver Control4.50

Table 4.3 Mesh Statistics and Mesh Settings4.51

Table 4.4 Flow parameters at Re = 100. Kim denotes numerical results by Kim and Choi [39], and Empirical denotes $C_d = \frac{24}{Re} + \frac{6}{1 + \sqrt{Re}} + 0.4$ 4.52

Table 4.5 Drag Coefficient at Re = 300. Patel denotes numerical results by Johnson and Patel [40], and Empirical denotes $C_d = \frac{24}{Re} + \frac{6}{1 + \sqrt{Re}} + 0.4$ 4.53

Table 4.6 Drag Coefficient at Re = 1.1 x 10⁶. experimental result from Achenbach [41], DES results from Constantinescu [42]4.54

Table 4.7 The drag coefficient of ROV II in its four principle motions4.60

Table 5.1 The Added Mass Coefficients and Drag Coefficients in Surge, Sway and Heave Direction5.80

Table 5.2 Oscillation Period of the Rectangular Iron Block in Air5.84

Table 5.3 Oscillation Period of the Scaled Model in Air.....5.84

Table 5.4 The Added Mass Coefficients and Drag Coefficients in Yaw Motion.....5.86

Table 5.5 Comparison of Added Mass from experiment and WAMIT5.87

Table 5.6 Comparison of Linear and Quadratic Drag Coefficient from Experiment and ANSYS CFX5.88

Table 6.1 Hydrodynamics Coefficients for Surge, Heave and Yaw of ROV II6.97

Table 6.2 Sensor Information to measure response in Surge, Heave and Yaw6.97

LIST OF ACRONYMS

CFD	=	Computational Fluid Dynamics
UUVs	=	Unmanned Underwater Vehicles
AUVs	=	Autonomous Underwater Vehicles
ROVs	=	Remotely Operated underwater Vehicles
WAMIT	=	Wave Analysis M.I.T
CAD	=	Computer Aided Design
MiltiSurf	=	A CAD software for Surface Modeling of ROV II
SolidWork	=	A CAD software for Slid Modeling of ROV II
Matlab	=	Matrix Laboratory
Simulink	=	Simulation and Model-Based Design
ANSYS CFX	=	ANSYS CFX software is a powerful and flexible general-purpose computational fluid dynamics (CFD) package.

CHAPTER 1. INTRODUCTION

1.1 BACKGROUND

More than 70% of the earth is covered by ocean. The difficulties of mankind to reach the deep ocean causes most of its resources are remain in natural condition. In the last twenty years, the use of Unmanned Underwater Vehicles (UUVs) as a tool to explore secret life underwater has developed rapidly. Generally, UUVs can be classified as Remotely Operated Vehicles (ROVs) and Autonomous Underwater Vehicles (AUVs). ROVs are tethered from a supporting vessel that allows tele-operation and power supplies to the vehicle. On the other hand, AUVs are completely autonomous vehicle that carry its owned power and make any decision on its own [1, 2].

Among the two, remotely operated vehicles (ROVs) are the main workhorses used in the industry. Compare to human diver, ROVs can go deeper and riskier area and carry out a vast variety of tasks. For oil industry, ROVs are used to carry out construction, maintenance and repairing of benthic stations and offshore platforms. ROVs can also be used in scientific and environmental data acquisition. Last but not least, ROVs might play an important role in homeland security by identifying the possible treats from marine. So, the utilization of underwater robotic system will greatly benefit all kinds of ocean activities [3].

In the last twenty years, the design of an underwater vehicle has benefit greatly from the technology advancement in both mechanic and electronic components. These include sensors that are more reliable, an embedded system with higher computing power and memory and etc. However, there are numbers of issues, which remain challenging in underwater robotics. First, the dynamics of ROVs are inherently nonlinear due to rigid body coupling and hydrodynamics forces. The ROVs are also required to be operated under unpredictable disturbances from its environment such as current and wave. Furthermore, the position of the ROV could not be directly measured. Therefore, we depend on other measurable data such as velocity to derive the position of the vehicle. However, the measured velocities are generally corrupted by the noise and operated at low sampling frequency.

The mentioned challenges and problems affect not only control system design but also the navigation and guidance ones. According to Fossen [4, 5], a marine vessel control system could be constructed as three independent blocks denoted as guidance, navigation, and control (GNC) systems. Guidance system is the reference generator that computes the optimal trajectory or path for the vessel to follow; Navigation is equivalent to a sensing system that estimates the position, orientation of the vehicle; Control system using feedback from the navigation system to compute the required actuator inputs to move the vehicle in the desired path.

A dynamics model is always needed in designing GNC systems of an underwater vehicle. This is described in Caccia [6]

“The need for performance improvements in navigation, guidance and control, motivates a deeper investigation on the hydrodynamics modelling and identification of the open frame ROVs.”

Generally, there are three main purposes to model the dynamics of an underwater vehicle. First, the dynamics model is essential in designing a high performance controller. Morrison [7] had stated that a reliable model is necessary for successful control of the vehicle during task performance. Many different control approaches have been proposed and studied. There are classical PID controls, optimal control, adaptive control, sliding mode control, feedback linearization based control, Lyapunov based robust control, gain scheduling control, fuzzy control and neuro-network control. These control approaches promise some robustness in dealing with model uncertainty and disturbances. However, a nominal model is needed for control synthesis. At the same time, the control performance would be greatly enhanced when the model uncertainty is reduced.

The dynamic model is also useful in navigation system design. As mentioned earlier, the navigation system is suffered from low sampling frequency and the data that is corrupted by noises. Usually, only the partially feedback of the vehicle's states is available. This generally requires a state estimation technique such as Kalman Filter to estimate the missing states. As an optimal state estimator, Kalman Filter will work more efficient if the dynamics of plant is known. The works done by Caccia [8], Smith [9] and Steinke, D.M [10] have demonstrated that performance of their navigation systems have been greatly enhanced by employing the dynamics model in Kalman Filter.

The dynamics model is also needed to construct the vehicle's simulator. The simulator enables an operator to be trained in the virtual environment and this will save both cost and time if compared with training that conducted in the field.

Obviously, an accurate dynamics model of the ROV plays an important role in various ROV applications. Unfortunately, the dynamics model of an underwater vehicle is hard to obtain in practice. This is mainly due to the unknown hydrodynamics forces that consist of two major components: added mass forces and drag forces. In standard 6 DOF dynamics model of an underwater vehicle, these hydrodynamics forces are described by a total of 144 coefficients [4]. These forces are not normally seen in land vehicle and airplane. This is because the density and viscosity of the air are magnitude lower than the water. Nevertheless, for an underwater vehicle, the hydrodynamics forces that resist the motion of the vehicle are very significant and thus cannot be ignored even in low speed. As such, the work presented in this thesis is to develop a systematic approach to compute the hydrodynamics coefficients.

1.2 RESEARCH PLATFORM

The platform for this research is the NTU RRC ROV II: The Twin Barrel developed by Robotics Research Center, Nanyang Technological University supported by British Gas [11]. ROV II as an inspection class ROV is designed for pipeline inspections and seabed surveys. The ROV II is designed with two cylinder hulls that serve two distinctive functions: navigation and control. It is equipped with an array of sensors such as magnetic compass tilt sensor, inertia measurement unit (IMU), depth sensor and Doppler Velocity Logger (DVL) for navigation purpose. It is also equipped with four electric thrusters for manoeuvring purpose.

Figure 1.1 NTU RRC ROV II - The Twin-Barrel

1.3 OBJECTIVE

The objective of this project is to construct the dynamic model of the ROV II with focus on the identification of the hydrodynamics added mass coefficients and drag coefficients.

Scope:

The hydrodynamics coefficient is derived by making following assumptions:

- The vehicle is deeply submerged in a homogeneous, unbounded fluid. The vehicle is far from the sea surface (no sea wave effect), seabed and other structure.
- The vehicle does not experience multidirectional underwater current.
- Tether Effect is ignored

1.4 ORGANIZATION OF REPORT

This report is structured as follows. Chapter 2 reviews the relevant literature in finding hydrodynamics coefficients of an underwater vehicle. Application of WAMIT in finding the added mass matrix is explained in chapter 3. In chapter 4, drag coefficients of the ROV II will be studied based on commercial CFD software ANSYS CFX. In chapter 5, a novel free decay test on a scaled down model of ROV II will be proposed and studied. The hydrodynamics coefficients found are verified through experiment in chapter 6. Finally, a conclusion is given in chapter 7.

Figure 1.2 Work Flow of the Report

CHAPTER 2. LITERATURE REVIEW

2.1 DYNAMICS MODEL

Modelling of an underwater vehicle has been widely discussed in research community. The lumped parameters model proposed by Fossen [4, 5] is generally accepted and used as the standard in underwater vehicle modelling. The model which derived from Newtonian and Lagrange mechanics, is a six DOF nonlinear dynamics equation that contain more than hundred parameters. The equation of motion can be expressed as:

$$M_{RB}\dot{v} + C_{RB}(v)v = \tau_{RB} \quad (2-1)$$

where

M_{RB} = Rigid body inertia matrix

$C_{RB}(v)$ = Coriolis and centripetal terms

τ_{RB} = Rigid body forces and moments

The above equation is simply an expression of Second Newton's Law, which relates the vehicle's acceleration to the acting forces and moments. The forces and moments come from three different sources: *hydrodynamics* (added mass, drag and restoring force), *environmental* (ocean wave and current) and *propulsion* forces from the vehicle's thrusters.

$$\tau_{RB} = \tau_H + \tau_E + \tau_P \quad (2-2)$$

Where τ_H = Hydrodynamics forces and moments

τ_E = Environmental forces and moments

τ_P = Propulsion forces and moments

Hydrodynamics forces and moments, τ_H is the focus in this report. The hydrodynamics forces and moments consist of following components:

$$\tau_H = -M_A \dot{v} - C_A(v)v - D_L(v)v - D_Q(v)v - g(\eta) \quad (2-3)$$

where M_A = Added mass matrix

$C_A(v)$ = Added mass coriolis and centripetal terms

$D_L(v)$ = Linear skin friction drag

$D_Q(v)$ = Quadratics pressure drag

$g(\eta)$ = Restoring force due to gravity and buoyancy

The negative sign is used to show that the hydrodynamics forces and moments are resistant in nature. Added mass is discussed in detail in chapter 3 and both linear and quadratic drags are treated in chapter 4. The restoring forces due to gravity and buoyancy are function of the vehicle's weight, buoyancy, centre of gravity (CG) and centre of buoyancy (CB). Those data can be found from the CAD model.

2.2 SYSTEM IDENTIFICATION

System identification is a general term to describe a method or technique to build a dynamics model from the measured data. The method often makes use of some mathematical tools and algorithms. From modelling perspective, one could model the dynamics of a plant with a "black box" or "white box". When a plant is considered as a black box, nothing is assumed to be known on the plant. Therefore, the model could be a transfer function or a neural-network that map inputs to the outputs. The model proposed by Fossen is a "white box" type because the plant is described by a system of equations with unknown parameters [2]. This type of model is preferred because the prior knowledge of the plant such as geometry, mass could be utilized in the modelling process. This makes the modelling process easier by reducing the number of unknown parameters.

Identification of the hydrodynamics parameters¹ is very difficult and challenging. Alessandri [2] stated that “The difficulty in tuning an accurate and complete dynamics model for an ROV is a very well known fact [4, 12, 13]. The difficulty is mainly caused by the hydrodynamics forces as mentioned in Morrison [7]: “The hover and tight maneuvering regime of an underwater vehicle is difficult to characterize hydrodynamically.” According to Conte.G [14], the greatest problem encountered in designing an efficient automatic controller is the difficulty in knowing with sufficient accuracy the value of the hydrodynamics parameters.

A wide variety of methods have been proposed by the research community in the identification of the hydrodynamics parameters. In order to facilitate the discussion, different methods have been reviewed and classified in Figure 2.1. As shown in Figure 2.1, identification methods of the hydrodynamics force could be separated into two main groups: theoretical method or experimental method.

Figure 2.1 Classification of Identification Method

Each identification method will be discussed in the following section.

¹ Hydrodynamic parameters are anonyms to hydrodynamics coefficients, which include both added mass and drag coefficients.

2.2.1 Analytical

By assuming flow is inviscid and irrotational, theoretical fluid dynamists had developed potential flow theory and strip theory that are useful for determination of added mass. The added mass of simple geometry such as sphere can be calculated analytically because potential function that describes flow around the sphere is available (see Appendix B). However, it is too complex to find the potential function that is able to describe the flow around ROV analytically. On the other hand, the strip theory is only suitable to estimate added mass for a slender body with large length to width ratio.

Hydrodynamics drag is caused by the viscosity of the fluid. Viscous flow in general could be described by Navier-Stokes equation¹. This equation is derived from conservation of mass and momentum of fluid continuum. However, the Navier-Stokes equation could not be solved analytically even for a simple geometry such as a sphere. However, it can be solved numerically in computational fluid dynamics software.

2.2.2 Semi-empirical

Some of the works done in calculation of drag coefficient use semi-empirical formula. The formula is derived by combining theoretical equations and empirically-derived formula [15]. For example, the axial drag experienced by the REMUS autonomous underwater vehicle can be obtained by the following empirical formula [16]:

$$\text{Axial drag coefficient, } C_d = \frac{c_{ss} \pi A_p}{A_f} \left[1 + 60 \left(\frac{d}{l} \right)^3 + 0.0025 \left(\frac{l}{d} \right) \right] \quad (2-4)$$

where c_{ss} is Schoenherr's value for flat plate skin friction

$A_p = ld$ is the vehicle plan area

A_f is vehicle frontal area

d is maximum hull diameter

l is vehicle total length

¹ Please referred to appendix D

Figure 2.2 REMUS Vehicle Profile (Taken from [17])

Unfortunately, these kinds of semi-empirical formulas are only available for axis-symmetry shapes such as torpedo and submarine.

2.2.3 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) provides one of the alternatives to compute both added mass and drag coefficients for a geometry complex object. CFD is capable to simulate the fluid flow by combining fluid dynamics theory, numerical recipe and both computer software and hardware. Due to the flexibility of CFD to simulate flow around different geometries and environments, it has been widely used for various engineering problem involving fluid flow. In year 1997, Sarkar [18] used CFD to simulate fluid flow past axis-symmetry AUVs to study the effect of different hull forms. This is followed by Ng [19], where Flow-3D, a commercial computer program was used in evaluation of drag, lift and static pressure contours of an underwater vehicle. In reference [20], CFD was used to find stability derivatives which assist in controller design of the heading angle.

Figure 2.3 HUGIN 3000 AUV and CR-02 AUV [21]

In year 2004, Fuglestad [21] who studied on HUGIN 3000 AUV showed that the experimental drag forces agreed well with the computational results. Based on their study in CR-02 AUV, reference [22, 23] concluded that hydrodynamics coefficients obtained

with CFX, a general purpose CFD software is of high accuracy. As far as author's knowledge is concerned, those successful applications of CFD in the calculations of hydrodynamic coefficients are limited to underwater vehicles with streamline profile. Compare to streamline profile vehicles, ROV II is more complex in geometry and the resultant fluid flow may be hard to predict.

2.2.4 Towed Tank Test (PMM)

Due to the complexity of the flow, the most established way to estimate hydrodynamics forces is by experiment. The experimental study of the hydrodynamics forces is dates back to the 18th century. At that time, people were interested in minimizing the drag acting on the ship's hull. Since then, hydrodynamics drag for simple shapes such as sphere, cylinder and thin circular disk have been studied in water tunnel.

The two main theories in experimental fluid dynamics are the Buckingham π Theorem and the Dynamics Similitude Law. The Buckingham π Theorem allows complex flow problem to be analyzed more easily by forming dimensionless numbers. The Similitude Law enables the use of scale model to study real flow pattern around the actual model under certain conditions. Scaled model tests are widely used in various areas including the design of ships, airplanes and dams. Usually, the scaled down models are made and tested in either wind or water tunnel. The measured forces are then scaled up to estimate the forces that will be experienced by the full-scaled model.

Figure 2.4 (a) Planar Motion Mechanism (PMM) attached above the Towed Tank (b) The PAP 104 Mine Disposal Vehicle was tested using the PMM (Picture taken from [24])

Traditionally, the hydrodynamics coefficients are identified through tow tank test of the vehicle itself or of its scaled model. Special equipment called planar motion mechanism (PMM) is built on top of the tow tank to move the vehicle in planar motion. The PMM test could be conducted in both horizontal plane and vertical plane as reported in [25]. As explain clearly in [26], the underwater vehicle is connected to a towing carriage though struts and force dynamometers. During the test, the towing carriage is moved at constant speed and the resultant forces and moments are measured. Then, the hydrodynamics forces acting on the body are obtained by subtracting contributions from body mass inertia and hydrostatic forces (buoyancy and gravity). Subsequently, the hydrodynamics coefficients are obtained using system identification technique. Since the measured forces and moments are available in six DOF, the tow tank test allows complete model identification. However, it involves very high cost in building the tow tank equipped with a PMM. In addition, the test procedures are highly time-consuming.

2.2.5 On-board Sensor Identification

More recently, on board sensor-based identification is very popular within the research community. The relevant studies have been reported in [2, 27, 28]. This technique is preferable because it makes use of on board sensors and thrusters in the process of identification. No other equipment is needed. The technique is cost effective, easy to execute and the repeatability is high. It is very suitable for variable configuration ROVs where payload and shape of ROVs changed according to different missions [6]. Ridao P. et al [27] had determined the hydrodynamics coefficients only for uncoupled one-dimensional models in the principle DOFs (surge, heave and yaw) of the URIS underwater vehicle. An on-line deterministic adaptive identification method was first proposed by [28], where JHUROV, a ROV from Johns Hopkins University was the subject of study. However, identification of full model of the ROVs through this method is very difficult. Most of the works simplify the model to an uncoupled model in single DOF. To conduct an uncoupled experiment, the motion of the ROV must be constrained in a single DOF, which is hard to implement practically. In addition, it is hard to model the thrusters' forces and to measure the vehicle's response accurately.

2.2.6 Free Decays Test

The use of free decay test in finding hydrodynamics coefficients was first done by Morrison [7] in 1993. In his study, the hydrodynamics coefficients of the *ROV Hylas* were determined successfully for heave motion. The ROV Hylas was allowed to oscillate in water by hanging it to an overhead crane using three springs. The position of the Hylas was determined using Sonic High Accuracy Ranging and Positioning System (SHARPS). Free decay test was further studied by Andrew [29] to identify a multiple-DOF model of an UUV. In his proposed experiment, the underwater vehicle is attached to four springs as shown in Figure 2.5. The method was tested in a computer simulation and the result converges to true values. The main problem is the ability to measure states of the vehicle accurately.

Figure 2.5 Experimental Set-up for Longitudinal Identification showing the UUV attached by 4 Springs. Picture Taken from [29]

The proposed free decay tests exhibit few problems in practical application. Firstly, all springs must be always kept in tension during the oscillations. When the vehicle is attached to the springs, it is very hard to constraint the motion within the predefined DOF. As a result, for a single DOF model, it may not be possible to capture the motion accurately and thus the identified results might be poor. Secondly, the vehicle's position is needed during identification. In Morrison [7], the problem is solved by employing an expensive underwater positioning system (SHARPS). In Andrew [29], only computer simulation was done.

2.3 RESEARCH GAP

As far as the NTU RRC ROV II is concerned, the analytical method could not provide solution due to the complexity of the vehicle's shape. Those published semi-empirical formulas are limited only to finding the drag for axis-symmetry vehicles. Planar Motion mechanism (PMM) could be used to estimate hydrodynamics parameters with high accuracy, but currently, there is no such facility in NTU and it is too expensive to build one. On-board sensor identification method requires the navigation system to provide accurate position and orientation data. For uncoupled identification, the vehicle's motion needs to be constrained to a single DOF. None of this can be done without a proper control and navigation system. Since navigation and control systems are currently under development, this method could not be used.

Computational Fluid Dynamics (CFD) provides an attractive alternative among the methods due to its ability in analyzing fluid flow involving complex geometry. Advancement in both computer software and hardware has allowed the CFD problem to be formulated and solved in a personal computer. Recently, successful application of CFD in finding drag coefficients of an underwater vehicle has been reported although it is limited to vehicles with streamline profile.

However, the result from CFD, a computer simulation could not be fully trusted without any experimental validation. So, in order to complement the CFD result, a free decay test on a scaled down model could be carry out to verify the results from CFD. By applying the Similitude Law, the force acting on the full-scale vehicle can be predicted provided the Reynolds's Number is kept the same. So in this report, an approach that combine both CFD and free decay tests will be used to find the hydrodynamics coefficients of ROV II.

CHAPTER 3. ADDED MASS

3.1 INTRODUCTION

According to Newman [30], the added mass can be interpreted as a particular volume of fluid particles that accelerate with the body. Due to the low density of the air compare to the water, the effect of added mass is often ignored when working in air but not so in the water.

Added mass can be treated as the weight added to a system when the body must move some volume of surrounding fluid with it during its acceleration or deceleration motion. The added mass force opposes the motion, and acts as a kind of resistance to motion of an object moving in the fluid. It is given the name “added mass” because it can be treated as an additional mass to the rigid body inertia mass.

The added mass term can be incorporated into the equations of motion by considering an effective mass as the sum of the mass and added mass. That is, $F = m \cdot a$ becomes $F = (m + \text{added mass}) \cdot a$. For example:

Equation of motion for a body of mass M that moves with acceleration $\dot{U}$ is:

$$M\dot{U} = \sum F = F_H + F_{other} = (-\dot{U}m_a) + F_{other} \quad (3-1)$$

The equation could be rewrite as:

$$(M + m_a)\dot{U} = F_{other} \quad (3-2)$$

The present of fluid around the body acts as an **added** or **virtual** mass to the body.

The added mass can also be understood in term of Kinetic Energy. That is any motion of the vehicle will induce a motion in the otherwise stationary fluid. Therefore, to allow the vehicle to pass through the fluid, the fluid must move aside and then close behind the vehicle.

For a general 6 DOF of body motion $U_i = (U_1, U_2, U_3, U_4, U_5, U_6)$,

Kinetic energy (KE) of the fluid that moving along with the vehicle can be described as

$$KE = \frac{1}{2} m_{ij} U_i U_j \quad (3-3)$$

where m_{ij} is the added mass acting on i direction due to the motion in j direction.

U_i and U_j is the instantaneous velocity of the vehicle in respective direction

The added mass coefficients are different depending on the direction of motion. Therefore, it is different from body mass inertia, which is independent of the direction of acceleration. To take into account of this directional property, there are total of 36 added mass coefficients as defined as following:

$$M_A = \begin{bmatrix} X_u & X_v & X_w & X_p & X_q & X_r \\ Y_u & Y_v & Y_w & Y_p & Y_q & Y_r \\ Z_u & Z_v & Z_w & Z_p & Z_q & Z_r \\ K_u & K_v & K_w & K_p & K_q & K_r \\ M_u & M_v & M_w & M_p & M_q & M_r \\ N_u & N_v & N_w & N_p & N_q & N_r \end{bmatrix} \quad (3-4)$$

Each term in the matrix is expressed using notation of SNAME. For example, the hydrodynamics added mass force X_A along x-axis due to acceleration in x-axis, $\ddot{u}$ has following relationship: $X_A = X_u \cdot \ddot{u}$

(Force) = (Added mass coefficient) $\times$ (Acceleration)

*Note: Properties of hydrodynamic added mass are discussed in appendix A.

3.2 WAMIT

The objective of this chapter is to find the added mass matrix of the ROV II using WAMIT, a CFD program based on potential flow theory and panel method. A brief introduction on WAMIT is given below:

WAMIT (Wave Analysis MIT) is a computer program that is based on the 3D panel method for analyzing hydrodynamic interactions with floating or submerged bodies, in the presence of ocean waves. Version one of WAMIT was first developed in MIT in 1987. Since then, WAMIT has gained widespread recognition for its ability to analyze complex structures with a high degree of accuracy and efficiency. WAMIT is used to model the potential flow around submerged structures. Its results have been validated and accepted as the industrial standard. In WAMIT, the ideal flow is assumed, and the free surface boundary condition is linearized. The radiation and diffraction velocity potentials are obtained using Greens' theorem with the free surface source potential as the Green function. In evaluation, the source strengths are evaluated using the source distribution method, which uses the same source potential.

3.3 PRE-PROCESSING

3.3.1 Geometry Build Up

The geometry of ROV II is modeled using MultiSurf, a surface-base computer aided design (CAD) software. MultiSurf is a rich and versatile system for parametric design of 3D geometric objects. It is particularly useful in drawing complex objects involving freeform curves and surfaces. The basic package consists of a complete CAD package with the ability to create freeform surface shapes that are either difficult to create or not available in other CAD packages.

The geometry of the ROV II is represented by a number of surfaces in MultiSurf as shown in Figure 3.1. In MultiSurf, surface is defined by curves and curve is defined by points. So, each surface is drawn individually and later assembled together to form the complete model. As shown in Figure 3.1, only half of the ROV II is drawn to make use of the symmetry of ROV II in XZ-plane. The property of symmetry is favorable because it will simplify the added mass matrix and computation resources (time and memory) required by the WAMIT.

Figure 3.1 Geometry of ROV II modeled in MultiSurf. Only half of the ROV II is modelled to take advantage of the symmetry of the ROV II in XZ-plane. (Save in .MS2 format)

3.3.2 Meshing Domain-Panel Method

The panel method was first developed by Hess and Smith [31] and it is now widely used to solve potential flow problem. The method uses the property of the Laplace’s Equation, which enable 3D flow field problem to be converted to 2D problem in finding the potential on the surface. The entire fluid-structure boundary is replaced by a set of sources such that the combined strength of all sources is equivalent to the source strength of the flow. In order to do so, the fluid structure boundary must be first discretized into a number of quadrilateral surface (panels). Then, the unknown sources of constant strength are assigned at the centroids of each panel. After that, the Laplace’s Equation is solved by solving a linear set of algebraic equations to determine the unknown strengths of the sources. A survey on panel method is given by Erickson [32].

Figure 3.2 Representation of ROV II Geometry by Panels. In Lower-order Method, Surfaces are represented by thousands of small surface with four edges.

3.4 BOUNDARY CONDITIONS & MODEL ENVIRONMENT SET UP

The following chart shows how three programs, namely MATLAB, MultiSurf, and WAMIT are used together in the added mass analysis and their respective input and output Files

Figure 3.3 Programs Flowchart in Simulation Setup

Table 3.1 Summary of the Files use in solving Added Mass Coefficients

File Extension	Description
.MS2	Output from MULTISURF, a geometry model file
.PAT	Output from MULTISURF, defines the geometry by patch
.GDF	Geometry Data File; defines the geometry in panel form. It can be generated from .PAT using pat2gdf.exe
.POT	Potential Control File: input various parameters to POTEN subprogram. (Reference frame, depth, gravity, length)
.FRC	Force Control File: input various parameters to the FORCE subprogram. Determine the desired hydrodynamic parameters
.OUT	Output from WAMIT. Post-processing using MATLAB
readAM.m	Matlab m-file program script to read the added mass matrix from .OUT file.
Plot_wamitgdf.m	Matlab m-file program script to display the panels by reading from .GDF file.
Simulink	Matlab Tool for simulation of dynamics model.

3.4.1 MultiSurf

The geometry build by MultiSurf is saved as .MS2 file.

3.4.2 WAMIT

Figure 3.4, 3.5 and 3.6 describe those parameters that can be specified in .POT, .FRC and .GDF respectively. These files are required by WAMIT.

- POT

Figure 3.4 POT File

The first line is the ‘header’ for the user to insert a brief description of the file. HBOT specify the depth of the water. ‘-1’ means infinite water depth. XBODY (1-3) are the dimensional coordinates of the origin of the *body-fixed coordinate system*. Other parameters are explained in the WAMIT user manual page 3-7

- FRC

Figure 3.5 FRC File

The .FRC file specifies the hydrodynamics output from the WAMIT. The “1” in IOPTN (1) specify WAMIT outputs the added mass in the .OUT file.

- GDF

Figure 3.6 GDF File

The first line is the “header”; ULEN is the dimensional length characterizing the body dimension and normally has a value of one. GRAV is the acceleration due to gravity. If the unit length is meter, then GRAV would be 9.80665. When there is mirror symmetry in the x and y axis, both ISX and ISY have value 1. NPANC is the number or panels. The four consecutive lines are the x-y-z coordinates for a panel specified by four points.

3.4.3 MATLAB

Matlab is used in matrix manipulation and for graphical results representation of the panels. For example, Matlab script such as *readAM.m* and *plot_wamitgdf.m* are used to read in the added mass matrix from .OUT and plot the panels respectively.

3.5 RESULT AND DISCUSSION

3.5.1 Case Study

Two case studies had been conducted to verify the results from WAMIT with the theoretical results of simple geometrical shapes. The main purpose of the case study is to verify that each step taken in setting up the problem is correct.

- Sphere

Figure 3.7 Sphere drawn in MultiSurf [Origin = (0,0,0) Radius, $r = 1\text{m}$ Density, ρ]

Theoretically, the added mass of a sphere is $\frac{2}{3}\pi \cdot \rho \cdot r^3$ for surge, sway and heave. The added mass output from WAMIT is normalized against the density. Hence, the added mass of a sphere after normalized against density is $\frac{2}{3}\pi \cdot r^3$.

Table 3.2 Low-order Method for Sphere (see 3.5.7)

Panel Number	Theoretical			Numerical		
	Surge	Sway	Heave	Surge	Sway	Heave
256	2.0944	2.0944	2.0944	2.0171	2.0892	2.0892
512				2.0183	2.0972	2.0929
1024				2.0749	2.0929	2.0929
2304				2.0861	2.0939	2.0940
			Error	-0.4%	-0.01%	≈0%

Table 3.3 Higher-order Method for Sphere (see 3.5.7)

Panel Size	Theoretical			Numerical		
	Surge	Sway	Heave	Surge	Sway	Heave
2 (12)	2.0944	2.0944	2.0944	2.0952	2.0919	2.0924
1 (24)				2.0944	2.0944	2.0944
0.5 (54)				2.0944	2.0944	2.0945
0.1 (135)				2.0944	2.0944	2.0945
			Error	0%	0%	0%

- Cylinder

Figure 3.8 Cylinder drawn in MultiSurf [Origin = (0,0,0) Radius = 1m Length = 80m Density = ρ]

The theoretical value using strip theory of the added mass is $\pi \cdot \rho \cdot r^2 \cdot L$ (for sway and heave).

Table 3.4 Low-order Method for Cylinder

Panel Number	Theoretical		Numerical	
	Sway	Heave	Sway	Heave
768	251.3274	251.3274	249.6437 (-0.7%)	249.8821 (-0.6%)
3072			248.0583 (-1.3%)	248.2957 (-1.2%)

Table 3.5 Higher-order Method for Cylinder

Panel Size	Theoretical		Numerical	
(Panel number)	Sway	Heave	Sway	Heave
5 (75)	251.3274	251.3274	247.6803	247.7656
2 (150)			247.4787	247.5613
1 (368)			247.4198	247.5017
		Error	(-1.4%)	(-1.4%)

For the case of sphere, the results obtained from WAMIT are within 0.5% of the theoretical results when lower order method was used. The results attained 100% accuracy when higher order method was used. For the case of cylinder, the results obtained from WAMIT are within 1.4% of the theoretical results. The results obtained from high-order method are more accurate and converged faster than the lower order method. One of the most important issues in computational fluid dynamics is the convergence of the result. The body of interests is divided into small pieces and solved individually. Therefore, the result should be converged to a certain value when more numbers of elements (panels) are used. This can be seen especially for lower order

method. As the number of panels used is increased, the calculated added mass converges to the theoretical value for a sphere (Table 3.2).

3.5.2 Effect of changing depth of the submerged body

In .POT file, the depth of the submerged body needs to be specified. Sphere is used as an example to study the change of the added mass when depth is varied. From above section, the ideal added mass of a sphere is 2.0944 kg. The result converges when depth is about 10 meter. Therefore, the depth of the object will be set to 10m for all later analysis.

Figure 3.9 Added Mass of a Sphere versus Depth of submerged

3.5.3 Effect of changing origin of the reference frame

The possible change of the added mass matrix when the origin of the sphere does not coincide with the frame of reference will be investigated. Both linear and rotation motion of the object are defined according to the frame of reference. In this case-study, the term M_{66} of the added mass matrix means that a hydrodynamics moment acts on the sphere when the sphere rotates about the Z-axis which is not through its own axis.

addMass = origin(0,0,0)					
2.0925	-0.0000	-0.0000	0.0000	-0.0000	-0.0000
0.0000	2.0928	0.0000	0.0000	0.0000	-0.0000
-0.0000	0.0000	2.0948	0.0000	0.0000	0.0000
-0.0000	-0.0000	-0.0000	0.0000	-0.0000	-0.0000
0.0000	-0.0000	0.0000	0.0000	0.0000	-0.0000
0.0000	-0.0000	0.0000	0.0000	-0.0000	0.0000
addMass = origin(1,0,0)					
2.0925	-0.0000	0.0000	0.0000	-0.0000	-0.0000
0.0000	2.0928	0.0000	0.0000	-0.0000	2.0928
0.0000	0.0000	2.0948	0.0000	-2.0948	0.0000
-0.0000	-0.0000	-0.0000	0.0000	0.0000	-0.0000
0.0000	0.0000	-2.0948	-0.0000	2.0948	-0.0000
0.0000	2.0928	0.0000	0.0000	-0.0000	2.0928
addMass = origin(2,0,0)					
2.0925	-0.0000	-0.0000	0.0000	-0.0000	-0.0001
0.0000	2.0928	0.0000	0.0000	-0.0000	4.1856
-0.0000	0.0000	2.0948	0.0000	-4.1895	0.0000
-0.0000	-0.0000	-0.0000	0.0000	0.0000	-0.0000
0.0000	-0.0000	-4.1895	-0.0000	8.3791	-0.0000
0.0001	4.1856	0.0000	0.0000	-0.0000	8.3712
addMass = origin(1,1,1)					
2.0926	-0.0000	0.0000	0.0000	2.0926	-2.0926
0.0000	2.0930	-0.0000	-2.0929	0.0000	2.0929
0.0000	0.0000	2.0951	2.0951	-2.0951	-0.0000
-0.0000	-2.0930	2.0951	4.1880	-2.0951	-2.0929
2.0926	-0.0000	-2.0951	-2.0950	4.1877	-2.0926
-2.0926	2.0930	-0.0000	-2.0930	-2.0926	4.1855

Figure 3.10 Effects of shifting the Origin on the Added Mass

Observation:

When the sphere's origin coincides with the origin of the reference frame (0, 0, 0), there are three non-zero added mass elements in x, y, z direction respectively. All of them are about 2.09. The three values should be close to each other because a sphere has three planes of symmetry.

When the sphere's origin is located at (1, 0, 0), added mass in x, y, z direction remain the same. M_{44} remains zero, because there is no kinetic energy transfer to the fluid when the sphere spins about the X-axis.

For off diagonal term, when the origin is moved out by r units, the corresponding value will increase by r . The diagonal terms roll, pitch, yaw, will increase by r^2 . For example, the diagonal pitch term M_{55} for origin (1,0,0) is 2.0948. When the origin become (2,0,0), M_{55} become $8.3781 \approx 2^2 \times 2.0948$.

3.5.4 Effect of changing orientation of the reference frame

Figure 3.11 Effect of changing Orientation of the Reference Frame on Added Mass Matrix: (a) Hemisphere locate at Positive X-axis (b) Hemisphere locate at Negative X-axis

The Figure 3.11 shows how two hemispheres are represented in two different reference frames; one is defined in the positive x region and the other is defined in the negative-x region. Both results have the same added mass value in x, y and z direction (m_{11}, m_{22}, m_{33}). Hence, reversing the axis' sign does not affect the added mass in surge, sway and heave.

3.5.5 Effect of change in the body hydrodynamics shapes

Ballast is used as the example. Three different shapes of ballast are study and their respective added mass is shown below:

Shape 1

0.7677	-0.0000	0.0000	-0.0000	-0.0000	-0.0000
0.0000	7.5612	-0.0000	0.0000	-0.0000	0.0000
0.0000	-0.0000	7.5611	-0.0000	0.0000	0.0000
0.0000	0.0000	-0.0000	-0.0000	0.0000	-0.0000
-0.0000	-0.0000	-0.0000	0.0000	0.3463	0.0000
0.0000	0.0000	0.0000	-0.0000	0.0000	0.3466

Shape 2

0.7114	-0.0000	0.0000	-0.0000	-0.0000	0.0000
-0.0000	7.6219	-0.0000	0.0000	0.0000	-0.0000
0.0000	-0.0000	7.6233	-0.0000	0.0000	-0.0000
-0.0000	0.0000	-0.0000	-0.0000	0.0000	-0.0000
-0.0000	-0.0000	0.0000	0.0000	0.3538	0.0000
0.0000	-0.0000	-0.0000	-0.0000	-0.0000	0.3538

Shape 3

0.4342	-0.0000	0.0000	-0.0000	-0.0000	0.0000
-0.0000	8.2120	-0.0000	0.0000	0.0000	-0.0000
0.0000	0.0000	8.2144	0.0000	0.0000	-0.0000
-0.0000	0.0000	0.0000	0.0000	0.0000	-0.0000
-0.0000	-0.0000	0.0000	0.0000	0.4447	-0.0000
-0.0000	-0.0000	-0.0000	-0.0000	0.0000	0.4446

Figure 3.12 Different Shapes of Ballast and their Added Mass Matrix

The value of M_{11} is reduced from shape 1 to shape 3 as expected. When the ballast is moving in the x-direction, shape 3 will have the smallest added mass, as the hydrodynamic force resistance is the lowest. However, for other directions, the added mass terms increase slightly because their projected areas have increased.

3.5.6 Effect of Scaling Object Size

In this study, the effect of model scaling on the added mass matrix will be investigated. A full-scale, half-scale and quarter-scale ROV II are constructed and the values of the added mass matrix are determined and shown below:

Scale	Added Mass Matrix					
Full-Scale	21.1407	0	0.0621	0	-0.5749	0
	0	51.7021	0	-2.0929	0	-0.3766
	0.0918	0	92.4515	0	0.5870	0
	0	-2.0089	0	3.6191	0	0.0235
	-0.5237	0	0.5592	0	2.6426	0
	0	-0.3783	0	0.0275	0	2.3033
Half-Scale	2.6425	0	0.0077	0	-0.0359	0
	0	6.4626	0	-0.1308	0	-0.0235
	0.0115	0	11.5562	0	0.0367	0
	0	-0.1256	0	0.1131	0	0.0007
	-0.0327	0	0.0350	0	0.0826	0
	0	-0.0236	0	0.0009	0	0.0720
Quarter-Scale	0.3303	0	0.0010	0	-0.0022	0
	0	0.8078	0	-0.0082	0	-0.0015
	0.0014	0	1.4445	0	0.0023	0
	0	-0.0078	0	0.0035	0	0.0000
	-0.0020	0	0.0022	0	0.0026	0
	0	-0.0015	0	0.0000	0	0.0022

The ratio of Full-scale to half-scale

$$\begin{bmatrix} 8 & 0 & 8 & 0 & 16 & 0 \\ 0 & 8 & 0 & 16 & 0 & 16 \\ 8 & 0 & 8 & 0 & 16 & 0 \\ 0 & 16 & 0 & 32 & 0 & 32 \\ 16 & 0 & 16 & 0 & 32 & 0 \\ 0 & 16 & 0 & 32 & 0 & 32 \end{bmatrix} = \begin{bmatrix} 2^3 & 0 & 2^3 & 0 & 2^4 & 0 \\ 0 & 2^3 & 0 & 2^4 & 0 & 2^4 \\ 2^3 & 0 & 2^3 & 0 & 2^4 & 0 \\ \hline 0 & 2^4 & 0 & 2^5 & 0 & 2^5 \\ 2^4 & 0 & 2^4 & 0 & 2^5 & 0 \\ 0 & 2^4 & 0 & 2^5 & 0 & 2^5 \end{bmatrix}$$

The ratio of Full-scale to Quarter-scale

$$\begin{bmatrix} 64 & 0 & 64 & 0 & 256 & 0 \\ 0 & 64 & 0 & 256 & 0 & 256 \\ 64 & 0 & 64 & 0 & 256 & 0 \\ 0 & 256 & 0 & 1024 & 0 & 1024 \\ 256 & 0 & 256 & 0 & 1024 & 0 \\ 0 & 256 & 0 & 1024 & 0 & 1024 \end{bmatrix} = \begin{bmatrix} 4^3 & 0 & 4^3 & 0 & 4^4 & 0 \\ 0 & 4^3 & 0 & 4^4 & 0 & 4^4 \\ 4^3 & 0 & 4^3 & 0 & 4^4 & 0 \\ \hline 0 & 4^4 & 0 & 4^5 & 0 & 4^5 \\ 4^4 & 0 & 4^4 & 0 & 4^5 & 0 \\ 0 & 4^4 & 0 & 4^5 & 0 & 4^5 \end{bmatrix}$$

Let added mass matrix to be composed by four 3 by 3 sub-matrixes, such that $M_A = \begin{bmatrix} M_{11} & M_{12} \\ M_{21} & M_{22} \end{bmatrix}$. If the model is scaled up by factor of r , then each element in M_{11} will scale up by r^3 ; each element in M_{12} & M_{21} will scale up by r^4 ; each element in M_{22} will scale up by r^5 . The results coincide with the work reported in reference [33].

3.5.7 Lower Order Method versus Higher Order Method

There are two different ways to represent the panels. They are the higher order method and lower order method. In this section, the two methods are used to study the main cylinder hull of the ROV II and the results are evaluated.

- Lower Order Method

Figure 3.13 Process in preparing Geometry File for Lower-order Method

Cylinder hull of the ROV II is first drawn in MultiSurf as shown in Figure 3.13 (a). After that, a Geometry Definition File (GDF) as shown in Figure 3.13 (b) is exported from MultiSurf. A panel is defined by four vertexes and each vertex is specified by x, y and z coordinates. A Matlab script has been written to read the GDF file and each panel is plotted as shown in Figure 3.13 (c). The cylinder hull is represented by 800 panels. The calculated added mass of the cylinder is:

7.2667	-0.0000	-0.0000	-0.0000	0.0000	-0.0000
-0.0000	28.2343	-0.0000	0.0000	0.0000	9.0230
-0.0000	0.0000	28.2343	0.0000	-9.0230	0.0000
-0.0000	0.0000	-0.0000	0.0000	0.0000	0.0000
0.0000	-0.0000	-9.0634	-0.0000	2.9685	-0.0000
-0.0000	9.0634	-0.0000	0.0000	0.0000	2.9685

• Higher Order Method

In the higher order method, the geometry of the object is represented by “relational geometry” (RG) framework. The geometry that defined under the framework can be directly accessed by WAMIT through the WAMIT-RGKernel Interface. In the higher order method, a user could specify the overall panel’s size. But this is not an efficient way. Surfaces with flat and simple geometry can be represented with bigger panels without affecting the result. On the other hand, surfaces with large curvature need to be represented by smaller panels to preserve the shape the object.

This problem is solved by specifying the number of panels used to represent a surface individually. The explanation on how WAMIT subdivide the body surface into patches and panels could be found in WAMIT manual section 6.1. A body surface such as the cylinder hull is first defined by “patches”, a smooth continuous surface. Then, the cylinder hull is divided to four patches: cylinder front, cylinder fillet, cylinder body and cylinder back as shown in Figure 3.14.

Figure 3.14 Cylinder Hull drawn using MultiSurf

Each patch is then further sub-divided to a number of rectangular meshes in parametric space. Each mesh is a panel. Although these panels are flat and rectangular in parametric space, they are unrestricted in physical space. In general, these panels are curve surface in the physical space. The number of panels in a patch is defined by specifying the values of N_u and N_v . An example taken from WAMIT manual is shown below:

Figure 3.15 Subdivision of one quadrant of a Cylinder into Patches and Panels.

There are two patches and each patch has $N_u = N_v = 2$ and totals four panels. The cylinder hull is first analyzed using overall panel's size of 0.1m. Then the size is further reduced. The added mass coefficient should converge to the desired value as shown below:

- Panel's Size = 0.100

Number of unknowns in linear system: NEQN = 196

Patch	NU	NV	
1	2	8	cylinder back
2	8	7	cylinder body
3	2	7	cylinder front
4	1	8	cylinder fillet

Added Mass:

6.3733	0.0000	-0.0000	0.0000	0.0000	0.0000
0.0000	25.6143	0.0003	0.0000	-0.0001	8.4451
-0.0000	-0.0003	25.6142	-0.0000	-8.4451	-0.0001
0.0000	-0.0000	-0.0000	0.0000	0.0000	-0.0000
0.0000	0.0001	-8.4451	0.0000	3.3908	0.0000
0.0000	8.4451	0.0001	0.0000	-0.0000	3.3908

- Panel's Size = 0.050

Number of unknowns in linear system: NEQN = 489

Patch	NU	NV	
1	3	16	cylinder back
2	16	13	cylinder body
3	3	13	cylinder front
4	1	16	cylinder fillet

Added Mass:

6.3523	0.0000	-0.0000	0.0000	0.0000	-0.0000
-0.0000	25.6031	0.0001	0.0000	-0.0000	8.4450
-0.0000	-0.0001	25.6033	0.0000	-8.4450	-0.0000
0.0000	-0.0000	0.0000	0.0000	-0.0000	-0.0000
0.0000	0.0000	-8.4450	-0.0000	3.3903	0.0000
-0.0000	8.4449	0.0000	-0.0000	-0.0000	3.3903

- Panel's Size = 0.025

Number of unknowns in linear system: NEQN = 1508

Patch	NU	NV	
1	5	32	cylinder back
2	32	25	cylinder body
3	6	25	cylinder front
4	2	32	cylinder fillet

Added Mass:

6.3406	-0.0000	-0.0000	0.0000	0.0000	-0.0000
-0.0000	25.5974	-0.0000	0.0000	0.0000	8.4445
0.0000	0.0000	25.5975	-0.0000	-8.4446	0.0000
0.0000	0.0000	-0.0000	0.0000	0.0000	0.0000
-0.0000	-0.0000	-8.4446	0.0000	3.3900	-0.0000
0.0000	8.4446	-0.0000	0.0000	0.0000	3.3900

The result shows that the added mass converges after panel's size equal to 0.05m. Further reduction of the panel's size does not change the added mass value. The result obtained from lower order method also confirms the accuracy of the result. In this study, higher order method converges faster than the lower order method. This is because the geometry representation can be transferred to WAMIT from MultiSurf without significant approximations. The WAMIT user manual states two precautions when using higher-order method. First, the iterative method for the solution of the linear system may fail to converge in many cases. The direct or block-iterative solution options are recommended in these cases. Second, the result may be less accurate when processing a geometry that has sharp corners.

3.5.8 ROV II

Figure 3.16 ROV II drawn in MultiSurf

The full-scale model of ROV II was drawn using MultiSurf. After that, the geometry was imported to WAMIT and problem was solved using the high-order panel method. Actually, only half of the ROV II modelled is needed because it has symmetry in XZ plane. Only main components of the ROV II are drawn to reduce the complexity of the computation. Other minor components such as thrusters, lamps and weights are studied separately as individual elements. The results show that their contributions of added mass are small and thus can be ignored.

Added Mass:

21.1407	0	0.0621	0	-0.5749	0
0	51.7021	0	-2.0929	0	-0.3766
0.0918	0	92.4515	0	0.5870	0
0	-2.0089	0	3.6191	0	0.0235
-0.5237	0	0.5592	0	2.6426	0
0	-0.3783	0	0.0275	0	2.3033

Eigen-value of the added mass matrix

$$Eig(M_A)=[21.1565 \quad 2.6227 \quad 92.4547 \quad 51.7914 \quad 3.5319 \quad 2.3004]^T \Rightarrow M_A > 0$$

Convergence Test

Figure 3.17 Convergence Test: Added Mass for ROV II

The added mass matrix M_A for ROV II is positive definite. This is as expected for a fully submerged vehicle [4]. The off-diagonal terms of the added mass matrix are relatively small when compared with the diagonal terms. This is true for most underwater vehicles with roughly 3 planes of symmetry. The added mass for surge is smaller than the added mass for sway; added mass for sway is smaller than added mass for heave ($m_{11} < m_{22} < m_{33}$). This could be explained by the geometry of the ROV II where surge has the smaller projection area and it is followed by sway and then heave.

3.6 CHAPTER SUMMARY

In this chapter, the added mass matrix for the ROV II has been determined using WAMIT, a CFD software based on potential flow theory and panel method. The added mass for surge, sway and heave motion is 21kg, 51kg and 93 kg respectively. After considering the added mass, the effective inertia for ROV II in heave motion is almost double, increasing from 115kg to 208kg. As such, the added mass forces are quite significant and thus cannot be neglected.

A series of tests had been conducted from section 3.5.2 until section 3.5.7 to verify the result from WAMIT. The calculated added mass from WAMIT is highly accurate when comparing with the theoretical results for simple body shapes. In the chapter 5, once again, the added mass terms obtained from WAMIT are in good agreement with the experimental data. As a conclusion, the added mass matrix for ROV II obtained from WAMIT is highly reliable.

CHAPTER 4. HYDRODYNAMICS DRAG

4.1 INTRODUCTION

While moving, a submerged body experiences a resultant force due to the interaction between the body and the surrounding water. The resultant force acting on the submerged body can be resolved into 2 components which are orthogonal to each other. The component force that is normal to the upstream velocity is known as the *lift* and the component force that is in the direction of the upstream velocity is known as the drag.

Figure 4.1 Lift and Drag

Drag force can further be classified into two categories, known as the frictional drag and pressure drag.

Frictional Drag

Frictional drag comes from the friction between the fluid and the surfaces over which it is flowing. The frictional drag is the summation of the wall shear stresses acting on the total surface of the submerged body. This friction is associated with the development of boundary layers, and it scales with the Reynolds number. Frictional drag is important for attached flows (that is, there is no separation), and it is related to the surface area exposed to the flow.

Pressure Drag

Pressure drag comes from the eddying motions that are set up in the fluid by the passage of the body. This drag is associated with the formation of a wake, which can be readily seen behind a passing boat, and it is usually less sensitive to Reynolds Number. The pressure drag is the summation of pressure acting on the total surface of the submerged

body. Pressure drag is important for separated flows, and it is related to the cross-sectional area of the body.

Flow Separation

Flow separation occurs when the boundary layer encounters a sufficiently large adverse pressure gradient. The fluid flow becomes detached from the surface of the object. The prediction of flow separation is important because the size of the wake is determined by the location of flow separation. The size of the wake will determine the magnitude of the pressure drag.

Figure 4.2 Flow Separation (Taken from [15])

*Note: please refer to appendix C for more detail discussion on hydrodynamics drag and appendix E on typical stages in CFD.

4.2 PRE-PROCESSING

4.2.1 Physical Model

Figure 4.3 Physical Model and Computer Simulation Model

Figure 4.3 case 1 shows the ROV moving forward at constant speed U . The fluid domain is static with no current flow. Computer simulation can be used to study the forces acting on the vehicle when it moves at constant speed. However, in computer simulation, the ROV is static and the flow is made to move at constant speed in the opposite direction (see Figure 4.3 case 2). Since the drag force depends only on the relative motion between the ROV and the fluid, the result obtained from the two situations will be the same. The benefit of keeping the ROV static is that boundary conditions can setup more easily for computer simulation. Moving ROV, on the other hand, creating moving meshes that is more complex to handler.

4.2.2 Mathematic Model

- **Turbulence Model**

In most ROV operations, considering its scale and speed, the typical Reynolds number will be greater than 1.0×10^6 , indicating turbulence in flow field. In turbulence flow, the fluid motion is characterized by a highly random, unsteady three-dimensional flow. The turbulent length and time scales are much smaller than the smallest finite volume meshes that are used in most numerical analysis. To generate such a small mesh is beyond the computing power currently available. The problem is solved by employing turbulence model.

In general, turbulence models seek to modify the original unsteady Navier-Stokes equations with the Reynolds Averaged Navier-Stokes (RANS) equations. In the resulting Statistical Turbulence Model, each variable is separated into a mean and a fluctuating component. Then the equations are time averaged over a time scale that is sufficiently longer than the time scale for the fluctuating components. The interval of the time scale is so selected that the resulting average term is independent of the length of the interval [34].

ANSYS CFX was used because of the availability of Shear-Stress-Transport (SST) model. Currently, the $k-\omega$ based Shear-Stress-Transport (SST) model is the most prominent two-equations models to predict flow transition from laminar to turbulent [35]. It was designed to give highly accurate predictions of the onset and the amount of flow separation under adverse eddy-viscosity.

- **Wall Model**

At high Reynolds Number, the wall-stress cannot be calculated correctly just by taking gradients on the coarse grids or meshes. The boundary layer is often too thin to be correctly represented by the meshes. Hence, there is a need to develop a model to approximate the wall-boundary condition. The wall-function used in ANSYS CFX is an extension of the method of Launder and Spalding [36]. In their log-law method, the near wall tangential velocity is logarithmically related to the wall shear stress. This relationship helps to predict the velocity profile in boundary layer and hence fine meshes are not necessary.

4.2.3 Geometrical Model

Figure 4.4 ROV II Model in SolidWork

The geometrical model of the ROV II had been drawn using computer-aided design software SolidWork. Not all components on the ROV II are drawn. The main purpose is to keep the geometry of the ROV simple and manageable for mesh generation. However, the main components of ROVs such as two cylinder hulls, two floats, frame and thrusters are modeled so that the characteristic of the flow is as realistic as possible.

4.2.4 Meshing Domain

Figure 4.5 CFX-Mesh. The domain meshes consist of tetrahedral, prisms and pyramids

- **Inflation Layer**

In near-wall regions, boundary layer effects give rise to velocity gradients that are greatest in the direction normal to the surfaces. Computationally, meshes in these regions require the elements to have high aspect ratios. If tetrahedral elements are used, then a prohibitively fine surface mesh may be required to avoid generating highly distorted tetrahedral elements at the face. CFX-Mesh overcomes this problem by using prisms to create a mesh that is finely resolved normal to the wall, but coarse parallel to it. This element is called inflation layer. Inflation can greatly improve accuracy, particularly in a model where turbulence is significant. Inflation should always be used when one is interested in lift, drag or pressure drop in the model [37]. There are five inflation layers in Figure 4.5.

- **Y-Plus Dimensionless Number**

The velocity next to a no-slip wall boundary changes rapidly from a value of zero at the wall to the free stream value at a short distance away from the wall. This layer of high velocity gradient is known as the boundary layer. In general, generated meshes are not fine enough near a wall to resolve the velocity profile in the boundary layer accurately. Wall functions such as log-law can be used in these cases to apply an assumed functional shape of the velocity profile. One indicator of the physical closeness of the first node to

the wall is the dimensionless wall distance y^+ . It is good practice to examine the values of y^+ at the end of simulation. At the lower limit, a value of y^+ less than or equal to 11 indicates that the first node is within the laminar sub-layer of the boundary flow. Values larger than this indicate that an assumed logarithmic shape of the velocity profile is being used to model the boundary layer portion between the wall and the first node. Ideally, it should be confirmed that there are several nodes (3 or more) resolving the boundary layer profile. If this is not observed, it is highly recommended that more nodes be added near the wall surfaces in order to improve simulation accuracy.

4.3 BOUNDARY CONDITION & MODEL ENVIRONMENT SETUP

4.3.1 Domain

The flow in the domain is expected to be turbulent and approximately isothermal. The Shear Stress Transport (SST) turbulence model with automatic wall function treatment will be used because of its highly accurate predictions of flow separation. To take advantage of the SST model, the boundary layer should be resolved with at least 10 mesh nodes. This is done by inspecting the y^+ value on the surface of the ROV that must be around one.

Water at 20 degree Celsius is used as the fluid. Water is modeled as an incompressible flow with fixed density. Thus, the fluid properties of the water are unchanged throughout the flow simulation. The reference pressure is set at one atmosphere.

The main difficulty is to determine the fluid domain dimension. In order to study drag force acting on the ROV in an unbound fluid domain, an infinite large fluid domain is needed. However, this is not practical. The fluid domain dimensions of five times the length, width and height of the ROV as shown in Figure 4.6 are chosen. Uncertainty analysis has to be carried out to change the domain size to see any variations in results.

Setting	Value
Basic Settings > Fluids List	Water
Domain Models > Pressure > Reference Pressure	1 [atm]
Heat Transfer > Option	Isothermal
Heat Transfer > Fluid Temperature	20 [C]
Turbulence > Option	Shear Stress Transport

Figure 4.6 Meshing Domain

4.3.2 Inlet

A flow speed is to be specified at the inlet boundary as shown in Figure 4.7. Physically, the flow upstream is uniform. It is set to be normal to the plane and has speed of 0.1m/s. Turbulence properties at the inlet flow is hard to specify because the turbulent kinetic energy and dissipation data are unknown. Therefore, their values have to be specified using sensible engineering assumptions and the effect of the choices made should be examined by sensitivity tests.

In this study, the turbulent properties are specified using turbulent intensity, I

$$I=\frac{u'}{U}$$

(4-1)

where u' is the root-mean-square of the turbulent velocity

$$u' = \sqrt{\frac{1}{3}(u_x'^2 + u_y'^2 + u_z'^2)} \tag{4-2}$$

and U is the mean velocity of the upstream flow

$$U = \sqrt{(U_x^2 + U_y^2 + U_z^2)} \tag{4-3}$$

To do this accurately it is good to have some form of measurements or previous experience to base the estimate on. Here are a few examples of common estimations of the incoming turbulence intensity:

High-turbulence case: High-speed flows inside complex geometries like heat exchangers and flow inside rotating machinery (turbines and compressors) have typically turbulence intensity between 5% and 20%

Medium-turbulence case: Flows in not-so-complex devices like large pipes, ventilation flows etc. or low speed flows have typically turbulence intensity between 1% and 5%

Low-turbulence case: External flow across cars, submarines and aircrafts in otherwise stationary fluid domain and flow generated by very high-quality wind tunnels have typically the turbulence intensity well below 1%.

The external flow over an underwater vehicle has turbulence intensity well below 1%. So, turbulent intensity of 0.1% is selected for this study.

Setting	Value
Boundary Type	Inlet
Location	Inlet

Setting	Value
Flow Regime > Option	Subsonic
Mass and Momentum > Option	Normal Speed
Mass and Momentum > Normal Speed	0.1 [m s ⁻¹]
Turbulence > Option	Intensity
Turbulence > Intensity	0.1%

Figure 4.7 Inlet Settings

4.3.3 Outlet

The outlet boundary condition is selected to have a weak influence on the upstream flow. Specifying static pressure at the outlet plane is recommended by the [38].

Setting	Value
Boundary Type	Outlet
Location	Outlet
Mass and Momentum > Option	Static Pressure
Mass and Momentum > Relative Pressure	0 [Pa]

Figure 4.8 Outlet Settings

4.3.4 Free Slip Wall Boundary

The top, bottom and side surfaces of the rectangular fluid domain can be modeled using free slip wall boundary conditions.

On free slip walls, the shear stress is set to zero so that the fluid is not retarded.

The velocity normal to the wall is also set to zero.

The velocity parallel to the wall is calculated during the solution.

This is not an ideal boundary condition, as the flow around the body will be affected by its close proximity to the walls. A much larger domain should be used to minimize the effect of the walls.

Setting	Value
Boundary Type	Wall
Location	FreeWalls
Wall Influence On Flow > Option	Free Slip

Figure 4.9 Free-Slip Boundary Condition

4.3.5 Non-Slip Wall Boundary

Due to fluid viscosity, the flow speed on the vehicle’s surfaces is zero. Hence, a non-slip wall boundary condition is applied on the vehicle’s body.

Setting	Value
Boundary Type	Wall
Location	Body
Wall Influence On Flow > Option	No Slip

Figure 4.10 Non-Slip Boundary Condition

4.3.6 Symmetry Plane

The ROV II has symmetry on XZ plane. The used of symmetry will greatly reduce the computational time and resources. At symmetry plane, the gradients perpendicular to the plane are zero.

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry

Tab	Setting	Value
	Location	SymP

Figure 4.11 Symmetry Boundary Condition

4.3.7 Initial Value

Initial values for all variables can be set before the solution can start. For a steady-state calculation, the initial variable values serve to give the ANSYS CFX-Solver a flow field from which to start its calculations. Convergence is more rapidly achieved if sensible initial values are provided. However, converged results should not be affected by the initialization. Unless there is information about the turbulent eddy dissipation, it is recommended to use the automatic initial guess.

Table 4.1 Initial condition of the flow domain

Setting	Value
Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
Initial Conditions > Cartesian Velocity Components > U	0.1 [m s ⁻¹]
Initial Conditions > Turbulence Eddy Dissipation	(Selected)

4.3.8 CFX’s Solver Control

The selection of an appropriate timestep size is essential to obtain good convergence rates for the simulations. Physical timestep is used to provide sufficient relaxation of the equation non-linearity so that a converged steady-state solution is obtained.

Physical timestep that are too large are characterized by oscillatory convergence or results that do not converge. Small timestep are characterized by very slow, steady convergence.

For advection-dominated flows, the physical timestep size should be some fraction of a length scale divided by a velocity scale. A good approximation is the dynamical time for

the flow. This is the time taken for the flow to make its way through the fluid domain. A reasonable estimate is easy to make based on one third of the length of the fluid domain L and the mean velocity U . In this study,

$$\Delta t = \frac{L}{3U} = \frac{10m}{3 * 0.5ms^{-1}} = 6.67s$$

So, time step is set to 5 second to obtain fast and convergence result.

Table 4.2 CFX’s Solver Control

Setting	Value
Convergence Control > Max. Iterations	60
Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
Convergence Control > Fluid Timescale Control > Physical Timescale	5 [s]
Convergence Criteria > Residual Target	1e-05

4.4 RESULTS & DISCUSSION

4.4.1 Case Study: Sphere

The accuracy of the numerical approach was established through comparison of the current study with empirical and other computational results. Flow at three different Reynolds Numbers has been select for comparison. They are $Re= 100$, where flow is in the steady and axis-symmetric regime; $Re = 300$, where the flow is in the laminar periodic vortex-shedding regime; $Re = 1.1 \times 10^6$ where flow is highly unsteady and boundary layer may be turbulent.

Figure 4.12 Mesh Domain of the Sphere.

Table 4.3 Mesh Statistics and Mesh Settings

No of Mesh Elements	1255570
Face Spacing	0.001m ²
Face Spacing Expansion Factor	1.2
No of Inflation Layer	35
Thickness Multiplier	1
Inflation Expansion Factor	1.2

Face spacing determine the surface area of a mesh that in contact with the sphere’s surface. Therefore, it must be small enough to capture the geometry of the sphere. The inflation layer is important in the capture of the boundary layer flow and prediction of the flow separation. Hence, the number of inflation layers need to be increase so that the y^+ value is close to unity on the surface of the sphere.

- $Re = 100$

Figure 4.13 Streamline Plot at $Re = 100$

Table 4.4 Flow parameters at Re = 100. Kim denotes numerical results by Kim and Choi [39], and

Empirical denotes $C_d = \frac{24}{Re} + \frac{6}{1 + \sqrt{Re}} + 0.4$

Re	Data	Cd
100	Current study	1.26
100	Kim	1.09
100	Empirical	1.18

Figure 4.14 Pressure coefficient around the Sphere and Wall Shear around the Sphere, Re = 100

The Figure 4.14 (a) presents the plot of pressure around the circumference of the sphere. Zero degree indicates front tip and 180 degree indicates rear tip. The stagnation point occurs at the front tip thus the pressure is the highest. As the flow increase in speed, the pressure drops until its minimum at around 90 degrees. After 90 degrees, flow starts to slow down and the pressure increases slowly. Flow starts to separate from the surface at around 130 degrees. This can be seen from Figure 4.14 (b) where shear stress crosses the x-axis. Wake is formed after the separation point and the pressure in wake remain low. The pressure drag is the result of the pressure difference between the front and rear of the sphere.

- **Re = 300**

Figure 4.15 Streamline Plot at Re = 300

Table 4.5 Drag Coefficient at Re = 300. Patel denotes numerical results by Johnson and Patel [40],

and Empirical denotes $C_d = \frac{24}{Re} + \frac{6}{1 + \sqrt{Re}} + 0.4$

Re	Data	Cd
300	Current study	0.814
300	Patel	0.626
300	Empirical	0.826

- **Re = 1.1 x 10⁶**

Figure 4.16 Streamline Plot at Re = 1.1 x 10⁶

Table 4.6 Drag Coefficient at $Re = 1.1 \times 10^6$. experimental result from Achenbach [41], DES results from Constantinescu [42]

Re	Data	Cd
1.1×10^6	Current study	0.117
1.1×10^6	Experiment	0.12-0.14
1.1×10^6	DES	0.084

The drag coefficients obtained in this study closely matched those published results for Reynolds Number 100, 300 and 1.1×10^6 . These give the assurance that the procedures in setting up the simulation are done correctly. For the study of sphere’s drag, it is observed that the y^+ value on the sphere surface must have value close to unity to obtained accurate result. Therefore, mesh need to be refined to obtained unity y^+ value.

4.4.2 Scaled Model of the ROV II

Figure 4.17 CAD Modeling of Scaled Model of ROV II and its Surface Mesh Plot

The CAD model of the scaled model of ROV II is drawn using DesignModeler in ANSYS™. The scaled model also has symmetry in XZ-plane. So, only half of the vehicle is used in Simulation as shown in Figure 4.17 (b).

Figure 4.18 Cut Surface Mesh Plot at the Middle Plane of the Cylinder Hull

Figure 4.18 (a) illustrates the effect of face spacing expansion factor. One could observe that the mesh sizes increase exponentially away from the surface of the cylinder. This is important in having a smaller number of meshes far away from the body but keeping the meshes fine near to the body. Figure 4.18 (b) is a zoom image of Figure 4.18 (a) to show the inflation layers. The boundary layer is contained within a thin layer in contact with the body. Thus, inflation layer is important in capturing the flow in boundary layer by providing finer mesh in that region.

Figure 4.19 Streamline Plot of the Scaled Model

Figure 4.19 illustrates the formation of wake at the rear of the vehicle. The wake creates low pressure region at the rear end and thus high pressure in the front resist the vehicle motion.

Figure 4.20 Iteration History of the Normal Force and Tangential Force acting on the Scaled Model

Figure 4.20 shows that both the normal force and the tangential force acting on the scaled model converge to a constant value. This is an important indication that the flow has reach steady state. If the forces keep oscillating, one may need to consider unsteady flow analysis.

Figure 4.21 Drag Force for Scaled Model in Surge

Figure 4.21 illustrates the drag force experienced by the scaled model when moving in the surge direction. From polynomial fitting, we have a linear damping, $K_L = 0.1643$ and quadratic damping, $K_Q = 10.88$.

Figure 4.22 Drag Coefficient versus Reynolds Number

In Figure 4.22, the drag coefficient converges to a constant value around one. This is an important indication that the point separation of the flow and thus formation of wake are

similar regardless of the flow velocity. Therefore, drag coefficient is insensitive to the change of the Reynolds Number within the operational range of the ROV II.

Figure 4.23 Comparison of experiment and CFD result of Drag

The experiment results shown in Figure 4.23 are obtained in Chapter Five using free decay tests. The CFD drag is consistently lower than the experimental drag. Quadratic damping, which is product of pressure drag, is predicted with high degree of accuracy but the linear damping, which is product of skin friction, is underestimated. This is expected because in CFD, the surface of the scaled model was assumed smooth, thus the skin friction is smaller. The surface roughness of the vehicle is difficult to define accurately in practice.

Figure 4.24 Contribution of Drag due to different components of Scaled Model

One of the most important findings in this study is that the removal of the ROV frame causes a 56.8% of reduction in the total drag. The drag of case 3 and 4 are relatively low because the geometry is streamline in shape. The flow attaches longer on the body surface and the corresponding wake formation is smaller. The addition of frames triggers the early flow separation and the corresponding wake formation is larger. Thus, ROV frame increase the pressure drag dramatically.

4.4.3 NTU ROV II

Figure 4.25 Meshing Domain

The mesh domain has 2,219,437 elements in total. They consist of 1,067,375 tetrahedras, 1,140,046 wedges and 12,016 pyramids.

Figure 4.26 Pressure Distribution on the ROV II Body.

Figure 4.26 shows that the front of the ROV II which facing the flow experiences the largest pressure force. The total pressure difference between front and back causes the pressure drag, which is the dominant drag acting on the bluff body like body.

Figure 4.27 Streamline and Vector Plot at the Centre Plane of the Cylinder Hull. Flow speed: 0.5m/s

Figure 4.27 illustrates clearly the formation of wake at the rear of the ROV II. Wake is the turbulence region that is characterized by the formation of eddies and vortexes.

Figure 4.28 Drag Force as the function of Velocity in Surge, Sway and Heave

The drag force in surge, sway and heave direction are plotted against the velocity in Figure 4.28. The drag value in heave is plotted between 0-0.2m/s because the ROV II only operates within this speed in actual operation. On the other hand, the vehicle operates between 0-0.5m/s in the surge and sway direction.

As expected, the vehicle has largest drag in heave follow by sway and surge. Heave has the largest frontal area normal to the flow direction of about 0.9m² compare to 0.5m² for sway and 0.45m² for surge. The drag in sway is only slightly larger than the drag in surge. This may be explained by the relative size of the frontal areas. From the second-order polynomial fit of the graph, the linear damping and quadratic damping for respective directions are obtained. The results are tabulated in following table:

Table 4.7 The drag coefficient of ROV II in its four principle motions

Direction	Surge		Sway		Heave		Yaw	
Range	(0-0.5m/s)		(0-0.5m/s)		(0-0.2m/s)		(0-0.5rad/s)	
Parameter	K_L	K_Q	K_L	K_Q	K_L	K_Q	K_L	K_Q
ANSYS	11.863	108.45	19.64	108.24	2.3756	351.98	0	10.39
CFX

Figure 4.29 Drag Torque s a Function of Square Angular Velocity

The results suggest that the linear damping in yaw can be ignored and the quadratic damping alone is sufficient to describe the drag correctly. The quadratic damping has value of 10.392 Nm/(rad/s)².

4.5 CHAPTER SUMMARY

In this chapter, the drag coefficients for ROV II have been determined by ANSYS CFX. It was carried out by following through the typical stages in CFD computation which include problem formulation, modelling, setting of boundary conditions and computation. Each stage is verified by using sphere as a case study. The result of current study matches well with the empirical result and other independent studies. Similar methods are used to study the drag of ROV II in surge, sway, heave and yaw direction.

CHAPTER 5. FREE DECAYS TEST

5.1 INTRODUCTION

In the previous chapters, the parameters associated with the hydrodynamics forces acting on the ROV II had been estimated using CFD simulations. However, as stated in CFD guidelines [38] these parameter values should be corroborated by other means. The accuracy of the results can only be truly determined if the results can be independently confirmed using different approach. Therefore, in this chapter, verification of the added mass parameter and the drag force coefficient will be done by comparing the value predicted by CFD and that to be obtained experimentally using the scaled down model of the ROV II (Hereafter will be referred to as the scaled model). By applying laws of Similitude, the hydrodynamics parameters of the scaled model can be scaled up to predict the corresponding values for the true model.

The hydrodynamics parameters can be extracted from an experiment in which the scaled model performs a free decay motion in water. The scaled model will oscillate in a simple harmonic motion in water when it is displaced from its equilibrium (see Figure 5.1). Due to the hydrodynamics forces that resist the motion, the amplitude of the swing will decay over time. The hydrodynamics forces could be deduced from the time history of the motion. As the scaled model is allowed to oscillate freely in the water tank, the experiment is classified as a free decay test.

Figure 5.1 Pendulum Motion in Water

The proposed method has the following advantages:

- The motion of the pendulum is restricted within the plane and has only one degree of freedom. The position of pendulum is fully described by variable θ . As the motion is highly constrained, the dynamics equation of motion would be able to represent the motion correctly. Thus, a reasonably accurate result can be obtained.
- The variable θ can be measured quite easily using a potentiometer or an encoder. However, in this experiment, the angle θ is obtained through an image processing method, which is also very simple and reasonably accurate.
- The experiment setup is simple and is low cost as compared with the building cost of a water tunnel facility and the installation of a Planar Motion Measurement (PMM) equipment.

5.2 THEORY

5.2.1 Nomenclature

M	Mass of the scaled Model
m_a	Added Mass in single DOF
g	Gravity
B	Buoyancy
θ	Angle of rotation of the pendulum
r	Length of the pendulum (radius)
K_L	Linear Damping Coefficient
K_Q	Quadratic Damping Coefficient
F_{rod}	Tension Force from the rod
F_H	Hydrodynamics Force

v	Tangential Velocity
-----	---------------------

5.2.2 Dynamics Equation

The pendulum is made up of a rod and the scaled model attached at the end and submerged in water. The model moves in a circular path of radius r , which is the length of the pendulum as shown in the Figure 5.2:

Figure 5.2 Free Body Diagram of the Pendulum swinging in the Water

As shown in Figure 5.3, from the earth-fix frame point of view, the model is rotating about the pivot point. However, from the point of view of a body-fix frame, it only moves in the surge (X) direction at any instance. In other word, the model has only velocity component in the surge direction and zero velocity in the heave (Z) direction.

Figure 5.3Transformation form earth fixed frame to body fixed frame

Kinematics

Using the SNAME (1950) notation for marine vessels

Earth Fix Frame: position and Euler angles $(x, y, z, \phi, \theta, \psi)$

Body Fix Frame: Linear and angular velocities (u, v, w, p, q, r)

Force and Moments (X, Y, Z, K, M, N)

Figure 5.4 Earth-Fixed Reference Frame and the Body-Fixed Reference Frame

As given in (2.18) of Fosson [5]:

$$\dot{p}^n = R_b^n(\theta)v_o^b \tag{5-1}$$

where $\dot{p}^n = [\dot{x} \ \dot{y} \ \dot{z}]^T$ are velocities defined in earth-fixed frame

$$R_b^n = \begin{bmatrix} c\theta & 0 & s\theta \\ 0 & 1 & 0 \\ -s\theta & 0 & c\theta \end{bmatrix} \text{ is the rotational transformation matrix}$$

$v_o^b = [u \ v \ w]^T$ are velocities defined in body-fixed frame

From the Figure 5.2, we have $\dot{p}^n = \begin{bmatrix} V \cdot c\theta \\ 0 \\ -V \cdot s\theta \end{bmatrix}$

$$\text{Then } v_o^b = \begin{bmatrix} u & v & w \end{bmatrix}^T = (R_b^n)^{-1} \dot{p}^n = \begin{bmatrix} c\theta & 0 & -s\theta \\ 0 & 1 & 0 \\ s\theta & 0 & c\theta \end{bmatrix} \begin{bmatrix} V \cdot c\theta \\ 0 \\ -V \cdot s\theta \end{bmatrix}$$

$$v_o^b = \begin{bmatrix} u & v & w \end{bmatrix}^T = \begin{bmatrix} V(c^2\theta + s^2\theta) \\ 0 \\ V(c\theta s\theta - c\theta s\theta) \end{bmatrix} = \begin{bmatrix} V \\ 0 \\ 0 \end{bmatrix}$$

As given in (2.24) of Fossen [5]:

$$\dot{\mathcal{G}} = T_{\mathcal{G}}(\mathcal{G}) w_{nb}^b \quad (5-2)$$

where $\dot{\mathcal{G}} = \begin{bmatrix} \dot{\phi} & \dot{\theta} & \dot{\psi} \end{bmatrix}^T$, angular velocities define in earth-fixed frame

$$T_{\mathcal{G}}^{-1}(\mathcal{G}) = \begin{bmatrix} 1 & 0 & -s\theta \\ 0 & c\phi & c\theta s\phi \\ 0 & s\phi & c\theta c\phi \end{bmatrix}, \text{ transformation matrix}$$

$w_{nb}^b = \begin{bmatrix} p & q & r \end{bmatrix}^T$, angular velocities define in body-fixed frame

$$\text{Then } w_{nb}^b = \begin{bmatrix} p & q & r \end{bmatrix}^T = T_{\mathcal{G}}^{-1}(\mathcal{G}) \dot{\mathcal{G}} = \begin{bmatrix} 1 & 0 & -s\theta \\ 0 & c\phi & c\theta s\phi \\ 0 & s\phi & c\theta c\phi \end{bmatrix} \begin{bmatrix} 0 \\ \dot{\theta} \\ 0 \end{bmatrix} = \begin{bmatrix} 0 \\ \dot{\theta} \\ 0 \end{bmatrix}$$

$$\therefore \phi = 0$$

$$\text{From the above derivation, it is proven that } \begin{bmatrix} u \\ v \\ w \\ p \\ q \\ r \end{bmatrix} = \begin{bmatrix} V \\ 0 \\ 0 \\ 0 \\ \dot{\theta} \\ 0 \end{bmatrix}$$

Therefore, in the body-fixed reference frame, the scaled model can only move in a pure surge and pitch motion. The added mass and damping coefficients are defined in body fix frame and the hydrodynamic forces acting on the model are given by:

$$\text{Such that, } F_H = m_a \ddot{x} + K_L \dot{x} + K_Q |\dot{x}| \dot{x}$$

Using Newton's second law of motion in the surge direction:

$$\sum F_x = M \ddot{x}$$

$$-mg \sin \theta + B \sin \theta - m_a \ddot{x} - K_L \dot{x} - K_Q |\dot{x}| \dot{x} = M \ddot{x}$$

Rearranging the equation gives:

$$(B - mg) \sin \theta - K_L \dot{x} - K_Q |\dot{x}| \dot{x} = (M + m_a) \ddot{x}$$

$$\ddot{x} = \frac{(B - mg)}{(M + m_a)} \sin \theta - \frac{K_L}{(M + m_a)} \dot{x} - \frac{K_Q}{(M + m_a)} |\dot{x}| \dot{x}$$

For a rotational motion, $\dot{x} = r \dot{\theta}$, also $\ddot{x} = r \ddot{\theta}$

$$r \ddot{\theta} = \frac{(B - mg)}{(M + m_a)} \sin \theta - \frac{K_L}{(M + m_a)} r \dot{\theta} - \frac{K_Q}{(M + m_a)} |r \dot{\theta}| r \dot{\theta}$$

$$\ddot{\theta} = \frac{(B - mg)}{(M + m_a) r} \sin \theta - \frac{K_L}{(M + m_a)} \dot{\theta} - \frac{K_Q \cdot r}{(M + m_a)} |\dot{\theta}| \dot{\theta}$$

$$\text{Let } \alpha = \frac{(B - mg)}{(M + m_a) r}, \beta = \frac{K_L}{(M + m_a)} \text{ and } \gamma = \frac{K_Q \cdot r}{(M + m_a)}$$

$$\text{Then } \ddot{\theta} = \alpha \cdot \sin \theta - \beta \cdot \dot{\theta} - \gamma \cdot |\dot{\theta}| \dot{\theta}$$

(5-3)

5.2.3 Least Square Algorithm [27]

Equation (5.3) can be solved for α, β, γ by using least square method:

$$\underbrace{\begin{bmatrix} \ddot{\theta}_1 \\ \ddot{\theta}_2 \\ \vdots \end{bmatrix}}_y = \underbrace{\begin{bmatrix} \sin \theta_1 & \dot{\theta}_1 & |\theta_1| \dot{\theta}_1 \\ \sin \theta_2 & \dot{\theta}_2 & |\theta_2| \dot{\theta}_2 \\ \vdots & \vdots & \vdots \end{bmatrix}}_{H(x)} \underbrace{\begin{bmatrix} \alpha \\ \beta \\ \gamma \end{bmatrix}}_{\theta} + error$$

subscript $i = 1, 2, 3 \dots$ represents the number of samples collected from experiment

$$\text{Result, } \hat{\theta}_{LS} = (H^T H)^{-1} H^T y \quad (5-4)$$

$$\text{Standard deviation, } \hat{\sigma}_{\theta} = \sqrt{\text{diag}(\text{cov}(\hat{\theta}_{LS}))}, \quad (5-5)$$

$$\text{where } \text{cov}(\hat{\theta}_{LS}) = \hat{\sigma}^2 (H^T H)^{-1}$$

$$\hat{\sigma}^2 = \frac{(y - H \hat{\theta}_{LS})^T (y - H \hat{\theta}_{LS})}{\dim(y) - \dim(\theta)} \quad (5-6)$$

5.2.4 Similitude [43]

Similitude Law states that two different systems with similarity in behaviour will have equal similarity in parameters. In other word, if the flow pattern for two different systems is similar, it is possible to predict the parameters for the true model by using the data from the scaled model.

In order to have similarity in flow, the following criteria are required:

Geometry similarity: The scaled model must have the identical shape as the true model. The angles and length ratio must be maintained. In this study, the size ratio of scaled down model to the true ROV II model is 0.3:1.

Kinematics similarity: Fluid flow of both the model and actual vehicle must undergo similar time rates of change in motions. (Fluid streamlines are similar). This is fulfilled when the scaled model and true model share similar displacement ratios and velocity ratios.

Dynamics similarity: Ratios of all forces acting on corresponding fluid particles and boundary surfaces in the two systems are constant. This is fulfilled when the scaled model and full model share similar forces ratios, stress ratios, and pressure ratios.

- Determining the drag Coefficient

From the dimensional analysis and by using Buckingham π theorem, the drag force acting on a submerged object is:

$$D = \frac{1}{2} \rho C_d A V^2 \quad (5-7)$$

Where

D , Drag Force

V , velocity

ρ , fluid density

C_d , drag coefficient

A , frontal area

From reference [44], the drag coefficient for a submerged object is a function of Reynolds Number. So, the similitude is satisfied by having the same Reynolds Number for the two flows. Reynolds Number is defined as:

$$Re = \frac{\rho V L}{\mu} \quad (5-8)$$

where ρ is fluid density; V is velocity; L is characteristic length; μ is viscosity.

Let the subscript m represent the scaled model and subscript t for true model. Equating the Reynolds Number for scaled model and the true model:

$$\begin{aligned} Re_m &= Re_t & (5-9) \\ \frac{\rho_m V_m L_m}{\mu_m} &= \frac{\rho_t V_t L_t}{\mu_t} \\ \Rightarrow V_m &= V_t \times \left(\frac{\rho_t}{\rho_m} \right) \times \left(\frac{L_t}{L_m} \right) \times \left(\frac{\mu_t}{\mu_m} \right) \\ &= V_t \times \left(\frac{1}{1} \right) \times \left(\frac{1}{0.3} \right) \times \left(\frac{1}{1} \right) = 3.33 V_t \end{aligned}$$

Since water is used as the working fluid for both cases, the density and viscosity are identical. In order to have the same drag coefficient, the model must move around three times faster than the true model. However, CFD study using the range of the flow speed had been discussed in previous chapter and the results shows that the drag coefficients are nearly constant for the range Reynolds Numbers studied. From equation (5-7), the ratio of drag force experience by the true model to the drag force experience by the scaled model at the same speed is:

$$\frac{D_t}{D_m} = \frac{\frac{1}{2} \rho C_d A_t V^2}{\frac{1}{2} \rho C_d A_m V^2} = \frac{A_t}{A_m} = \left(\frac{L_t}{L_m} \right)^2 = \left(\frac{1}{0.3} \right)^2 = 11.111 \quad (5-10)$$

For rotational drag [45],

$$\begin{aligned} T_d &= \sum_i r_i f_{di} \\ &= \sum_i r_i \left(\frac{1}{2} C_d \rho A_{pi} r_i^2 \dot{\theta}^2 \right) = C_D \rho A_p L^3 \dot{\theta}^2 \\ \text{So, } \frac{T_t}{T_m} &= \frac{C_D \rho A_{pt} L_t^3 \dot{\theta}^2}{C_D \rho A_{pm} L_m^3 \dot{\theta}^2} = \frac{A_{pt} L_t}{A_{pm} L_m^3} = \left(\frac{L_t}{L_m} \right)^5 = \left(\frac{1}{0.3} \right)^5 = 411.52 \end{aligned} \quad (5-11)$$

- Determining the added mass coefficient

For the added mass terms, the scaling factor from the scaled model to real ROV II has been explained in section 3.5.6. Each element in the added mass coefficient matrix can be scaled up as indicated below.

$$\begin{bmatrix} (R)^3 & (R)^3 & (R)^3 & (R)^4 & (R)^4 & (R)^4 \\ (R)^3 & (R)^3 & (R)^3 & (R)^4 & (R)^4 & (R)^4 \\ (R)^3 & (R)^3 & (R)^3 & (R)^4 & (R)^4 & (R)^4 \\ (R)^4 & (R)^4 & (R)^4 & (R)^5 & (R)^5 & (R)^5 \\ (R)^4 & (R)^4 & (R)^4 & (R)^5 & (R)^5 & (R)^5 \\ (R)^4 & (R)^4 & (R)^4 & (R)^5 & (R)^5 & (R)^5 \end{bmatrix}, \text{ the ratio } R = \left(\frac{L_t}{L_m} \right) = \left(\frac{1}{0.3} \right)$$

5.3 EXPERIMENT PROCEDURES

5.3.1 Experiment Setup

Figure 5.5 Experiment Setup for Pendulum Test

Experimental Set-up

- The scaled model of the ROV (shown in the Figure 5.5)
- Fixture to hang the scaled model
- A transparent water tank, size 1 x 2 x 1 meter
- A normal digital video-camera (3Mega-Pixels resolution)
- A force sensor (to measure weight)

Experimental Procedure

- Place a small black mark on the rod
- Fix the fixture to the structure of the water tank with a G-camp

- Setup the camera to capture the trajectory of the black mark when pendulum is swinging.
- The black mark must distinguish itself from its background.
- Displace the pendulum from its equilibrium, up to about 45 degree and release it to swing freely in the water.
- Record the trajectory of the black mark using the video camera.
- Download the video into PC and then cut it into frames.
- Obtained the x and y coordinates of the black mark for each frames using MATLAB Image Processing Toolbox
- Find the time history of the angle θ from the x and y coordinates
- Apply the least square algorithm to calculate the added mass, linear damping and quadratic damping (m_a, K_L, K_Q).

5.4 EXPERIMENT RESULT & DISCUSSION

5.4.1 Experiment in Air

The objectives to conduct the experiment in air are:

- To verify the feasibility of the method by comparing the pendulum length calculated from experiment data and pendulum length measured by ruler.
- To determine the friction at the pivoting point by calculating the linear and quadratic damping in air

In air, buoyancy $B = 0$.

Weight of the scaled model, $Mg = 32.4\text{N}$

Mass of the scaled model, $M = \frac{32.4}{9.81} = 3.3028\text{Kg}$

Figure 5.6 Series of Frames Cut from the Video during swinging in Air

A video camera is used to record the pendulum motion. The recorded video is “spliced” into multiple frames at a rate of 20 frames per second. Samples of some frames are shown in Figure 5.6 to illustrate the pendulum motion. From each image frame, the x and y coordinates of the black marker are identified. The centre of rotation (pivoting point) is found by fitting the arc over the x and y coordinates as shown in Figure 5.7.

Figure 5.7 X-Y Coordinates of the Marker and their Centre of Rotation

After obtaining the centre, the angle is found from following formula:

$$\theta = \tan^{-1} \left(\frac{x_i - x_{center}}{y_i - y_{center}} \right) \tag{5-12}$$

Figure 5.8 Time History of Angle Theta

To obtain the angular velocity and acceleration, a polynomial fit of order n is applied to the sample data. Then a polynomial function of order n given below is obtained:

$$\theta = at^n + bt^{n-1} + \dots + pt^2 + qt + r \quad (5-13)$$

$$\text{So, } \frac{d\theta}{dt} = ant^{n-1} + b(n-1)t^{n-2} + \dots + 2pt + q \quad (5-14)$$

$$\text{and } \frac{d^2\theta}{dt^2} = an(n-1)t^{n-2} + b(n-1)(n-2)t^{n-3} + \dots + 2p \quad (5-15)$$

Figure 5.9 Angular Velocity and Angular Acceleration derived from Polynomial Fitting

The angle θ measured from experiment is plotted in blue color and marked as '*'. A polynomial of order 11 best fit the angle data. From equation (5-14) and (5-15), it is possible to find the angular velocity and angular acceleration. The angular velocity is drawn in red and angular acceleration is drawn in black.

From the Figure 5.9, it can be seen that both angular velocity and acceleration are distorted at the beginning and at the end of the plot (at time $t = 0$ & time $t = 5$). When a time derivative is performed, data beyond this interval are needed, and this causes the errors indicated. The problem can be solved by taking data only from interval 0.5 to 4.5

second as shown in Figure 5.10. Those data before 0.5 second and after 4.5 second are excluded.

Figure 5.10 Data between the Intervals of 0.5 to 4.5 sec

Using the least square estimation,

$$\hat{\mathbf{g}}_{LS} = \begin{bmatrix} \alpha \\ \beta \\ \gamma \end{bmatrix} = \begin{bmatrix} -25.1280 \\ 0.0260 \\ -0.0481 \end{bmatrix}$$

Since the test is conducted in air, both buoyancy and added mass are zero.

$$\alpha = \frac{B - Mg}{(M + m_a)r} = -25.1280 \Rightarrow r = \frac{-9.81}{-25.128} = 0.3904m$$

$$\beta = \frac{-K_L}{(M + m_a)} = 0.0260 \Rightarrow K_L = \frac{-0.0260}{3.3028} = -0.0079N/ms^{-1}$$

$$\gamma = \frac{-K_Q r}{(M + m_a)} = -0.0481 \Rightarrow K_Q = \frac{0.0481 \times 0.3904}{3.3028} = 0.0057N/(ms^{-1})^2$$

The radius of pendulum is found to be 0.39 m. This matches the length from the point of pivot to the centre of the mass of the model as measured using a ruler. Friction forces at the pivoting point are very close to zero. Thus, the friction force at the pivoting point could be neglected and will not be considered in the subsequent tests.

The following second-order differential equation given in (5.3),

$$\ddot{\theta} = \alpha \cdot \sin \theta - \beta \cdot \dot{\theta} - \gamma \cdot |\dot{\theta}| \dot{\theta}$$

can be solved using α, β, γ and two initial conditions. The initial conditions are the angle $\theta(t=0)$ and angular velocity $\dot{\theta}(t=0)$ which at time equal zero. These are found from the experiment data. The unknowns α, β, γ have been found by least square method. A simulated model is created in Simulink base on the equation and the unknowns. Thus, the simulated data could be compared to experimental data as shown in Figure 5.11.

Figure 5.11 Experimental versus Simulated Angle Theta

Figure 5.11 shows the experimental data versus simulated data generated by Simulink model. The pendulum performs a simple harmonic motion, which could be described by a sinusoidal function. The amplitude decay is small proving that the friction at pivoting

point could be neglected. The simulation model is able to capture the motion of the pendulum with high accuracy. Thus, lead us to believe the feasibility and the reliability of the method used.

5.4.2 Experiment in Water (Surge, Sway and Heave)

Similar test are done for the surge, sway and heave directions using water. In water, there is buoyancy force $B = 10.5N$. Then the net force apply to the vehicle is $B - Mg = -21.9N$. Plots of the results are given below.

Figure 5.12 Image Sequence of the scaled Model when swinging in the Surge Direction.

Figure 5.13 Experiment Data versus Simulated Data for Angle θ in Surge Direction

Figure 5.14 Experiment Data versus Simulated Data for Angle θ in Sway Direction

Figure 5.15 Experiment Data versus Simulated Data for Angle θ in Heave Direction

Three tests had been conducted for each of the direction namely surge, sway and heave. The results are tabulated as shown:

Table 5.1 The Added Mass Coefficients and Drag Coefficients in Surge, Sway and Heave Direction

SURGE Maximum Speed¹ 0.55m/s

test	Added-Mass (Kg)	Linear Damping (N/(ms ⁻¹))	Quad Damping (N/(ms ⁻¹) ²)	RMS Error (radian)
1	0.5581	1.2736	9.9392	0.0568
2	0.6054	1.7768	9.1907	0.0470
3	0.5765	1.6040	9.5018	0.0552
Average	0.5800	1.5515	9.5439	0.0530

SWAY Maximum Speed 0.51m/s

Test	Added-Mass (Kg)	Linear Damping (N/(ms ⁻¹))	Quad Damping (N/(ms ⁻¹) ²)	RMS Error (radian)
1	1.4347	3.1388	7.8691	0.0277
2	1.5465	3.6694	7.4974	0.0234
3	1.4878	3.4671	7.3418	0.0369
Average	1.4897	3.4251	7.5694	0.0293

HEAVE Maximum Speed 0.35m/s

Test	Added-Mass (Kg)	Linear Damping (N/(ms ⁻¹))	Quad Damping (N/(ms ⁻¹) ²)	RMS Error (radian)
1	3.0759	6.561	8.9937	0.0238
2	3.1407	6.8327	8.1637	0.0219
3	2.9909	6.1905	11.035	0.0172
Average	3.0692	6.5281	9.3975	0.0210

¹ Maximum Speed = radius x maximum angular velocity

*The error is defined as the difference between the experiment result and the simulated result. (RMS = Root Mean Square)

Figure 5.16 Drag Force versus Velocity for Surge, Sway and Heave

5.4.3 Experiment in Water (Yaw)

A torsional spring was used to provide for the rotational oscillation of the scaled model in the yaw direction. This kind of setup is called a torsional pendulum because the object swings in a clockwise and anticlockwise direction alternatively.

T_s is the torque applied by the torsional spring and is proportional to the angle of rotation θ . So, $T_s = K\theta$ and K is the torque coefficient.

Figure 5.17 Free Body Diagram of Torsional Pendulum

T_H is the hydrodynamic torques that resist the motion of the body and consists of the added moment of inertia, linear and quadratic damping.

$$T_H = I_a \ddot{\theta} + K_L \dot{\theta} + K_Q |\dot{\theta}| \dot{\theta} \quad (5-16)$$

From Newton's Second Law:

$$\sum M = I \ddot{\theta}$$

$$F_s - F_H = I \ddot{\theta}$$

$$K\theta - (I_a \ddot{\theta} + K_L \dot{\theta} + K_Q |\dot{\theta}| \dot{\theta}) = I \ddot{\theta}$$

Rearrange the equation,

$$\ddot{\theta} = \frac{K}{(I+I_a)}\theta - \frac{K_L}{(I+I_a)}\dot{\theta} - \frac{K_Q}{(I+I_a)}|\dot{\theta}|\dot{\theta}$$

Let

$$\alpha = \frac{K}{(I+I_a)}, \beta = \frac{-K_L}{(I+I_a)}, \gamma = \frac{-K_Q}{(I+I_a)}$$

Then the equation become,

$$\ddot{\theta} = \alpha\theta + \beta\dot{\theta} + \gamma|\dot{\theta}|\dot{\theta} \quad (5-17)$$

The added moment of inertia, linear damping and quadratic damping coefficient can be found by Least Square method as described in the previous case.

The torque coefficient K is to be determined first. The period of oscillation is related to moment of inertia I and torque coefficient K :

$$T = 2\pi\sqrt{\frac{I}{K}} \quad (5-18)$$

A rectangular iron block with length 15.55cm and width 5.8cm is used as a standard object with known moment of inertia. The moment of inertia of the block is:

$$\begin{aligned} I &= \frac{1}{12}(m)(a^2 + b^2) \\ &= \frac{1}{12}(0.305)(0.1555^2 + 0.058^2) \\ &= 7 \times 10^{-4} \text{ kgm}^2 \end{aligned}$$

The total period of five oscillations is record as below:

Table 5.2 Oscillation Period of the Rectangular Iron Block in Air

Test	Total Time Taken for 5 oscillations (sec)	Period (sec)
1	8.5	1.70
2	8.6	1.72
3	8.5	1.70
4	8.7	1.74
5	8.6	1.72

The average of the period (T) = 1.72 s

Thus, the torque coefficient K from equation (5-18) is:

$$1.72 = 2\pi \sqrt{\frac{7 \times 10^{-4}}{K}}$$

$$\Rightarrow K = 0.00934 \text{ Nm/rad}$$

The experiment is repeated for the scaled model. The period is record as below:

Table 5.3 Oscillation Period of the Scaled Model in Air

Test	Total Time Taken for 5 oscillations (sec)	Period (sec)
1	53.3	10.66
2	51.5	10.30
3	52.3	10.46
4	51.7	10.34
5	52.5	10.50

The average period (T) is 10.45 second.

The moment of inertia of the body (URV) is:

$$10.45 = 2\pi \sqrt{\frac{I}{0.00934}}$$

$$I = 0.00251 \text{ kg m}^2$$

The result obtained matches closely with the moment of inertia calculated using CAD software, SolidWork. The CAD result is 0.0026 kgm^2 .

Figure 5.18 Image Sequence of the Scaled Model when oscillated in the Yaw Direction

The scaled model is rotated about 5.8 radian (330 degrees) from its equilibrium point and release to swing freely as shown in Figure 5.18.

Figure 5.19 Experiment Data versus Simulated Data for Angle θ in Yaw Direction

The simulated data match well with the experimental data in the yaw direction as shown in Figure 5.19. It can be observed that the oscillatory motion damped out quickly from 5.8 radians to 1 radian after one oscillation. Three tests were conducted and their results are tabulated in Table 5.4.

Table 5.4 The Added Mass Coefficients and Drag Coefficients in Yaw Motion

YAW Maximum Speed 1.3 rad/s $\approx$ 74deg/s

Test	Added-Moment of Inertia (Kgm ²)	Linear Damping (Nm/(rads ⁻¹))	Quad Damping (Nm/(rads ⁻¹) ²)	RMS Error (radian)
1	0.0065	0.00073	0.022	0.032
2	0.0090	0.0031	0.018	0.031
3	0.011	0.0054	0.015	0.048
Average	0.0088	0.0031	0.019	0.037

The magnitude of the damping coefficient for yaw motion is found to be smaller than those coefficients for translation motions (surge, sway and heave). This is because the magnitude of inertia for yaw motion is smaller than the magnitude of the inertia for translation motions.

The moment of inertia of the scaled model in yaw motion is 0.00251kgm^2 whereas the inertia mass in translation motion is 115kg .

5.4.4 Comparison: (Experiment Vs CFD)

- Added Mass

The added mass of the scaled model was calculated using WAMIT.

Figure 5.20 CAD model used in WAMIT and the Real Model used in Experiment.

Table 5.5 Comparison of Added Mass from experiment and WAMIT

	Surge	Sway	Heave	Yaw
Experiment	0.5800	1.4897	3.0692	0.0088
WAMIT	0.5317	1.4014	2.8197	0.0097

The calculated added masses from WAMIT match closely with the experimental values in surge, sway, heave and the yaw direction. From this comparison, it has been shown that the values calculated from WAMIT are reasonably reliable and accurate. So, the added mass matrix for ROV II calculated in chapter 3 will be used.

• Drag

Table 5.6 Comparison of Linear and Quadratic Drag Coefficient from Experiment and ANSYS CFX

Direction	Surge		Sway		Heave		Yaw	
Range	(0-0.5m/s)		(0-0.5m/s)		(0-0.2m/s)		(0-0.52rad/s)	
Parameter	K_L	K_Q	K_L	K_Q	K_L	K_Q	K_L	K_Q
Experiment	17.24	106.03	38.06	84.10	72.53	104.41	1.18	7.51
ANSYS CFX	11.863	108.45	19.64	108.24	2.3756	351.98	0	10.39

Comparison should not be done on the basic of K_L versus K_L or K_Q versus K_Q . Indeed, only the combination effect of both should be compare by plotting the total drag force within the velocity range of operation as following:

Figure 5.21 and Figure 5.22 show that the drag forces for surge and sway match closely with values from CFX although they are consistently larger. The main reason is that smooth surface is assumed in ANSYS CFX computation and this probably underestimates the drag value due to skin friction.

Figure 5.21 Drag in Surge Direction.

Figure 5.22 Drag in Sway Direction.

For the heave direction, the experimental values are consistently larger than the CFX values by a constant for velocity range from 0.1 m/s to 0.2m/s. This is expected because there are two rectangular metal plate attached to the scaled model where increases the drag in the heave direction (as one can see from Figure 5.20). In ANSYS CFX study, the two iron plats are not existed in full scale ROV II.

Figure 5.23 Drag in Heave Direction.

The results for the yaw direction from both experiment and ANSYS CFX are closely matched as shown in Figure 5.24. This is an unexpected result because calculating the drag moment is considered difficult by the CFD community. The experiment data of ROV II is obtained by scaling up the drag moment from a scaled model tests that involve a very big scaling factor of 411.52 (see page 5.70). Therefore, any error in scaled model tests will also be scaled up as well. This is probably a limitation in the use of the free decay test for a scale model. Nevertheless, the magnitude of the moment drag is reasonable and could be a good estimation of the true moment drag that will be experienced by the underwater vehicle.

Figure 5.24 Drag in Yaw Motion.

CHAPTER 6. VERIFICATION OF THE HYDRODYNAMICS PARAMETERS

6.1 INTRODUCTION

In the previous chapters, the hydrodynamic parameters of the ROVII including the added mass and drag coefficients have been determined from free decay pendulum tests and Computational Fluid Dynamics (CFD) simulations. The accuracy of the CFD simulation results are dependent on the model representation and the numerical methods used. As a scaled down model is used in the free decay pendulum test, it is uncertain that the coefficients can be scaled up accurately to the true size of the ROVII. In order to find answer to these questions, experimental verification using the actual ROVII have to be performed.

An experimental method which identifies the hydrodynamic parameters of an underwater vehicle by exciting the vehicle using its own thrusters and measuring the response using on-board sensors in real environment is called an *on-board sensor-based identification method*. More recently, the use of on board sensor-based identification has become very popular within a research community [2, 27, 28]. This technique is attractive to use because it makes use of on board sensors and thrusters in the process of identification. No other equipment is needed. It is cost effective, easy to execute and the repeatability is high. It is suitable for variable configuration ROVs where the payload and the profile of ROVs change according to different missions [6]. An on-line deterministic adaptive identification method was first proposed by [28], where JHUROV (John Hopkins University) was the subject of study. Ridao P. et al [27] had determined the hydrodynamic coefficients only for uncoupled one dimensional model in the principal DOFs (surge, heave and yaw) of the URIS underwater vehicle.

However, identification of full model of the ROVs with this method is not without its difficulties. Much of the reported work simplifies the model to an uncoupled model in a single DOF. To conduct an uncoupled experiment, the motion of the ROV must be constrained in a single DOF, which is hard to implement in practice. In addition, it is also hard to model the thruster forces and to measure the vehicle's response accurately. Most

of the time, the vehicle response is affected by different sources of disturbance such as the tether force and current. These issues will be discussed in detail in following section.

6.2 ERROR SOURCES

The verification experiment was carried out with consideration of the following error sources. Better result is obtained if the experiment can be done in a more controlled environment. For example, the tether can be placed such that the force exert on the ROV is minimal. All collected data must be examined to identify usable data and discard outliers.

6.2.1 Unwanted Excitation in other degree of freedoms

A multi-input multi-output (MIMO) PID controller had been designed to move the vehicle in a particular direction without exciting the others. This is important because a decoupled model has been used to describe the vehicle motion. However, due to the geometrical shape and the placement of the thrusters, the motion of the ROV is not so easily decoupled. The best that can be done is to minimize those effects which will contribute errors to the measurements.

6.2.2 Thrusters Modelling

Modelling of the thrust force under different voltage inputs to the thruster is a practical challenging problem. Until now, there is no appropriate model reported by the research community that can reproduce the thrust and propeller velocity transient response in actual thruster step response experiments [46]. Moreover, the thruster experiences a significant efficiency loss due to propeller-propeller and propeller hull interaction when it is mounted on the ROV [6]. In this research project, the simplest representation of the thruster model taking into account the efficiency loss is adopted.

6.2.3 Sensors Limitation

Sensors play an important role in measuring the vehicle responses. For example, surge motion is measured by an altimeter, heave motion is measured by a depth sensor and yaw motion is measured by a compass. However, these sensors are not perfect. They exhibit nonlinearity and the signals are corrupted by random noise and are affected by long

sampling interval, for example. Filters can be designed to minimise the effect of Gaussian noise, but not for the random bias (random walk).

6.2.4 External Disturbance

In addition to the above mention possible sources of errors, the vehicle is also subjected to vast variety of disturbances. These include the pull of the tether force and the flow of current. The effect of current flow may be small in a pool condition but the force exert by the tether is significant. According to [47], an estimate of the cable tension is about 10N. With the maximum force generated by thruster of about 50N, the tether force may contribute up to 20% of the total forces and moments applied to the ROV. The forces and moments exerted by the tether are difficult to predict as the cable may form different curvatures during the ROV motion (see Figure 6.1). Interested reader on tether effect may refer to the [47] and references herein. Authors of [47] propose to find the tether effect by solving two-end boundary-value problem which assume the tether cable form a nice curvature in 2D-plane. This is obviously not valid in practice as the curvature of the tether is 3 dimensional and arbitrary in nature. Considering the complexity involved in modelling the tether and the accuracy obtained, it is not worthwhile to include it in the current dynamics model. In this project, the tether effect is treated as the external disturbance which will be handled by the controller. In practice, the tether effect is minimised by employing a Tether Management System (TMS) which will keep the cable tension small.

Figure 6.1 ROV with Umbilical Cable (Tether)

6.3 VERIFICATION METHODOLOGY

The purpose of this experiment is not to repeat the process of identification of the hydrodynamic parameters of the ROVII which had been done in previous chapter. The main purpose is to verify the validity of the coefficients that are found through free decay pendulum test and CFD. The idea is illustrated by Figure 6.2.

Figure 6.2 Verification Flowchart

6.3.1 Forces and Moments (Detail in Appendix F)

The thrust generated by each thruster mounted on the ROVII had been identified and will be discuss in detail in appendix F. Their relationships with the input voltages are as follows:

$$\begin{array}{cc}
 T_1 \text{ and } T_2 & T_3 \text{ and } T_4 \\
 T_{1,2} = \begin{cases} 0.0326V^2, V > 0 \\ -0.0153V^2, V < 0 \end{cases} & T_{3,4} = \begin{cases} 0.0175V^2, V > 0 \\ -0.0122V^2, V < 0 \end{cases}
 \end{array}$$

$$\tau = \begin{bmatrix} X \\ Y \\ Z \\ K \\ M \\ N \end{bmatrix} = \begin{bmatrix} T_1 + T_2 \\ (T_3 - T_4) \sin \beta \\ (T_3 + T_4) \cos \beta \\ -\delta(T_3 - T_4) \\ -\delta(T_1 + T_2) - \alpha(T_3 + T_4) \cos \beta \\ \gamma(T_1 - T_2) + \alpha(T_3 - T_4) \sin \beta \end{bmatrix}$$

6.3.2 Dynamics Model

The dynamics model of the ROV as described in the literature [4] is presented in chapter 2. The equations of motion for three principal motions (surge, heave and yaw) are given below:

Surge $(m + M_u)\dot{u} = -k_u w - k_{u|u}|u| + X$ (6-1)

Heave $(m + M_w)\dot{w} = -k_w w - k_{w|w}|w| + B + Z$ (6-2)

Yaw $(m + M_r)\dot{r} = -k_r r - k_{r|r}|r| + N$ (6-3)

Nomenclature

m	Mass of the ROVII
$(M_u, M_v, M_w, M_p, M_q, M_r)$	Added Mass
u, v, w, p, q, r	Linear and angular velocities
B	Buoyancy
(X, Y, Z, K, M, N)	External Applied Force and moment
$(k_u, k_v, k_w, k_p, k_q, k_r)$	Linear Damping Coefficient
$(k_{u u }, k_{v v }, k_{w w }, k_{p p }, k_{q q }, k_{r r })$	Quadratic Damping Coefficient

Simplification

- For simplification and for practical considerations the body-fixed reference frame is chosen to be at the centre of gravity, i.e. $x_G=y_G=z_G=0$. This means that the rigid body inertial matrix, $\mathbf{M}_R$ term has a block diagonal structure. The first 3×3 block consists of the ROV mass and the second 3×3 block consists of the inertia terms. In this second block, the off-diagonal elements of inertia term I_{xy} , I_{xz} and I_{yz} in $\mathbf{M}_R$ are much smaller than the diagonal counterparts. Instead of using all the thirty-six

added mass parameters, the diagonal approximation is found to be quite good for many applications especially when the ROV is moving at low speed [4, 5].

- In slow and decoupled motion, the added mass matrix $\mathbf{M}_A$ can be assumed to be diagonal with the contributions of the off diagonal terms relatively smaller.
- The linear and quadratic damping matrix $\mathbf{D}(\mathbf{v})\mathbf{v}$ is assumed to have only diagonal terms with all off-diagonal term equal relatively smaller, especially if the system has planes of symmetry where the terms higher than second-order are negligible. The diagonal approximation is found to be quite good for many applications[6].
- Moreover, if the vehicle is actuated in a single DOF during the identification experiment, all the cross coupling terms will become zero.

6.3.3 Simulated Response

Simulink, a tool for modelling, simulation and analyzing dynamic system, has been used to simulate the responses of the model under the given forces and moments inputs. The dynamic model of ROV II is constructed in Simulink and is shown in fig 6.3:

Figure 6.3 Simulink Block Diagram for Decoupled Model

There are five parameters in the model. They are vehicle mass, added mass, linear drag, quadratic drag and buoyancy force. These parameters have been identified in the previous chapters and are restated here for ease of reference:

Table 6.1 Hydrodynamics Coefficients for Surge, Heave and Yaw of ROV II

Axis	Mass/ Moment of Inertia	Added Mass /Added Moment of inertia	Linear Drag	Quadratic Drag	Buoyancy
Surge	115kg	21.1kg	17.2N/ms ⁻¹	106.02N/(ms ⁻¹) ²	0
Heave	115kg	51.7kg	72.5N/ms ⁻¹	104.4 N/(ms ⁻¹) ²	12N
Yaw	9.6kgm ²	2.3 kgm ²	1.2Nm/rads ⁻¹	7.5Nm/(rads ⁻¹) ²	0

Runge-Kutta integration is used to solve the continuous time dynamic models with fixed step-size according to sensor’s sampling interval.

6.3.4 Real Plant

The real plant refers to the ROVII in the poll environment. In the experiment, the vehicle responses are measured by the on-board sensors.

6.3.5 Measured Response

Table 6.2 tabulates the sensors used to measure response for respective axis or degree of freedom.

Table 6.2 Sensor Information to measure response in Surge, Heave and Yaw

Axis	Sensor	Model	Measurement	Accuracy	Sampling Time
Surge	Altimeter	PA500	Distance	± 1mm	0.10s
Heave	Depth Meter	2” Micro CTD	Depth	± 0.2m	0.01s
Yaw	Compass	TCM2	Heading	± 0.1 °	0.10s

6.3.6 Verification

The last step is to verify the validity of the estimated parameters by plotting the simulated response and measured response on the same graph. If the simulated responses are quite close to the measured response, in magnitude and trend, then one may conclude that the estimated parameters are sufficiently accurate.

6.4 EXPERIMENT RESULT & DISCUSSION

The three major motions of ROV II are the surge, heave and yaw. With these, the ROVII can navigate to any point in the 3D space. The ROVII is not designed to move in the sway direction as the hydrodynamic resistance in this particular mode is high. There are minimal motions in roll and pitch as it is self-stabilized by the buoyancy force. This is a desirable feature to keep the ROV II stable during depth positioning and manoeuvring. So, only verification on surge, heave and yaw motion are needed.

For each degree of freedom (DOF), the relevant thrusters will be used to excite the ROVII in that particular DOF while minimizing motions in other DOF. For example, when conducting experiment for surge, the ROVII needs to maintain a certain depth and the only motion is forward. Input voltages to the thruster and the vehicle response will be recorded.

It is important to state that experiment data is not used to train the simulated response to match the experiment measured response, i.e. there are no data learning or fitting. The simulated response is the prediction from the model obtained by free decay pendulum tests and CFD.

6.4.1 Surge

Figure 6.4 Thrust for Surge Verification

Figure 6.5 Experimental versus Simulated Position in a positive Surge Movement

Figure 6.6 Experimental versus Simulated Speed in a positive Surge Movement

Figure 6.4 shows the force that excites the ROV II in the surge direction. The ROV II was commanded to move forward with speed 0.3 m/s. At time equals to zero, the vehicle is static, so a very big force input is required to overcome the vehicle inertia. After around 4 seconds, when the vehicle is moving at 0.3 m/s, force of 20N is required to maintain this speed. However, one could see that the force is not stable but fluctuate. It is evident that the ROV II is subjected to continuous disturbance from the tether. So, the controller tries to adjust the thrust output to maintain the speed.

Figure 6.5 and 6.6 show the comparison of the simulated and experiment measured position and speed respectively. From Figure 6.5, it can be seen that the simulated result matches the experiment result quite well when the ROV II travel up to 5m forward. The maximum error is less than 0.5m or 10% of the distance travel. The discrepancy could be found by looking at the velocity plot. There is a sudden drop in ROVII velocity at around 1 second which is unusual because the thrust input is positive and large. The vehicle should accelerate at that moment. It is obvious that there is impulsive disturbance acting on the ROVII, probably due to the tether force. Despite the sudden impulse, the model is capable in predicting the nominal speed.

6.4.2 Heave

Figure 6.7 Thrust for Heave Verification

Figure 6.8 Experimental versus Simulated Depth in a Heave Movement

Figure 6.9 Experimental versus Simulated Speed in a Heave Movement

At time equals to zero, the vehicle is fully submerged to a depth of 1.3m. It is important that the vehicle is fully submerged before the start of the experiment to avoid surface effect. The vehicle is made to hover at 1.3m depth for 40 seconds before moving deeper to 2.6m. During the hovering period, a thrust of 12N is needed. Thus the buoyancy force is identified to be 12N. The simulated responses follow closely the measure responses both in magnitude and trend. Hence, the model works remarkable well in the heave estimation.

6.4.3 Yaw

Figure 6.10 Torque for Yaw Verification

Figure 6.11 Experimental versus Simulated Depth in a Yaw Movement

Figure 6.12 Experimental versus Simulated Depth in a Yaw Movement

The vehicle was commanded to rotate 4.8 radians or 275 degrees within 5 seconds. There was large moment input to the system at the beginning to overcome the moment of inertia as shown in Figure 6.10. After that, the ROV II maintained heading for 10 seconds. The simulated response matches quite well with the experimentally measured responses for both heading and angular velocity. Compared to the surge and heave, the simulation and the experiment measurement of yaw are better matched each with other. This may be due to the disturbance from the tether being relatively smaller in this particular experiment. Once again, the coefficients estimated by pendulum test and CFD have been shown sufficiently accurate.

6.5 CONCLUSION

This chapter serves the important role to verify the validity of the hydrodynamic coefficients found using both the free decay pendulum tests and the CFD method. A verification methodology has been proposed for this purpose by experimenting with the ROV II in the diving pool. The thrust forces generated by the four thrusters are identified through measurement of thrust under different voltage inputs. Finally, by comparing simulated responses and experiment measured responses, hydrodynamic parameters for surge, heave and yaw DOF have been verified.

CHAPTER 7. CONCLUSION AND FUTURE WORK

7.1 CONCLUSION

The dynamics model is essential for a successful implementation and operation of guidance, navigation and control (GNC) system of a semi-autonomous underwater vehicle. The main difficulty in constructing a suitable model with the appropriate parameters is due to the uncertainty of the hydrodynamics forces experienced by vehicle while maneuvering underwater. In general, the hydrodynamics forces can be classified into two main categories: added mass and drag. The objective of this report is to find reasonable accurate added mass and drag coefficients for NTU RRC ROV II. Two approaches experiment method and computational fluid dynamics (CFD) method had been adopted and the results collaborated well with each other.

The added mass of the ROV II had been calculated using WAMIT which is a CFD software based on the panel method to solve potential flow around the vehicle. The calculated added mass is in good agreement with the experiment data on the scaled model. Thus, we are confident that the calculated added mass of ROV II is sufficient accurately enough for the purpose of GNC application.

The finding of the drag force is trickier than the added mass one. The use of ANSYS CFX with Shear Stress Transport (SST) turbulence model, allows one to estimate the onset of flow separation and thus predict pressure drag with sufficient accuracy. The main difficulty is the prediction of linear skin friction drag. As the surface roughness of ROV II is hard to quantify, a smooth surface is assumed in CFD simulation. Therefore, the resulting drag prediction is smaller than the experiment one by a constant value. Fortunately, the pressure drag is the dominant drag component during the ROV operation. Hence, the impact on the accuracy may be small.

A new free decay test has been proposed to identify the added mass and drag coefficient of a scaled down model of the ROV II. The test makes use of the pendulum swing motion to identify coefficients in surge, sway and heave and a torsional pendulum motion to identify coefficients in yaw. Although the experiment setup is very simple and low cost, the result obtained is reasonably reliable. The results can be scaled up to predict the coefficients for ROV II. These have been verified by comparing the simulated results of

model of the real ROV II using the scaled up parameters for the real ROV II and experimentally measured results of the ROV II in a pool test. The match is particularly good in the yaw directions where the effect of the tether disturbance is minimal followed by that in the heave directions and then the surge direction. The last direction is most affected by the tether. Hence for a class of medium and small class of underwater vehicles, it is possible to determine the hydrodynamic parameters without extensive and expensive facility and instrumentation.

The use of both experiment and CFD method in parallel has given remarkable insight in finding and understanding the effect of added mass and drag coefficients in underwater motion. By comparing the results obtained from both methods, some important conclusions can be made. From the CFD viewpoint, the experimental results provide a strong validation for the simulation results. On the other hand, CFD enables us to understand the flow phenomenon such as the drag coefficients of ROV II which remain almost constant within the operating velocity of 0-0.5m/s. CFD results also helped in demonstrating the Scaling Law. So, the combined approach is recommended for finding the added mass and drag coefficients of an underwater vehicle.

7.2 FUTURE WORK

The future work can be explored in the following areas:

7.2.1 Navigation

The dynamical model could be used to aid the navigation system design of ROV II. As discussed early in the report, the navigation system suffers from low sampling frequency and data that corrupted by noises. In some cases, the sensors data may not be available all the time. By knowing the dynamics model and the control inputs to the plant, one could predict the motion of the vehicle. Therefore, in future studies, we will try to incorporate the dynamics model in Kalman-filter base estimator to enhance the states estimation of the navigation system.

7.2.2 Control

A cascade PID controller can be designed using the dynamics model of the ROV II. The PID parameters could be fine-tuned in computer simulation (Simulink) to meet specific

control requirements. Then the control capability will be tested by conducting real experiment in diving pool.

References

- [1] G. Conte and A. Serrani, "Modelling and simulation of underwater vehicles," presented at Proceedings of the IEEE International Symposium on Computer-Aided Control System Design, Dearborn, MI, USA, 1996.
- [2] A. Alessandri, R. Bono, M. Caccia, G. Indiveri, and G. Veruggio, "Experiences on the modelling and identification of the heave motion of an open-frame UUV," presented at Oceans Conference Record (IEEE), Nice, Fr, 1998.
- [3] G. Bekey, R. Ambrose, V. Kumar, A. Sanderson, B. Wilcox, and Y. Zheng, "International Assessment of Research and Development in Robotics," World Technology Evaluation Center, Inc. 2006.
- [4] T. I. Fossen, *Guidance and Control of Ocean Marine Vehicles*: John Wiley and Sons Ltd. New York, 1994.
- [5] T. I. Fossen, *Marine Control Systems: Guidance, Navigation and Control of Ships, Rigs and Underwater Vehicles*: Marine Cybernetics, Trondheim, 2002.
- [6] M. Caccia, G. Indiveri, and G. Veruggio, "Modeling and identification of open-frame variable configuration unmanned underwater vehicles," *IEEE Journal of Oceanic Engineering*, vol. 25, pp. 227-240, 2000.
- [7] A. T. Morrison, III and D. R. Yoerger, "Determination of the hydrodynamic parameters of an underwater vehicle during small scale, nonuniform, 1-dimensional translation," Victoria, BC, Canada, 1993.
- [8] M. Caccia, G. Casalino, R. Cristi, and G. Veruggio, "Acoustic motion estimation and control for an unmanned underwater vehicle in a structured environment," *Control Engineering Practice*, vol. 6, pp. 661-670, 1998.
- [9] P. J. Smith, J. S. Smith, and J. Lucas, "Localization of a UUV within structures using range data and world modelling techniques," *International Journal of Systems Science*, vol. 30, pp. 915-928, 1999.
- [10] D. M. Steinke and B. J. Buckham, "A Kalman filter for the navigation of remotely operated vehicles," presented at Proceedings of MTS/IEEE OCEANS, 2005, Washington, DC, 2005.
- [11] T. H. Koh, M. W. S. Lau, E. Low, G. Seet, S. Swee, and P. L. Cheng, "Development and improvement of an underactuated underwater robotic vehicle," presented at Oceans Conference Record (IEEE), 2002.
- [12] J. Yuh, "Modeling and control of underwater robotic vehicles," *IEEE Transactions on Systems, Man and Cybernetics*, vol. 20, pp. 1475-83, 1990.
- [13] K. R. Goheen and E. R. Jefferys, "System identification of remotely operated vehicle dynamics," *Journal of Offshore Mechanics and Arctic Engineering*, vol. 112, pp. 230-236, 1990.
- [14] G. Conte, S. M. Zanolli, D. Scaradozzi, and A. Conti, "Evaluation of hydrodynamics parameters of a UUV. A preliminary study," presented at International Symposium on Control, Communications and Signal Processing, ISCCSP, Hammamet, 2004.
- [15] S. F. Hoerner, *Fluid Dynamic Drag*: Published by author, 1965.
- [16] T. Presterio, "Verification of a Six-Degree of Freedom Simulation Model for the REMUS Autonomous Underwater Vehicle," MASSACHUSETTS INSTITUTE OF TECHNOLOGY & WOODS HOLE OCEANOGRAPHIC INSTITUTION 2001.

-
- [17] T. Prestero, "Verification of a Six-Degree of Freedom Simulation Model for the REMUS Autonomous Underwater Vehicle," vol. Master's thesis: Mechanical and Oceanographic Engineering, Massachusetts Institute of Technology and the Woods Hole Oceanographic Institution, 2001.
 - [18] T. Sarkar, P. G. Sayer, and S. M. Fraser, "A study of autonomous underwater vehicle hull forms using computational fluid dynamics," *International Journal for Numerical Methods in Fluids*, vol. 25, pp. 1301-1313, 1997.
 - [19] E. Y. K. Ng and C. K. Tan, "Viscous flow simulation around a moving projectile and URV," *International Journal of Computer Applications in Technology*, vol. 11, pp. 350-362, 1998.
 - [20] K. Kangsoo, T. Sutoh, T. Ura, and T. Obara, "Route keeping control of AUV under current by using dynamics model via CFD analysis," Honolulu, HI, USA, 2001.
 - [21] A. L. Fuglestad and M. Grahl-Madsen, "Computational fluid dynamics applied on an autonomous underwater vehicle," presented at Proceedings of the International Conference on Offshore Mechanics and Arctic Engineering - OMAE, Vancouver, BC, 2004.
 - [22] T. Kang, K. Hu, Z. Q. Hu, and Y. Lin, "Research on maneuverability simulation calculation of underwater vehicle by CFX and USAERO," *Jiqiren/Robot*, vol. 27, pp. 535-538, 2005.
 - [23] Z. Q. Hu, Y. Lin, and H. T. Gu, "On numerical computation of viscous hydrodynamics of unmanned underwater vehicle," *Jiqiren/Robot*, vol. 29, pp. 145-150, 2007.
 - [24] A. Heron, A. Duncan, and B. Anderson, "Hydrodynamic testing of underwater vehicles at the Australian Maritime Engineering Co-operative Research Centre," 2000.
 - [25] P. Egeskov, A. Bjerrum, A. Pascoal, C. Silvestre, C. Aage, and L. W. Smitt, "Design, construction and hydrodynamic testing of the AUV MARIUS," Cambridge, MA, USA, 1994.
 - [26] C. Aage and L. Wagner Smitt, "Hydrodynamic manoeuvrability data of a flatfish type AUV," Brest, France, 1994.
 - [27] P. Ridao, A. Tiano, A. El-Fakdi, M. Carreras, and A. Zirilli, "On the identification of non-linear models of unmanned underwater vehicles," *Control Engineering Practice*, vol. 12, pp. 1483-1499, 2004.
 - [28] D. A. Smallwood and L. L. Whitcomb, "Adaptive identification of dynamically positioned underwater robotic vehicles," *IEEE Transactions on Control Systems Technology*, vol. 11, pp. 505-515, 2003.
 - [29] T. I. F. A. Ross, and T. A. Johansen, "Identification of underwater vehicle hydrodynamic coefficients using free decay tests," *IFAC Conference on Control Applications in Marine Systems, Ancona, Italy*, 2004.
 - [30] J.N.Newman, *Marine Hydrodynamics*: MIT Press, 1977.
 - [31] J. L. Hess and A. M. O. Smith, "Calculation of nonlifting potential flow about arbitrary three-dimensional bodies," *Journal of Ship Research*, vol. 8, pp. 22-44, 1964.
 - [32] L. L. Erickson, "Panel methods An introduction," NASA Center, Ames Research Center Dec 1, 1990.
-

-
- [33] M. S. Triantafyllou and F. S. Hover, *Maneuvering and Control of Marine Vehicles*: Department of Ocean Engineering, MIT, Cambridge, Massachusetts USA, 2003.
 - [34] I. ANSYS, "ANSYS CFX-Solver, User Manual Release 10.0: Theory," 2005.
 - [35] I. ANSYS, "Innovative Turbulence Modeling: SST Model in ANSYS® CFX®," 2006.
 - [36] B. E. Launder and D. B. Spalding, "The numerical computation of turbulent flows," *Comp. Methods Appl. Mech. Eng.*, pp. 269-289, 1974.
 - [37] I. ANSYS, "ANSYS CFX-Mesh Tutorial, User Manual Release 10.0: Theory," 2005.
 - [38] W. A. Consultants and M. o. t. NSC, "Best Practice Guidelines for Marine Applications of Computational Fluid Dynamics."
 - [39] D. Kim and H. Choi, "Laminar flow past a sphere totating in the streamwise direction," *Journal of Fluid Mechanics*, pp. 365-386, 2002.
 - [40] T. A. Johnson and V. C. Patel, "Flow past a sphere up to a reynolds number of 300," *Journal of Fluid Mechanics*, pp. 19-70, 1999.
 - [41] E. Achenbach, "Experiments on the flow past spheres at very high reynolds number," *Journal of Fluid Mechanics*, pp. 565-575, 1972.
 - [42] G. S. Constantinescu, R. Pacheco, and K. D. Squires., "Detached-eddy simulation of flow over a sphere.," *ALAA paper, Aerospace Sciences Meeting*, pp. 425, 2002.
 - [43] S. J. Kline, *Similitude and Approximation Theory*: Springer-Verlag, New York, 1986.
 - [44] Munson, Young, and Okiishi, *Fundamentals of Fluid Mechanics, 5th Edition*: Wiley Higher Education.
 - [45] R.-C. C. T.-L. L. Chen-Chou Lin, "Experimental determination of the hydrodynamic coefficients of an underwater manipulator," *Journal of Robotic Systems*, vol. 16, pp. 329-338, 1999.
 - [46] L. L. Whitcomb and D. R. Yoerger, "Development, comparison, and preliminary experimental validation of nonlinear dynamic thruster models," *IEEE Journal of Oceanic Engineering*, vol. 24, pp. 481-494, 1999.
 - [47] M. C. Fang, C. S. Hou, and J. H. Luo, "On the motions of the underwater remotely operated vehicle with the umbilical cable effect," *Ocean Engineering*, vol. 34, pp. 1275-1289, 2007.
 - [48] P. D. K. P. Yue, "Marine Hydrodynamics 2.20 Lecture 14," 2005.
 - [49] P. Ridao, J. Battle, and M. Carreras, "Dynamics Model of an Underwater Robotic Vehicle," 2001.
 - [50] T. Chklovski, "Pointed-Tip Wings at Low Reynolds Numbers."
 - [51] Jeff Scott, "Golf Ball Dimples & Drag," 2005.
 - [52] Jeff Scott, "Drag of Cylinders & Cones," 2005.
-

APPENDIX A: PROPERTIES OF HYDRODYNAMICS ADDED MASS

1. Added mass coefficients represent the amount of fluid accelerated with the body. However, the added mass coefficients for translation generally differ depending on the direction of the body motion. In absence of body symmetry, the cross-coupling coefficients, m_{12} , m_{13} and etc are non-zero.
2. Added mass $m_{ij} = \rho$ (density of the surrounding fluid) x function (of geometry only)
3. M_A coefficients are constant and thus independent of the waves circular frequency.
4. For completely submerged vehicles M_A will always be strictly positive, that is $M_A > 0$
5. For a rigid body at rest ($U \approx 0$) under assumption:
 - Ideal fluid
 - No incident waves
 - No sea current
 - Frequency independent

So $\Rightarrow$ added inertia matrix is positive definite: $M_A = M_A^T > 0$

(Violate at the seabed, near underwater installation, at the surface.)

6. Symmetry of added mass matrix $m_{ij} = m_{ji}$ reduces from 6x6, 36 terms to 21 terms.

Figure A 1 Symmetry of the Added Mass Matrix [48]

From plane of symmetry, we know motion in '1' direction could not create force in '3' direction. However, it is not obvious motion in '3' direction will not create force in '1' direction.

7. For xz-plane of symmetry $M =$

$$\begin{bmatrix} m_{11} & 0 & m_{13} & 0 & m_{15} & 0 \\ & m_{22} & 0 & m_{24} & 0 & m_{26} \\ & & m_{33} & 0 & m_{35} & 0 \\ & & & m_{44} & 0 & m_{46} \\ & & & & m_{55} & 0 \\ & & & & & m_{66} \end{bmatrix}$$

∴ 12 elements to be determined

8. Many ROV applications the vehicle will only be allowed to move in low speed. If the vehicle also has nearly three planes of symmetry, this suggests that we can neglect the contribution from the off-diagonal terms in the M_A . It should be noted that the off diagonal terms of M_A will be small compare to the diagonal. So, diagonal approximation is found to be quite good for many practical applications. These greatly simplify the added mass matrix and its coriolis matrix to:

$$M_A = \begin{bmatrix} X_u & 0 & 0 & 0 & 0 & 0 \\ 0 & Y_v & 0 & 0 & 0 & 0 \\ 0 & 0 & Z_w & 0 & 0 & 0 \\ 0 & 0 & 0 & K_p & 0 & 0 \\ 0 & 0 & 0 & 0 & M_{\dot{q}} & 0 \\ 0 & 0 & 0 & 0 & 0 & N_r \end{bmatrix} \quad C_A(v) = \begin{bmatrix} 0 & 0 & 0 & 0 & -Z_w w & Y_v v \\ 0 & 0 & 0 & Z_w w & 0 & -X_u u \\ 0 & 0 & 0 & -Y_v v & -X_u u & 0 \\ 0 & -Z_w w & Y_v v & 0 & -N_r r & M_{\dot{q}} q \\ -Z_w w & 0 & -X_u u & N_r r & 0 & -K_p p \\ Y_v v & -X_u u & 0 & -M_{\dot{q}} q & K_p p & 0 \end{bmatrix}$$

APPENDIX B: POTENTIAL FLOW THEORY

Added Mass is the product of potential flow theory. Having said that, it means added mass is well defined by potential flow theory and can be calculated using potential flow theory.

When external flows around a body can be treated as inviscid (frictionless), incompressible and irrotational (fluid particles are not rotating), a potential function $\phi(x, y, z, t)$ can be defined to fulfill the basic laws of fluid mechanics (conservation of mass and momentum).

The potential function has direct relationship with velocity of the fluid particle in the flow:

Let velocity of the fluid particle in Cartesian coordinate is

$$V = [u \quad v \quad w]^T, \text{ then}$$

$$u = \frac{\delta\phi}{\delta x}, v = \frac{\delta\phi}{\delta y}, w = \frac{\delta\phi}{\delta z}$$

(B-1)

From the conservation of mass equation,

$$\nabla \cdot V = \frac{\delta u}{\delta x} + \frac{\delta v}{\delta y} + \frac{\delta w}{\delta z} = 0$$

(B-2)

By substituted equation (A-1) into equation (A-2), we have

$$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0$$

(B-3)

So, we end up with Laplace's equation $\Rightarrow \nabla^2 \phi = 0$, a first order partial differential equation that can be solved by applying proper boundary condition. Once the potential function is solved, we would able to able to find velocity field in the flow. With the help of Bernoulli's equation:

$$p = -\rho\left(\phi + \frac{1}{2}(\nabla\phi)^2 + gz\right) + C(t)$$

(B-4)

Which is derived from conservation of momentum, we are able to solve the pressure field of the flow. So, with both velocity field and pressure field, the fluid flow is fully described. The six components of forces and moments can be calculated by the integrals of the pressure over the body surface.

$$\text{Forces, } F = \iint_{S_B} p \hat{n} dS$$

$$\text{Moments, } M = \iint_{S_B} p(r \times \hat{n}) dS$$

Figure B 1 Forces and Moments exerted on the Body [49]

where p , pressure

S_B , Body Surface

$\hat{n}$, unit vector normal to the surface

Example: Sphere

One can derive the added mass of an object by considering the hydrodynamic force acting on it as it accelerates. Consider a sphere of radius, R , accelerating at rate $a = \frac{\delta U}{\delta t} = \dot{U}$. We find the hydrodynamic force in the x direction by integrating the pressure over the area projected in the x direction:

$$F_x = \int P dA_x$$

(B-5)

where

$$dA_x = \cos\theta dA \quad F_x = \int_0^\pi \left[-\rho \left[\frac{\delta\phi}{\delta t} + \frac{1}{2} |\nabla\phi|^2 \right] \right] \cos\theta \cdot 2\pi R^2 \sin\theta d\theta$$

$$dA = 2\pi r ds$$

$$r = R \sin \theta$$

$$ds = R d\theta$$

$$P = -\rho \left[\frac{\delta \phi}{\delta t} + \frac{1}{2} |\nabla \phi|^2 \right], \text{ by unsteady Bernoulli's equation}$$

$$\phi = U \cos \theta \frac{R^3}{2r^2}, \text{ for axis-symmetric flow around a sphere}$$

$$\left. \frac{\delta \phi}{\delta t} \right|_{r=R} = \dot{U} \cos \theta \frac{R^3}{2r^2} = \dot{U} \cos \theta \frac{R}{2}$$

$$\left. \frac{1}{2} |\nabla \phi|^2 \right|_{r=R} = \frac{1}{2} \left(-U \cos \theta \frac{R^3}{r^2}, -U \sin \theta \frac{R^3}{2r^2} \right)^2 = \frac{1}{2} \left[U^2 \cos^2 \theta + \frac{1}{4} U^2 \sin^2 \theta \right]$$

$$F_x = \int_0^\pi \left[-\rho \left[\frac{\delta \phi}{\delta t} + \frac{1}{2} |\nabla \phi|^2 \right] \right] \cos \theta \cdot 2\pi R^2 \sin \theta d\theta$$

$$\begin{aligned} &= \int_0^\pi \left[-\rho \left[\dot{U} \cos \theta \frac{R}{2} + \frac{1}{2} \left(U^2 \cos^2 \theta + \frac{1}{4} U^2 \sin^2 \theta \right) \right] \right] \cos \theta \cdot 2\pi R^2 \sin \theta d\theta \\ &= -\rho \cdot 2\pi R^2 \cdot \dot{U} \frac{R}{2} \underbrace{\int_0^\pi \sin \theta \cos^2 \theta d\theta}_{=\frac{2}{3}} - \rho \cdot 2\pi R^2 \cdot \frac{1}{2} \dot{U} \underbrace{\int_0^\pi \left[\sin \theta \cos^3 \theta + \frac{1}{4} \sin^3 \theta \cos \theta \right] d\theta}_{=0} \\ &= -\frac{2}{3} \rho \pi R^3 \dot{U} \end{aligned}$$

where $\dot{U} = a$ is the acceleration of the body and the negative sign indicates that the force is in the negative x direction, opposing the acceleration. Thus, the body must exert this extra force, and the apparent added mass is $\frac{2}{3} \rho \pi R^3$

Strip Theory

In many application that involve slender body such as ship hulls, submarine, mooring cables, the 3D added mass coefficients can be estimated by strip theory. The 3D added

mass coefficients can be found by summing up the 2D coefficients for whole length of the body. This is because flow at each section can be assumed locally to be two-dimensional.

The 2D added mass coefficients for some simpler geometry are given in Figure B-2:

Figure B 2 Added Mass Coefficients for Various Two-Dimensional Bodies [30]

The remaining added mass terms of the body shown in Figure B-2 are zero due to the geometry symmetry of the object. So, there is no coupling between the three principle modes of motion.

Since Strip theory only applicable to slender body shape, it could not be used to estimate the added mass of ROV II, which is an open frame ROV from, by multiple bodies and shapes. However, Panel method can be used in order to solve this geometry more complex problem.

APPENDIX C: PHYSIC OF HYDRODYNAMICS DRAG

What is drag?

A body moving through a fluid medium experiences a drag force, which always resist the motion of the body. The magnitude of drag force is related to the velocity of the object relative to the surrounding fluid. The drag force is larger when velocity is higher. Since it has similar effect as damping in control engineering terminology, sometime it is also call hydrodynamics damping.

Why there is drag?

Drag force is due to two distinctive phenomena of flow. So, it is usually divided into two components: frictional drag and pressure drag.

Friction Drag

Frictional drag comes from friction between the fluid and surface it flows over. This friction is developed in thin boundary layer, which has close contact with the surface. For most of the practical fluids like water are Newtonian fluid, which has following relationship:

$$\tau = \mu \frac{du}{dx} \quad (C-1)$$

Where

τ , is the shear stress exerted by the fluid ("drag") [Pa]

μ , is the fluid viscosity - a constant of proportionality [Pa·s]

$\frac{du}{dx}$, is the velocity gradient perpendicular to the direction of shear [s^{-1}]

The frictional drag is the summation of all the shear force acting on the surface. So, it is also called skin friction. For low frequency motion of a vessel, the boundary layer is laminar and thus the skin friction is linear with velocity. There is also quadratic or nonlinear skin friction due to turbulent boundary layer for high frequency motion.

When water is moving past an object then the water in direct contact with a surface is actually brought to a stop (relative to the surface). Water slightly off the surface would move slightly faster. If we go far enough from the surface then the air is moving at its full (or free stream) speed. The region just above the surface where the air is not moving at its full speed is known as the boundary layer.

Figure C 1 Boundary Layer Thickness δ and Velocity Profile i.e Variation in Velocity with y . Taken from Tara Chklovski [50]

Skin friction drag is not dependent on the particular material that an object is made of, but it is affected by how rough the surface is. As you might expect, smoother surfaces are better than rough surfaces. Also, the amount of surface in contact with the air is a factor, so minimizing the surface area will reduce skin friction drag.

Pressure Drag

While skin friction drag is a function of the surface roughness and the length of vehicle in contact with the water, the form, or pressure drag, is dependent on the shape of the vehicle. The concept of pressure drag is better explained through example of sphere.

Figure C 2 (a) Actual Separated Flow Field around a Sphere and (b) The Resulting Pressure Distribution Taken from[51]

The flow field around a sphere is shown in Figure C2. As one can see that the flow field around the sphere is no symmetrical on its front and rear face. The flow does not follow along the surface of sphere at the entire rear face. The flow is “attached” when the airflow follows along the surface. The point at which the flow breaks away from the surface is called the separation point. The region of separated flow is dominated by unsteady, recirculating vortices that create a wake.

The lower half of Figure C2 also displays the pressure distribution around the surface of the sphere, as represented by the non-dimensional pressure coefficient C_p . Positive (+) values of C_p indicate high pressure while negative (-) values indicate low pressure. The differences between high-pressure regions and low-pressure regions create hydrodynamics forces on a body, like lift and drag.

A separated flow creates a region of low pressure in the wake. This can be seen that the pressure distribution over the rear face of the sphere from $90^\circ < \theta < 270^\circ$ remain

negative (low pressure). The pressure is now much higher on the front face than it is on the rear face. So the net pressure different is the pressure drag force exerted on the sphere which is cause by the flow separation over its rear face.

What cause separation?

The cause of this separation can be seen in the above pressure distribution around the sphere. As the flow moves downstream from the $\theta = 90^\circ$ or $\theta = 270^\circ$ position, it encounters an increasing pressure. Whenever a flow encounters increasing pressure, we say that it experiences an adverse pressure gradient. The change in pressure is called adverse because it causes the airflow to slow down and lose momentum. As the pressure continues to increase, the flow continues to slow down until it reaches a speed of zero. It is at this point that the air no longer has any forward momentum, so it separates from the surface.

What affect the point of separation?

One of the key factors affecting flow separation is the behavior of the boundary layer. The boundary layer is a thin layer of air that lies very close to the surface of a body in motion. It is within this layer that the adverse pressure gradient develops that causes the airflow to separate from the surface.

At low Reynolds numbers, the boundary layer remains very smooth and the flow is laminar. Laminar boundary layers are normally very desirable because they reduce drag on most shapes. Unfortunately, laminar boundary layers are also very fragile and separate from the surface of a body very easily when they encounter an adverse pressure gradient. This separated flow is what causes the drag to remain so high below the critical Reynolds number.

At that Reynolds number, however, the boundary layer switches from being laminar to turbulent. The location at which this change in the boundary layer occurs is called the transition point. A turbulent boundary layer causes mixing of the air near the surface that normally results in higher drag. However, the advantage of turbulence is that it speeds up the airflow and gives it more forward momentum. As a result, the boundary layer resists the adverse pressure gradient much longer before it separates from the surface. Since separation is delayed, the resulting wake is much narrower. This thin wake reduces the

low-pressure region on the rear face and reduces the difference in pressure between the front and back of the sphere. This smaller difference in pressure creates a smaller drag force comparable to that seen above the transition Reynolds number.

Blunt and Streamlined Body

Figure C 3 Comparison of Flow Separation and Drag on Blunt and streamlined Shapes [51]

When the drag is dominated by viscous drag, we say the body is streamlined, and when it is dominated by pressure drag, we say the body is bluff. Whether the flow is viscous-drag dominated or pressure-drag dominated depends entirely on the shape of the body. For our ROV, which has two cylindrical tank is considered as bluff body. So, we should concentrate on the study of pressure drag.

From the above explanation, pressure drag is the result of the flow separation. Without flow separation that create low pressure wake at the rear of the sphere, there will not be any pressure drag. The magnitude of drag (drag coefficient) is very much depending on the point of separation, which will determine the thickness of wake. The point of separation is very sensitive to the condition of flow. Laminar boundary layer will separate early compare to the turbulent boundary layer. Whether the boundary layer is laminar or turbulent is depend on turbulent intensity of the flow, the roughness of the surface and the shape of the vehicle.

However, shape like thin circular disk, the separation always happen on its peripheral. So the drag coefficient is a constant at 1.17 regardless of Reynolds Numbers as shown below:

Figure C 4 Drag Coefficient of a Circular Disk and a Sphere versus Reynolds Number [52]

The coefficient of drag for a cylinder in this orientation is about 0.81 so long as the l/d (length-to-diameter ratio or fineness ratio) is greater than 2. As the fineness ratio shrinks to zero, the cylinder collapses to a flat circular disk. It is therefore not surprising that the drag coefficient for a cylinder with $l/d=0$ is about 1.17, the same as a flat disk.

Figure C 5 Drag Coefficient of Blunt Nose and rounded Nose Cylinders versus Fineness Ratio l/d [52]

If the drag coefficient is stable at high Reynolds Number, we would no experience sudden drop of drag coefficient as in the case of sphere. This makes our analysis much easier and the result is more reliable. So, for our CFD analysis on the ROV, we need to observe closely how points of separation change with velocity.

APPENDIX D: VISCOUS FLOW: NAVIER-STOKES EQUATION

Continuity equation (Conservation of mass)

$$\underbrace{\frac{\delta \rho}{\delta t}}_{\text{rate_of_mass_increase}} + \underbrace{\nabla \cdot (\rho V)}_{\text{Net_mass_flow}} = 0 \quad (\text{D-1})$$

Momentum equations (Conservation of momentum)

X component:

$$\underbrace{\frac{\delta(\rho u)}{\delta t}}_{\text{rate_change_of_momentum}} + \nabla \cdot (\rho u V) = - \underbrace{\frac{\delta p}{\delta x}}_{\text{pressure}} + \underbrace{\frac{\delta \tau_{xx}}{\delta x}}_{\text{viscous_normal}} + \underbrace{\frac{\delta \tau_{yx}}{\delta y} + \frac{\delta \tau_{zx}}{\delta z}}_{\text{viscous_shear}} + \underbrace{\rho f_x}_{\text{body_force}} \quad (\text{D-2})$$

Similarly,

Y component:

$$\frac{\delta(\rho v)}{\delta t} + \nabla \cdot (\rho v V) = - \frac{\delta p}{\delta y} + \frac{\delta \tau_{xy}}{\delta x} + \frac{\delta \tau_{yy}}{\delta y} + \frac{\delta \tau_{zy}}{\delta z} + \rho f_y \quad (\text{D-3})$$

Z component:

$$\frac{\delta(\rho w)}{\delta t} + \nabla \cdot (\rho w V) = - \frac{\delta p}{\delta z} + \frac{\delta \tau_{xz}}{\delta x} + \frac{\delta \tau_{yz}}{\delta y} + \frac{\delta \tau_{zz}}{\delta z} + \rho f_z \quad (\text{D-4})$$

For Newtonian fluids, (fluids in which τ is proportional to velocity gradient), Stokes in 1845 obtained

$\tau_{xx} = \lambda(\nabla \cdot V) + 2\mu \frac{\delta u}{\delta x}$ $\tau_{yy} = \lambda(\nabla \cdot V) + 2\mu \frac{\delta v}{\delta y}$ $\tau_{zz} = \lambda(\nabla \cdot V) + 2\mu \frac{\delta w}{\delta z}$ $\lambda = -\frac{2}{3}\mu$	$\tau_{xy} = \tau_{yx} = \mu \left[\frac{\delta v}{\delta x} + \frac{\delta u}{\delta y} \right]$ $\tau_{xz} = \tau_{zx} = \mu \left[\frac{\delta u}{\delta z} + \frac{\delta w}{\delta x} \right]$ $\tau_{yz} = \tau_{zy} = \mu \left[\frac{\delta w}{\delta y} + \frac{\delta v}{\delta z} \right]$ <div style="text-align: right;">(D-5)</div>
--	--

Substituting the above equation in momentum equation, we obtain the complete Navier-Stokes equations.

Assumption of Incompressibility

In hydrodynamics, the fluid is normally assumed to be incompressible. If the fluid is Newtonian and viscosity is constant throughout the flow, we have:

Continuity equation (Conservation of mass)

$$\nabla \cdot V = 0$$

Momentum equations (Conservation of momentum)

X component: $\rho \frac{Du}{Dt} = -\frac{\delta p}{\delta x} + \mu \nabla^2 u + \rho f_x$ (D-7)

Y component: $\rho \frac{Dv}{Dt} = -\frac{\delta p}{\delta y} + \mu \nabla^2 v + \rho f_y$ (D-8)

Z component: $\rho \frac{Dw}{Dt} = -\frac{\delta p}{\delta z} + \mu \nabla^2 w + \rho f_z$ (D-9)

where D/Dt is the substantial derivative $\frac{D}{Dt} = \frac{\delta}{\delta t} + u \frac{\delta}{\delta x} + v \frac{\delta}{\delta y} + w \frac{\delta}{\delta z}$ (D-10)

APPENDIX E: TYPICAL STAGES IN CFD

Computational Fluid Dynamics (CFD) is a computer-based simulation tool used to analyze system involving fluid flow, heat transfer and other related physical process such as chemical reaction. It works by solving a set of mathematics equations that describe the system behavior over a region of interest. Generally, the region is represented by using discrete element (grid or mesh) and proper conditions must be applied at the boundary.

Development of general purpose CFD is started at the mid-1970. In the early 1980's, CFD was used exclusively in research because it need a very powerful computer, in-depth knowledge in fluid dynamics and large amount of time. It is only until recent advancement in computer power and development of interactive graphic user interface, CFD is widely used in industry for variety of engineering problems.

Advantages

- Cheaper than building a high quality experiment facility
- Reduction of lead times and costs of new designs
- Idea for parametric study
- Ability to study system under different condition where real experiments are difficult or impossible to perform
- Great level of detail result

Disadvantages

- Not cheap to purchase software license
- Required High Level of Expertise and Experiences to obtain reliable result

Figure E 1 Flow chart of CFD process

There are three typical stages of a CFD project. We will briefly examine the function of each of these three stages.

Pre-processing

Physical Model

The very first stage of setting up a CFD simulation is a stage of identification and formulation of the flow problem. The objectives of the study must be stated clearly, so that relevant physical or chemical phenomena will be understood and considered. For example, users have to decide whether to model a problem in two or three dimensions, to include or exclude the effect of temperature and pressure on fluid properties, the use of lamina or turbulence model or both of them. It required a good modeling skills and solid

understanding of the fluid flow phenomena to make right decisions. Most of the time, the fluid flow phenomena are complex in nature and simplification of the problem is needed. Therefore, the users need to aware all the assumptions that have been made and their effect to the simulation result.

The successfulness of using CFD in flow simulation is much depends on the understanding of the fundamental flow phenomenon. So, in this step the concept of fluid flow need to be addressed clearly.

Mathematical Model

The physical phenomena need to be described by a set of mathematics equation. The mostly used equation is known as the Navier-Stokes equations, which describe the exchange of mass and momentum of the fluid particles. Equations describing other processes, such as turbulence, heat transfer and chemical reaction can be solved conjunction with Navier-Stokes equations.

Geometry Build Up

The geometry of physical objects needs to be defined by mean of computer aided design (CAD). Geometry of the objects needs to be defined and drawn using commercial available CAD software in market such as SolidWorkTM, ProETM, SolidEdgeTM, AutoCadTM. The choice of CAD software is much depend on the next step of pre-processing where mesh need to be generated throughout the whole domain of interest. So, the CAD model of the geometry must define in a format (file extension) that can be imported to the mesh generator.

Meshing Domain

The mathematic equations that describe the physical system form a set of partial differential equations (PDEs) for the variable of interest such as pressure p and velocity V . Those equations are valid over a continuous domain thus representing an infinite number of degree of freedom.

To solve these equations, first we must perform 'discretization'. Discretization is a process by which a closed form mathematic expression is approximated by analogous (but different) expressions, which prescribe values at only a finite number of discrete

points or volume in the domain. So, the problem is reduced to finding the unknown values of variables at the node points.

We need to transform the continuum equation into finite set of equations that form a matrix, which can be solved numerically. Different types of elements are available to conduct the transformation. The user has to decide according to the condition, experience and choices given by the software.

The arrangement of these discrete points throughout the flow field is simply call a grid or mesh. The determination of a proper grid for the flow over or through a given geometry shape is a serious matter-one that by no means trivial. The matter of grid generation is a significant consideration in CFD; the type of grid you choose for a given problem can make or break the numerical solution. Because of this, grid generation has become an entity by itself in CFD. More than 50% of the time spent on a CFD project is on definition of the domain geometry and mesh generation.

Computations

Boundary Condition & Environment Model Set Up

In order to solve a set of PDEs, proper boundary conditions need to be defined depend on the real system. Some commonly used boundary conditions are inlet, outlet, free slip wall, non-slip wall and symmetry. In addition, the initial conditions of the fluid flow such as fluid flow speed, and pressure need to be set also. The domain fluid and its property also required to be select from the build-in library or defined by the users. Besides, the choice of the solver advection scheme, convergence criteria and maximum iteration also need to be defined

Solver (Iteration)

The CFD problem is solved numerically and iteratively by the solver. Generally, a solver performs the following three steps to obtain the solution of the fluid flow. First, the unknown flow variables are approximated by means of simple function. Then, the approximations are substituted into the governing equations and converted to a system of algebraic equations. Finally, the algebraic equations are solved iteratively until the convergence criteria is meet.

Normally, solver is run as a batch process and required no user interaction. However, a good solver software will allow users to monitor the solving process by showing the residual of some key variables. So, user may stop the iteration or edit the solvers' setting if something unexpected is observed. This is important because solver usually take a very long time to yield the solution. There will be waste of time and computer resource if error is found only after the iteration.

Post-processing

Result (visualization)

In the post-processing stage, all type of flow phenomena can be plotted or visualized in graphical form. These include but not limited to:

- Domain geometry and mesh display for examine the quality of mesh
- Vector Plot to display vector quantity like velocity, forces
- 2D and 3D surface and contour plot to show scalar quantity like pressure, magnitude of velocity
- Particle tracking and streamline plot to visualize the flow
- Animation

Analysis & Validation

The result of the simulation cannot be trusted simply because the result has converged and produced high quality of color plot. So the main tasks in this step are to determine whether the result is sensible and accurate. There are number of ways to do the checking. But the most straight forward method is to compare the simulation result with the experiment result. If the result is closely matched, then the simulation is representing the reality. However, the experiment result is rarely known.

Besides comparing with experiment result, user could check the conserved variable such as overall force or momentum balance. Flow variables such as velocities, pressure must have reasonable value and distributed continuously over the domain. It is only changing rapidly only at where it is expected such as flow over a shape edge. Discontinuities are

the signal of poor panel definition or insufficient mesh. The convergence history may tell whether the solution has reach steady state.

APPENDIX F: THRUSTERS IDENTIFICATION

Thrusters' Configuration:

Figure F1 Thruster Configuration on the ROV platform

There are six degrees of freedom for any object in 3-dimensional space, namely 3 positions (x , y and z) and 3 orientations (roll, pitch and yaw). The ROV II has only four thrusters as showed in Figure F1. The number of actuators is less than the degrees of freedom. From the control theory perspective, this is classified as an under-actuated system, and this makes it harder to control as compared to fully-actuated system.

The thrusters' positions have great influence on the way the ROV maneuvers in water. The allocation of the thrusters must enable the ROV to move in the surge, heave and yaw direction. These form a cylindrical coordinate as shown in Figure F2 and allow ROV to reach any point in 3-D space. The ROV is not likely to be positioned with a non-zero pitch and roll angles. Thus, this allocation of thrusters is usually sufficient.

Figure F2 Cylinder Coordinate

- **Surge**

For this direction, both thrusters T_1 and T_2 are needed to advance ROV straight ahead. The torque generated in z-axis should cancel each other. So, there will not be any yaw motion. However, torque generated about y-axis could not be annulled as thrusters T_1 and T_2 are mounted higher than the centre of gravity. Hence, there will be a resultant pitch motion which will have to be stabilized by buoyancy.

- **Heave**

To move ROV in the heave direction, thrusters T_3 and T_4 must be activated simultaneously. Any imbalance between the generated thrust will cause motions in the sway and roll direction. Hence, the ROV must work under closed loop control to minimize unwanted motion.

- **Yaw**

In order to turn ROV about its z-axis, voltage are pumped to thrusters T_1 and T_2 simultaneously but with opposite sign. As each thruster is located at a fixed distance from the centre of gravity, the resultant torque will cause desired yaw movement. If the thrust force generated by T_1 and T_2 are the same and in opposite direction, then the force in x-component will cancel each and other. In this way, there will be not motion in pitch and surge direction.

The resultant 3 forces and 3 moments could be calculated using the following equation when the thrusts from each thruster is known:

$$\tau = \begin{bmatrix} X \\ Y \\ Z \\ K \\ M \\ N \end{bmatrix} = \begin{bmatrix} T_1 + T_2 \\ (T_3 - T_4) \sin \beta \\ (T_3 + T_4) \cos \beta \\ -\delta(T_3 - T_4) \\ -\delta(T_1 + T_2) - \alpha(T_3 + T_4) \cos \beta \\ \gamma(T_1 - T_2) + \alpha(T_3 - T_4) \sin \beta \end{bmatrix} = \begin{bmatrix} 1 & 1 & 0 & 0 \\ 0 & 0 & \sin \beta & -\sin \beta \\ 0 & 0 & \cos \beta & \cos \beta \\ 0 & 0 & -\delta & \delta \\ -\varepsilon & -\varepsilon & -\alpha \cos \beta & -\alpha \cos \beta \\ \gamma & -\gamma & \alpha \sin \beta & -\alpha \sin \beta \end{bmatrix} \begin{bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \end{bmatrix}$$

Thruster dynamics

In the great majority of the applications, thrust provided by the thruster is described by [4]

$$T = an|n| - bnv_a$$

T thrust

n angular velocity

v_a velocity of advance of the water through the propeller blades

a, b constant

The saturation term bnv_a may be very important at high speeds, but is usually neglected in standard low speed operating conditions of the ROV is widely accepted in the literatures.

A simple steady state model is used. The propeller revolution rate is assumed to be linear with respect to the applied DC motor voltage. The model which neglects the motor dynamics is a good approximation when the propeller does not suddenly change direction. Generally the servo velocity loop of the velocity controlled thruster system has a negligible time constant with respect to the overall vehicles' one, and thus the thruster dynamics can be neglected.

$$n = KV$$

In the sequel the thruster applied force will be modeled as

$$T = aK^2V|V| = C_T V|V|$$

V is the applied DC motor voltage and C_T is a constant to be experimentally identified. The identification of the C_T constant in above equation is generally performed with a single thruster in a cavitation tunnel and by measuring the thrust as a function of the applied voltage. However, due to propeller-propeller and propeller-hull interactions, the thrust generated by a single thruster in the cavitation tunnel may differ from that in the actual implementation when thrusters are mounted close to each other and to the vehicle.

“Experimental data relative to the ROMEO vehicle show that the thruster’s efficiency loss, with respect to the open water thrust tunnel thruster model, due to propeller–

propeller and propeller–hull interactions is greater than 10% for each considered axis and reaches the significant value of 44% for heave.”[6]

As there will be efficiency loss after the thrusters are installed in the vehicle, due to propeller–propeller and propeller–hull interactions, an experiment has been set up to measure the thrust directly with thrusters operating on the vehicle. The resultant thrust measured will be more accurate than using the algebraic sum of the thrusts from each thruster computed using the model determined from cavitation tunnel test.

Figure F 3 Experiment Setup

In many marine applications two different parameters are required for positive and negative revolution, as thrusters do not behave symmetrically in the two directions. So, experiment must be done for both operating conditions.

Experiment Result

Angular velocity versus voltage inputs for four thrusters

The relationship between input voltages and propeller revolutionary rate for each thruster is studied. The angular velocity of the propeller is measured using a digital optical handheld tachometer. The experiment is done in open air environment.

The results for the four thrusters are plotted in Figure 7. All four thrusters operate at identical revolution speed under different input voltages. Hence, it is reasonable to assume that T1 and T2 will produce similar thrust if given same input voltage. T3 and T4 also produce similar thrust when given same input voltage.

Figure F 4 Angular velocity versus Input Voltage for four thrusters

Steady State angular velocity of the thruster under varying input voltages (5V to 48V).

The relationship between angular velocity and voltage is linear and can be expressed as

$$n \approx 43V$$

Thrust versus voltage input

Equipments

Figure F 5 Force sensor attached on the frame

Figure F 6 Real-Time Data Acquisition of the force

For thruster 1 and 2 (Horizontal thrusters)

Forward				Reverse			
Voltage(V)	V V	Total Forward thrust(N)	Forward thrust/2	Voltage(V)	V V	Total Backwards thrust(N)	Backwards thrust /2
0	0	0	0	0	0	0	0
10	100	6.83	3.42	10	100	2.04	1.02
15	225	16.30	8.15	15	225	6.89	3.45
20	400	27.92	13.96	20	400	11.66	5.83
25	625	44.67	22.34	25	625	19.37	9.69
30	900	59.12	29.56	30	900	27.54	13.77
35	1225	76.74	38.37	35	1225	37.59	18.80

For thruster number 3 and 4 (vertical thrusters)

Forward				Reverse			
Voltage(V)	V V	Total Forward thrust(N)	Forward thrust/2	Voltage(V)	V V	Total Backwards thrust(N)	Backwards thrust /2
0	0	0	0	0	0	0	0
10	100	1.86	1.32	10	100	1.86	1.32
15	225	3.96	2.80	15	225	5.21	3.68
20	400	7.57	5.35	20	400	9.94	7.03
25	625	13.49	9.54	25	625	16.2	11.46
30	900	22.26	15.74	30	900	21.92	15.50
35	1225	31.86	22.53	35	1225	26.61	18.82

Discussion

The relationship between the input voltage to the thruster and its generated thrust had been identified. The identification is carried out with the thrusters mounted on the ROV. So, the thrusters' efficiency losses due to propeller-propeller and propeller-hull interactions are taken into account. Thus, more accurate result is obtained compared to the open water thrust tunnel thruster test.

The thrust generated by the thruster is not symmetrical in both forward (clockwise) and reverse (anti-clockwise) direction. The thrust provided by the two horizontal thrusters, T1 and T2 have the following relationship:

$$T = 0.0326V^2 \text{ Forward}$$

$$T = -0.0153V^2 \text{ Reverse}$$

The thrust provided by the two vertical thrusters, T3 and T4 have the following relationship:

$$T = 0.0175V^2 \text{ Forward}$$

$$T = -0.0122V^2 \text{ Reverse}$$

The vertical thrusters produce lower effective thrust compare with the horizontal thrusters. This is expected as the vertical thrusters' location has a higher induced propeller-hull interaction. The interaction causes a loss in thruster efficiency.

Remark:

The transient property of the thruster is not captured in the above steady-state equation. However, the transient effect is negligible for slow moving underwater vehicle