

TITLE:**Scale effect on the wake field of a Single Screw Ship****CANDIDATE NAME:**

Yue Ding Candidate No. 2110

DATE:

29/05/2015

COURSE CODE:

IP501909

COURSE TITLE:

MSc thesis, discipline oriented

RESTRICTION:**STUDY PROGRAM:**

MSc – Ship Design

PAGES/APPENDIX:

69/54

LIBRARY NO.:**SUPERVISOR(S):**

Karl H. Halse

ABSTRACT:

Nowdays, the CFD method has been become a famous approach that use to solve the problem involved the ship hydrodynamic problems. The objective of this master thesis is attempting to simulate a single screw stern ship sailing in the calm water with several different ship speed with the help of CFD program Star CCM+. Simulations contain both model scale situations and full scale situations. By analysis the results of the ship resistance coefficient and wake field of both cases, we figure out that the resistance and wake differences between model scale ship and full scale ship and what caused the scale effect between the real full scale ship and the model scale ship.

Master thesis mainly consist of four parts: background and methods, which is use to define the basic theoretical foundation of this thesis; Case study part introduces the model, mesh and physical parameters setting process in the CFD computer program: Star CCM+; Results part shows the results from simulations we get and Discussion part discuss the questions we encounter during the simulation. The entire simulation process in this thesis contains five phases: 1. Setup simulated models and meshes; 2. Setup physical parameters; 3. Verification of results; 4. Simulate all the test cases; 5. Analyze the results.

This thesis is submitted for evaluation at Ålesund University College.

Postal address:Høgskolen i Ålesund
N-6025 Ålesund
Norway**Visit adress**Larsgårdsvegen 2
Internett
www.hials.no**Telephone**

70 16 12 00

E-mailpostmottak@hials.no**Fax**

70 16 13 00

Bank

7694 05 00636

Enterprise no.

NO 971 572 140

MASTER THESIS 2015

FOR

STUD. TECHN. YUE DING

Scale Effects on the Wake Field of a Single Screw Ship

Background.

In these year civilian ships designers has drawn more attention to focus the fuel consumption and ship power efficiency problem because the economic crisis. Both ship owners and ship builders are put the economy in the first consideration. And the nominal wake effect, is one of the major factor that influence the propulsion efficiency of the ship.

But during the ship model towing test, nominal wake is quite different from the full scale ship due to the reasons such as the difference of viscous resistances between model scale ships and real ships, So find out what is the differences by compare the results is the objective of this master thesis. This research will be helpful for the future ship model resistance test and real ship nominal wake estimation.

Objectives.

In this master thesis we will research the scale effects on the wake field behind a single screw ship, which is named Japan bulk carrier, the objective is trying to find out the differences of ship resistance coefficient and nominal wake between model scale ship and full scale ship with the help of CFD software. And try to find out the discipline of these differences. Research mainly consists of two parts: CFD simulation phase and results analysis phase.

The thesis work shall include the following:

- Literature survey
 - Knowledge of ship resistance estimation: Model test and Full scale numerical analysis
 - Knowledge of CFD and ship modeling ability that related to computer software
 - Details research plan
- Prepare the simulation resource
 - Select one single screw vessel (JBC).
 - Specification the simulated cases: Froude number & Draught
 - Set the measure region aft ship and define the amounts of measurements
 - Parameters variation:
 - Turbulence model
 - Prism layer
- Simulate test cases by using software Star CCM+
 - Set the simulated model and the parameters before simulation, consist of simulate method, mesh and physical parameters.
 - Simulate the cases in different ship speed (Model scale and full scale). For each condition, draught must be fixed and has no free sinkage and trim.

- Using different CFD method or mesh types in Star CCM+ to check if the results are correct
- Record the flow rate in the measure region. Calculate the nominal wake.
- Analyze the results
 - Compare the resistance in different conditions (Model and Full scale).
 - Compare the nominal wake in different conditions (Model scale and Full scale), find differences between model and full size.
 - Give the reasons why scale effects happen.

The scope of work may prove to be larger than initially anticipated. Subject to approval from the advisor, topics from the list above may be deleted or reduced in extent.

The thesis should be written as a research report with summary, conclusion, literature references, table of contents, etc. During preparation of the text, the candidate should make efforts to create a well arranged and well written report. To ease the evaluation of the thesis, it is important to cross-reference text, tables and figures. For evaluation of the work a thorough discussion of results is needed. Discussion of research method, validation and generalization of results is also appreciated.

In addition to the thesis, a research paper for publication shall be prepared.

Three weeks after start of the thesis work, a pre-study have to be delivered. The pre-study have to include:

- Research method to be used
- Literature and sources to be studied
- A list of work tasks to be performed
- An A3 sheet illustrating the work to be handed in.

A templates and instructions for thesis documents and A3-poster are available on the Fronter website under MSc-thesis. Please follow the instructions closely, and ask your supervisor or program coordinator if needed.

The thesis shall be submitted in electronic version according to new procedures from 2014. Instructions are found on the college web site. In addition one paper copy of the full thesis with a CD including all relevant documents and files shall be submitted to your supervisor.

Supervision at AAUC: Karl H. Halse,
Contact at : Aalesund University College

Karl H. Halse
Supervisor



Delivery: 15.01.2015

Signature candidate: Yve Diney

PREFACE

The project of the master thesis is attempting to use the famous CFD computer program to solve the ship hydrodynamic problems. Topic's name is "Scale effect on the wake field of a Single Screw Ship", which is given by the Norwegian Marine Technology Research Institute (MARINTEK). These simulations are doing at the same time in Marinetek.

The author's name is Yue Ding, who has graduated from Dalian Maritime University, Dalian, China in 2012. Major in ship design, and has the relevant knowledge of the ship hydrodynamics and structure. Right now Yue Ding is study in the ship design master degree in Ålesund University College from 2013.

The supervisor of this master thesis is Karl Henning Halse, the Professor of Ålesund University College. Thanks he gave lots of help and advices to this master thesis. We also obtained much help from HIA's teacher Gunnar Hugo Nyland and the research scientist of Marinetek Vladimir Igorevich Krasilnikov.

ABSTRACT

Nowdays, the CFD method has been become a famous approach that use to solve the problem involved the ship hydrodynamic problems. The objective of this master thesis is attempting to simulate a single screw stern ship sailing in the calm water with several different ship speed with the help of CFD program Star CCM+. Simulations contain both model scale situations and full scale situations. By analysis the results of the ship resistance coefficient and wake field of both cases, we figure out that the resistance and wake differences between model scale ship and full scale ship and what caused the scale effect between the real full scale ship and the model scale ship.

Master thesis mainly consist of four parts: background and methods, which is use to define the basic theoretical foundation of this thesis; Case study part introduces the model, mesh and physical parameters setting process in the CFD computer program: Star CCM+; Results part shows the results from simulations we get and Discussion part discuss the questions we encounter during the simulation. The entire simulation process in this thesis contains five phases: 1. Setup simulated models and meshes; 2. Setup physical parameters; 3. Verification of results; 4. Simulate all the test cases; 5. Analyze the results.

TABLE OF CONTENT

ABSTRACT	5
TABLE OF CONTENT	6
LIST OF FIGURES.....	9
LIST OF TABLE.....	10
TERMINOLOGY.....	11
SYMBOLS.....	11
ABBREVIATIONS.....	11
1 INTRODUCTION.....	12
1.1 PROJECT BACKGROUND.....	12
1.2 PROBLEM FORMULATION	12
1.3 OBJECTIVES.....	13
2 BACKGROUND AND THEORETICAL BASIS.....	13
2.1 CFD.....	13
2.1.1 <i>What is CFD?</i>	13
2.1.2 <i>EFD</i>	14
2.1.3 <i>EFD VS CFD</i>	14
2.2 SHIP RESISTANCE (ITTC SHIP TOWING TEST).....	14
2.2.1 <i>Resistance components</i>	14
2.2.2 <i>ITTC Towing Test</i>	15
2.3 NOMINAL WAKE	15
2.3.1 <i>What is nominal wake?</i>	15
2.3.2 <i>How to measure wake</i>	16
2.4 SCALE EFFECT	17
3 METHODS	19
3.1 MAIN STAGES OF CFD SIMULATION	19
3.2 THE FUNDAMENTAL LAWS OF FLUID.....	19
3.2.1 <i>Forces acting in the fluid</i>	19
3.2.2 <i>Basic laws of physics</i>	21
3.2.3 <i>Simplifications of equations (Assumptions of fluid)</i>	21
3.3 DISCRETIZATION METHOD.....	23
3.4 SHIP SIMULATION IN STAR CCM+	23
4 CASE STUDY.....	25
4.1 MODEL DESCRIPTION.....	25

4.2	SET UP PARAMETERS	26
4.2.1	<i>Main stage</i>	26
4.2.2	<i>Domain Description</i>	27
4.2.3	<i>Generate Mesh</i>	28
4.2.4	<i>Physics setting</i>	32
5	RESULTS	40
5.1	TEST CASES DESCRIPTION	40
5.2	EMPIRICAL VALUE	41
5.3	RESIDUALS:	42
5.4	K- Ω TURBULENCE MODEL & K-E TURBULENCE MODEL (VERIFICATION).....	44
5.5	FULL SHIP AND HALF SHIP	47
5.5.1	<i>Model scale</i>	47
5.5.2	<i>Full scale</i>	49
5.6	COMPARISON OF SHIP RESISTANCE COEFFICIENT.....	51
5.6.1	<i>Resistance coefficient for model scale ship</i>	51
5.6.2	<i>Resistance of full scale ship</i>	53
5.6.3	<i>Scale effect of resistance coefficient</i>	55
5.6.4	<i>The distribution of resistance</i>	56
5.7	WAVE PATTERNS	57
5.7.1	<i>The scale effects of wave patterns</i>	59
5.8	WAKE FIELD.....	60
5.8.1	<i>Comparison of wake field on the propeller plane</i>	60
5.8.2	<i>The mean value of nominal wake fraction</i>	61
6	DISCUSSION	63
6.1	CP INFLUENCED BY BULBOUS BOW	63
6.2	THE CAUSES OF SCALE EFFECTS	64
6.3	RESULTS INFLUENCED BY MESH & PHYSICAL PARAMETERS	66
6.3.1	<i>Friction resistance coefficient & Prism layers</i>	66
6.3.2	<i>Pressure resistance coefficient & VOF Waves Damping length</i>	66
6.3.3	<i>Waves & Parameters setting</i>	66
7	CONCLUSION	68
	REFERENCES	69
APPENDIX A1.	Y⁺ NUMBER	1
APPENDIX A2.	WAVE PATTERNS	7
APPENDIX A3.	CF	13
APPENDIX A4.	CP	19
APPENDIX A5.	CT	25
APPENDIX A6.	WAKE FIELD	31
APPENDIX A7.	WAKE TABLE	37

APPENDIX B. PROPSCALE OF MASTER THESIS..... 1

LIST OF FIGURES

2. BACKGROUND AND THEORETICAL BASIS

Figure2. 1. Ship resistance distribution	15
Figure2. 2. The causes of wake	16
Figure2. 3. Measurement region in model towing test	17
Figure2. 4. Measurement region in Star CCM+	17
Figure2. 5. Equations of simulation environments.....	18

3. METHOD

Figure3. 1. Fluid particle	20
Figure3. 2. Simplification of fluid model	22
Figure3. 3. Discrete method	23
Figure3. 4. Simulation method in Star CCM+	24

4. CASE STUDY

Figure4. 1. The JBC Model	25
Figure4. 2. Domain of simulation	27
Figure4. 3. Half domain	28
Figure4. 4. Hexahedron mesh cell.....	29
Figure4. 5. Automated mesh technology.....	29
Figure4. 6. Mesh of domain	29
Figure4. 7. Near-wall prism mesh around ship surface.....	30
Figure4. 8. Y^+	31
Figure4. 9. Wall Y^+ number in Model scale ($Fr = 0.1423$).....	31
Figure4. 10. Physical settings.....	32
Figure4. 11. The pressure-based segregated algorithm	33
Figure4. 12. The boundary conditions of our region and half region.....	36
Figure4. 13. CFL number plot of simulation.....	38

5. RESULTS

Figure5. 1. Residuals for model scale_ $Fr = 0.1423$ _ SST $k-\omega$ turbulence model	43
Figure5. 2. Residuals for model scale _ $Fr = 0.1423$ _ $k-\epsilon$ turbulence model	43
Figure5. 3. Wave patterns of ship (SST $k-\omega$ turbulence model)	45
Figure5. 4. Wave patterns of ship ($k-\epsilon$ turbulence model)	45
Figure5. 5. Wake in propeller plane (SST $k-\omega$ turbulence model).....	46
Figure5. 6. Wake in propeller plane ($k-\epsilon$ turbulence model).....	46
Figure5. 7. Streamlines of full ship (Model scale)	48
Figure5. 8. Comparison of the wake field of full ship and half ship	49
Figure5. 9. Comparison of streamline of full and half (Full scale)	50
Figure5. 10. C_p for half ship case (Full scale)	51
Figure5. 11. Residuals for half ship case (Full scale).....	51
Figure5. 12. C_p number of model scale ship ($Fr = 0.1423$)	52

Figure5. 13. Resistance coefficient of model scale ship.....	53
Figure5. 14. The resistance coefficient.....	54
Figure5. 15. Resistance coefficient of full scale ships	55
Figure5. 16. Comparison of resistance coefficient of model and full scale	56
Figure5. 17. The proportion of friction resistance.....	57
Figure5. 18. The components of wave pattern	58
Figure5. 19. Wave patterns for model scale ship ($Fr = 0.1423$)	58
Figure5. 20. Wave pattern s for full scale ship ($Fr = 0.1423$)	59
Figure5. 21. Comparison of wave patterns.....	59
Figure5. 22. The wake field on the propeller plane of model scale ship ($Fr = 0.1423$).....	60
Figure5. 23. The wake field on the propeller plane of full scale ship ($Fr = 0.1423$).....	61
Figure5. 24. Velocities on the propeller	61
Figure5. 25 Mean value of wake fraction for all cases.....	62

6. DISCUSSION

Figure6. 1. Pressure resistance coefficient	63
Figure6. 2. The curve of the C_p with/without bulbous bow.....	63
Figure6. 3. a) Boundary layer b) Flow separation.....	64
Figure6. 4. Streamline at aftship of model and full scale ship ($Fr = 0.1423$).....	65
Figure6. 5. Wave patterns for different speed ships (Model scale).....	66

LIST OF TABLE

4. CASE STUDY

Table4. 1. The dimensions of JBC ship.....	26
Table4. 2. Position of ship.....	26
Table4. 3. Domain of model scale and full scale.....	27
Table4. 4. The mesh setting for model and full scale case	32
Table4. 5. Basic values of the environment	34
Table4. 6. The dimensions of propeller.....	39

5. RESULTS

Table5. 1. Test cases for model scale	41
Table5. 2. Test cases for full scale	41
Table5. 3. Compare with two method and EFD value	44
Table5. 4. Mean wake value in the propeller plane.....	46
Table5. 5. Comparison of ship resistance coefficient of full and half ship (Model scale)	48
Table5. 6. Comparison of ship resistance coefficient of full and half ship (Full scale)	50
Table5. 7. The resistance of model scale ship	52
Table5. 8. Resistance coefficient of full scale ship	54
Table5. 9. Average wake fraction.....	62

TERMINOLOGY

Symbols

ρ	Fluid density [kg/m ³]
V	Ship speed [m/s]
V_N	Flow advanced speed in front of propeller [m/s]
w	wake fraction
η_H	Ship hull efficiency
t	Ship propulsion reduction
Fr	Froude number
Re	Reynolds number
P	Pressure [Pa]
μ	Viscosity of fluid
C	Courant number
C_R	Resistance coefficient

Abbreviations

CFD	Computational Fluid Dynamics
EFD	Experimental Fluid Dynamics
ITTC	International Towing Tank Conference
GUI	Graphical User Interface
N-S equation	Navier – Stokes equations
RANS	Reynolds Average Navier – Stokes equations
JBC	Japan Bulk Carrier
SRC	Ship Building Research Centre of Japan
NMRI	Naval Medical Research Institute
CFL number	Courant–Friedrichs–Lewy number

1 INTRODUCTION

1.1 *Project background*

For a long time, the ship hydrodynamics tests, such as the resistance tests are always based on the laboratory towing test. This method not only to spend lots of human and material resources, but also has lots of disadvantages, before the real ship sea trial, naval architects don't know if their results from towing test are correct enough. This master thesis paper aims to study how to use the new CFD method in a computer to arrive at test results that used to guide the hull form design.

In the past, ship designers most use parent ship transformation method to design a new type of ship hull. But after using CFD method, designers can design a totally new type of ship by determined the requirements. This method will bring ship design industry a good development.

Computational Fluid Dynamics (CFD) first appeared in 1960s with the development of computers and the rapid rise of disciplines. After nearly fifty years progress, this discipline has been quite mature. An important sign of maturity is that over the past decade, a variety of commercial CFD software is used in various industries. The performance and the range of applications are expanding. So far, the use of CFD technology has long been beyond the scope of traditional hydrodynamics and fluid engineering, such as aviation, aerospace, Ships, power, water conservancy, so as to extend to the chemical industry, nuclear energy, metallurgy, construction, environment and many other related fields.

In ship hydrodynamic area, CFD method mainly uses to solve the problems such as simulate the ship sailing status, compute the hydrodynamic coefficients of the ship and simulate the propeller. From the past literatures, we can know that for low single-hull vessels, using CFD technology to simulate can basically meet the needs of engineering applications and the satisfactory results. And in this thesis, we try to use this CFD method to simulate the sailing status of a single screw ship which named "Japan Bulk Carrier".

1.2 *Problem formulation*

When naval architects want to design a new type of ship or refining the design of a ship to improve the ship's performance at sea, they always make a ship model that used to carry out hydrodynamic tests, which we call it model testing. Designers estimate the data of ship hydrodynamics of full scale ship by measuring the value in model testing.

But even though for years research for the relationship between model scale and full scale, when we try to design a new type ship, we can't assert that the empirical equations are suitable for the new ship, the differences between different ship type still exist. So, the problems are:

1. What are the resistance and wake differences between model scale ship and full scale ship?
2. How much these differences are?

3. What caused the scale effect between the real full scale ship and the model scale ship?

In this master thesis, when we do the simulations, we bring these questions into account and try to figure out the solutions to them.

1.3 Objectives

In general, there are two reasons that why we measuring the velocities behind the ship model without propeller in towing condition:

1. The knowledge of velocity distribution in the wake of ship gives us a possibility to determine the viscous resistance component of the ship and in this way "we can separate the different resistance components.
2. The knowledge of the mean value of the velocity in the place of the propeller gives us a possibility to determine the resistance coefficient between the ship hull and the water going through the propeller disc area. [1]

In this master thesis, the objective is researching the scale effects on the wake field after a single screw ship, the ship is sailing in a calm water condition with several constant speeds, By simulate the both model scale and full scale ship in the CFD program, we want to find out the resistance coefficient and nominal wake differences between them with the help of CFD computer software: Star CCM+. And by analyzing the results, figure the reasons cause these scale effects.

2 BACKGROUND AND THEORETICAL BASIS

This section shows the overview of applied theories to solve the problem what we have define. Contains the theoretical foundation what we used and basic knowledge we concerned in this master thesis.

2.1 CFD

2.1.1 What is CFD?

CFD is the abbreviation for the computational fluid dynamics. This is an approach that uses numerical methods and algorithms to solve problems that involve fluid flows. Usually, computer software is the tool that used to prepare the data, build the computational domain, mesh and setting parameters to simulate the interaction of liquids and gases with surfaces defined by boundary conditions [2]. Generally speaking, CFD method provides scientists and engineers a place to perform “numerical experiments”.

The fundamental basis of almost all CFD problems is the Navier–Stokes equations, which define any single-phase (gas or liquid, but not both) fluid flow. Besides of this, CFD program also has a multiphase flow model, free surface flow model as well as non-Newtonian fluid models with Navier-Stokes equations coupled. Most of the additional source model is to add some additional items on the body equations, additional transport equations and relationships. With the advent of expanding the range of applications and new methods, the new model is also increasing. [2]

Throughout the whole this thesis project process, we are used CFD method to simulate the ship sailing condition in both model and full scale. Try to figure out the results of the ship resistance and wake field aft ship with the help of CFD computer software. The computer tool is called “Star CCM+”.

2.1.2 EFD

Experimental Fluid Dynamics (EFD), which is by using the physical experimental methodology and procedures to solve fluids engineering systems problems [3]. For ship fluid dynamics problems, the usual practice is performed for a laboratory scale model possibility of scale effect to perform the experiments. Always, the experiments cost lots of time and money in the preparation phase because of geometry modelling and manufacture. And only can provide one quantity at one time.

2.1.3 EFD VS CFD

Compared with Experimental Fluid Dynamics (EFD), CFD simulation costs less time and money because all of the experiments in CFD are built and simulate in a virtual computer environment. Also, CFD simulations can measure all desired quantities at one time, don't need to consider the special care of the simulation condition, and the most important, CFD can do virtually for any scale and any actual flow domain.

In general, CFD can't replace the experimental measurements completely, but the amounts of experimentation and the overall cost can be significantly reduced. [2]

But, in another way, the results of a CFD simulation are never 100% reliable, because:

- The input data may has too much guessing or assumptions
- The mathematical model of the problem may be inadequate
- The accuracy of the results is limited by the available computing power[4]

Overall, each method has its own advantages. In real life, for a new type ship designing process, we suggest that do both EFD and CFD calculations in order to ensure the correctness of results.

2.2 Ship Resistance (ITTC Ship Towing Test)

2.2.1 Resistance components

Ship resistance is the force that required to tow the ship when the ship is sailing in calm water with a constant velocity. In general, the total resistance consists of two parts: one is friction resistance that caused by the viscosity between water and ship surface, or we can say caused by tangential stresses due to the drag of the water moving parallel to the surface of the vessel. This resistance is related to the Reynolds number Re ; The other one is called residual resistance, which is caused by the distribution of pressure which develops about the hull because of the waves and eddies occasioned by the ship's motion and related to the Froude number Fr . Figure 2.1 show the compositions of the ship resistance.[5]

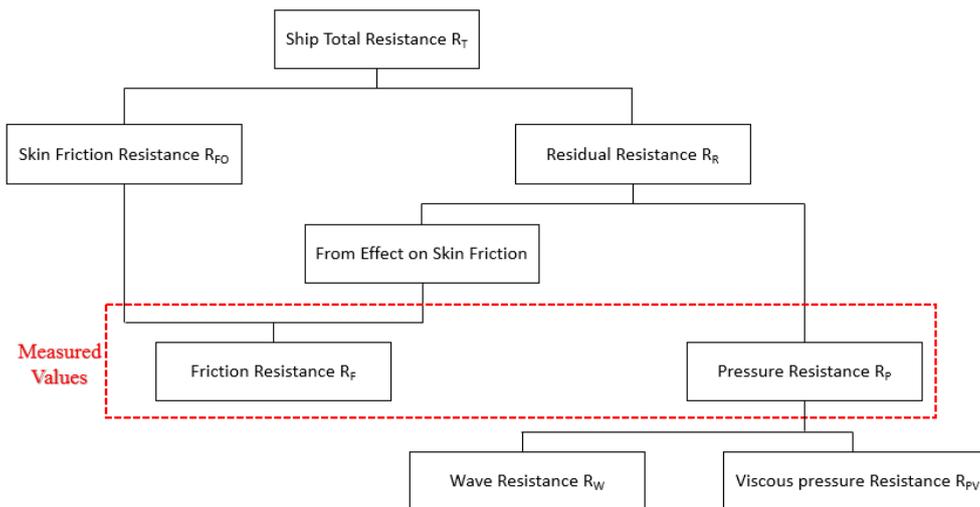


Figure2. 1. Ship resistance distribution

Ship resistance coefficient is one of the results that have to be processed in this master thesis project. In Star CCM+, “Friction Resistance” and “Pressure Resistance” can be measured directly

Air resistance also is one of the components of ship resistance, in the CFD simulation phase, air condition is considered as well, but when it compare with water resistance, air resistance’s value too small that can be ignored in the simulation.

2.2.2 ITTC Towing Test

The International Towing Tank Conference is a voluntary association of worldwide organizations that have responsibility for the prediction of hydrodynamic performance of ships and marine installations based on the results of physical and numerical modeling.

Ship model towing tank resistance tests follow the ITTC procedure 7.5-02-02-01, “Resistance Test” (2002b) is aim to measure the total resistance of a ship model sailing in calm water with a constant speed, and by using empirical formula to calculate the theoretical value of friction resistance, to obtain the relationship between residuary resistance coefficient C_R and Froude number Fr of a ship model. This is a method tries try to use the measurements of model scale ship gets from the towing test experiment to estimate the full scale ship’s resistance. The details method we will describe later. [6]

ITTC towing test is the theoretical foundation of our master thesis, also is the source of the empirical data that we use to compare.

2.3 Nominal Wake

2.3.1 What is nominal wake?

Nominal wake is ship velocities deduction happened in the place that behind the ship hull and in front of the propeller. The flow around the ship and in front of propeller are affected by the presence of ship hull, the potential and viscous nature of the boundary layer around the ship contribute to the development of the wake, and the result is: the advanced speed of the water through the propeller plane is usually less

than the ship speed. And the mean velocity in the place of a propeller of a ship without any acting propeller is the nominal wake ω_N :

$$\omega_N = \frac{V - V_N}{V} \quad (2.1)$$

V is the ship sailing speed, V_N is the average nominal propeller advance speed, or we say the mean value of the water flow velocity in front of propeller.

The wake is generated by several reasons, consists of: frictional wake component, caused by viscous effect between water and ship; potential wake and wave making wake. And the total wake equals the sum of these three components. [7]

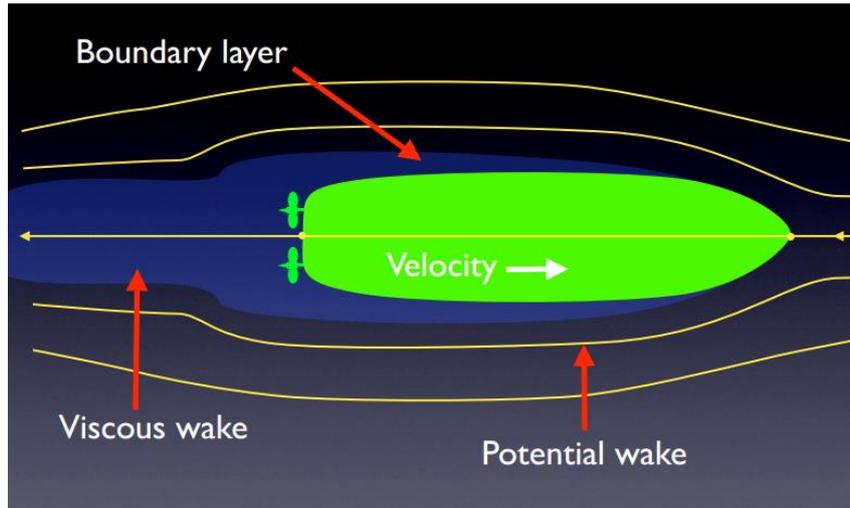


Figure2. 2. The causes of wake

(Resistance & Propulsion MAR 2010, Presentaion of ships wake, Rod Sampson – School of Marine Science and Technology)

For ship designer, a larger wake value is what we wanted because wake is helpful for the ship propulsion efficiency. The ship hull efficiency η_H can be expressed by equation:

$$\eta_H = \frac{1-t}{1-\omega} \quad (2.2)$$

t is thrust deduction coefficient, ω is wake fraction coefficient. From the equation it shows that a larger wake can give ship a better hull efficiency.

In addition, the knowledge of velocity distribution in the wake can give us the chance to determine the friction resistance component of the ship and distinguish the different resistance components, also determine the resistance coefficient between the ship hull and the water going through the propeller disc area.

2.3.2 How to measure wake

Measure region

As shown in figure 2.3 and 2.4, the region we mainly focus on is a circle panel that locate on the propeller plane at the aftship.

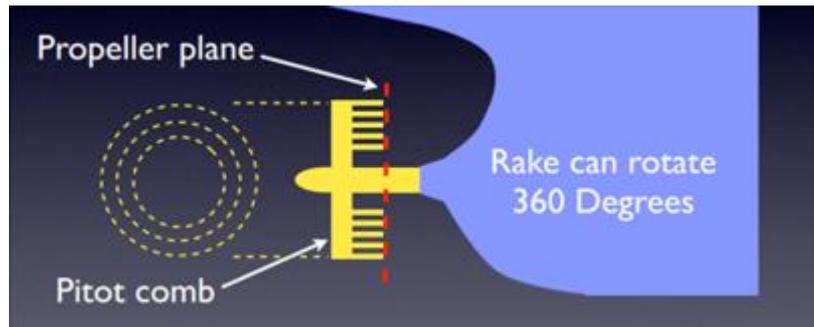


Figure2. 3. Measurement region in model towing test

(Resistance & Propulsion MAR 2010, Presentaion of ships wake, Rod Sampson – School of Marine Science and Technology)

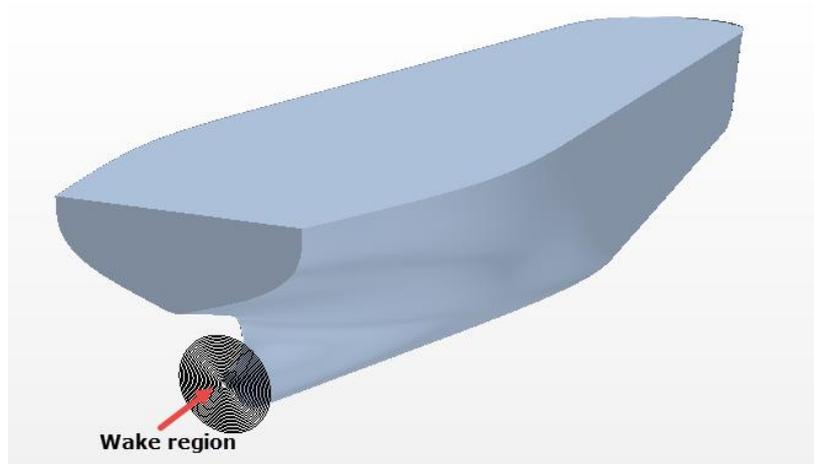


Figure2. 4. Measurement region in Star CCM+

Figure 2.3 shows what we do to measure the wake in real model towing test experiment with the equipment pitot comb. And figure 2.4 shows the region we measure in our Star CCM+ simulations.

2.4 Scale Effect

During the traditional general ship design process, naval architects always do the towing tests for insight into ship hydrodynamics: measure each ship resistance components and water velocities around the ship. Because the experiment size limits, the model always much smaller than the full scale ship, in order to ensure the results credibility, naval architects need to keep all the special care of environment equal to the real full scale environment, what we know are the Froude number Fr and Reynolds number Re .

The scale effect comes from the scale of the size difference between the model scale and full scale ship. When naval architects perform a laboratory model test, what we want is all the environment variables are the same, these variables contain: Froude number Fr and Reynolds number Re :

$$Re_m = Re_s \quad Fr_m = Fr_s$$

But in fact, we can't keep this two coefficient equivalent at the same time, from figure 2.5, we can find the reason:

<u>Reynolds number identity</u>	<u>Froude number identity</u>
$\frac{V_{0M} \cdot L_M}{\nu_M} = \frac{V_{0S} \cdot L_S}{\nu_S}$ $V_M \approx V_S$ $V_{0M} = V_{0S} \cdot \frac{L_S}{L_M} = V_{0S} \cdot M$	$\frac{V_{0M}}{\sqrt{g \cdot L_M}} = \frac{V_{0S}}{\sqrt{g \cdot L_S}}$ $V_{0M} = \frac{V_{0S}}{\sqrt{M}}$

Figure2. 5. Equations of simulation environments

Where V_{0M} is model scale ship speed, V_{0S} is full scale ship speed, $M = L_S/L_M$. In fact, M always is quite big number, if we use M multiple by ship speed V_{0S} , V_{0M} will be very big that we can't let ship model in water sailing so fast. So only Froude number identity can practically be met in. This is the origin of the scale effect. And because the Re number is different, ship friction resistance coefficient will be different, too.

3 METHODS

In this chapter we explain how you planned to solve the task including relevant procedures. And some equations and solve problem method involved in this master thesis.

3.1 Main stages of CFD simulation

The CFD simulation process contains the three main stages [1]:

1. Pre-processing:

Pre-processor enables the input of the data and setup of the flow simulation problem through a user-friendly interface and transformation of those into the format read by the solver. It can be a separate program offering export possibility to the solver(s), or it can be integrated with the solver under the same Graphical User Interface (GUI). The following activities by the user are usually associated with the pre-processing stage:

- *Preparation of geometry to be modeled;*
- *Definition and sub-division of computation domain;*
- *Choice of the mesh model and mesh generation;*
- *Definition of fluid properties;*
- *Selection and setup of the adequate solution models;*
- *Specification of appropriate boundary conditions*

2. Solving

There are four distinct streams of numerical solution techniques: finite difference methods, finite element methods, spectral methods and finite volume methods. All numerical methods that are used for solving the governing flow equations shall perform the following steps:

- *Approximation of the unknown variables by means of simple functions;*
- *Discretization of governing flow equations by substitution of these approximations and subsequent reduction to a system of algebraic equations;*
- *Solution of the algebraic equations.*

The main differences between the solve techniques named above are associated with the ways in which the flow variables are approximated and the discretization is done.

2. Post-processing:

Post-processing in CFD serves the purposes of facilitation of solution setup, execution control and interpretation of simulation results.

3.2 The fundamental laws of fluid

In this part we presents fundamental laws of governing fluid motion step by step. The CFD program, such as Star CCM+ we use in this master thesis, calculate the simulation by using these fundamental laws of fluid.

3.2.1 Forces acting in the fluid

Before analysis the motion of fluid element, we need to know the force acting in the fluid particle. The molecular structure of fluids does not create resistance to external forces. All the fluid media obey the same laws of motion, both liquid and gas. In most cases, a fluid is regarded as continuous medium. And

because if forces are applied at one point of fluid may cause fluid discontinuity, not similar to the rigid body mechanics consider about two types forces (at one point and distributed force), when in fluid mechanics, we are mainly concerned with distributed forces. [1]

One type of forces are acting on the surface, which are named “*surface forces*”, these forces are acting on the surface S surrounding the fluid volume. In most of cases, an elemental surface force $\Delta\vec{F}_S$ is applied at an arbitrary angle to the surface element ΔS whose external normal is \vec{n} . This force causes a certain stress on the surface element:

$$\vec{P}_V = \lim_{\Delta S \rightarrow 0} \frac{\Delta\vec{F}_S}{\Delta S} \quad (3.1)$$

The dimension of surface stress is [Pa], (Pressure). We divided the surface stress into two parts in *pressure forces* and *viscous forces* in category, and accordingly, the state of stress of a fluid element is defined in terms of pressure and nine viscous stress components (Sedov,1971).

Surface stress also can be distinguish to normal stresses and tangential stresses. In a hexahedral control volume (cell) of a computation mesh used in CFD simulations. The normal stresses include pressure p and viscous stresses τ_{xx} , τ_{yy} and τ_{zz} . The tangential stresses include viscous stresses τ_{xy} , τ_{xz} , τ_{yx} , τ_{yz} , τ_{zx} and τ_{zy} . And finally we get 9 unknowns in one fluid particle, shown in figure 3.1.

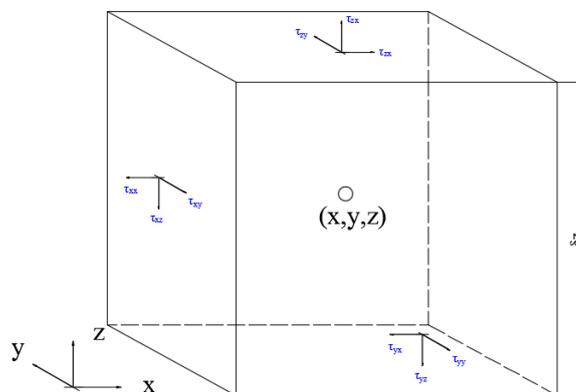


Figure3. 1. Fluid particle

The forces that act on every particle of the considered fluid volume and proportional to the particle mass are named “*mass forces*”. The corresponding stress is expressed as follows:

$$\vec{P}_V = \lim_{\Delta V \rightarrow 0} \frac{\Delta\vec{F}_V}{\rho \cdot \Delta V} \quad (3.2)$$

Where ρ is the fluid density. The dimension of mass stress is [m/s^2], (Acceleration). The mass forces include gravitation forces, inertia forces and electromagnetic forces. In the derivation of the governing flow equations it is common to present the contributions due to surface forces (pressure and viscous) as separate terms and to include the effects of body forces as additional source terms

3.2.2 Basic laws of physics

In order to analysis the motion of a fluid element, we need to derive the equation of fluid motion. And *the governing equations* of fluid motion represent mathematical formulation of the basic laws of physics. These laws contain:

1. The mass of fluid is conserved (continuity equation)

The laws states that the fluid mass is conserved. Or we can say rate of increase of mass in fluid element equals to the net rate of mass flow into the fluid element. The equation is:

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \vec{U}) = 0 \quad (3.3)$$

Where ρ is fluid density; $\vec{U} = (u, v, w)$ which is the fluid velocities in three directions.

2. Momentum conservation law: (Newton's second law)

This laws stats that the rate of change of momentum is equal to the sum of forces acting on a fluid particle. The equations, which are follow the Newton's second law, also well known as the "*Navier-Stokes equation*":

$$\begin{aligned} \frac{\partial(\rho u)}{\partial t} + \nabla(\rho u \vec{U}) &= -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho F_x \\ \frac{\partial(\rho v)}{\partial t} + \nabla(\rho v \vec{U}) &= -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho F_y \\ \frac{\partial(\rho w)}{\partial t} + \nabla(\rho w \vec{U}) &= -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho F_z \end{aligned} \quad (3.4)$$

Where p is the surface pressure; τ_{nm} is viscous stresses and $\vec{F} = (F_x, F_y, F_z)$ is the body force (mass force).

3. Energy conservation

Energy conservation laws states that the rate of change of energy equals the sum of the rate of heat addition to and rate of work done on a fluid particle, which is known as first law of thermodynamics. The equation is:

$$\begin{aligned} \frac{\partial}{\partial t} \left(\rho \left(e + \frac{\vec{U}^2}{2} \right) \right) + \nabla \left(\rho \vec{U} \left(e + \frac{\vec{U}^2}{2} \right) \right) = \\ \rho \cdot \dot{q} + \frac{\partial}{\partial x} \left(k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(k \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left(k \frac{\partial T}{\partial z} \right) - \frac{\partial(u\rho)}{\partial x} - \frac{\partial(v\rho)}{\partial y} - \frac{\partial(w\rho)}{\partial z} + \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \\ \frac{\partial(u\tau_{xy})}{\partial x} + \frac{\partial(u\tau_{yy})}{\partial y} + \frac{\partial(u\tau_{zy})}{\partial z} + \frac{\partial(u\tau_{xz})}{\partial x} + \frac{\partial(u\tau_{yz})}{\partial y} + \frac{\partial(u\tau_{zz})}{\partial z} + \rho \vec{F} \cdot \vec{U} \end{aligned} \quad (3.5)$$

Where T is temperature; e is internal energy per unit mass; k is thermal conductivity and \dot{q} is rate of volumetric heat addition per unit mass;

If we combine equation 3.3, 3.4 and 3.5, we can get 7 unknown values but in 5 equations: three velocities u, v, w , pressure p , temperature T , density ρ and internal energy i . It obvious that these equations can't be solve. However, for most of the cases we mainly concerned with CFD, the fluid is incompressible and has a constant density, so that the continuity and momentum equations are uncoupled from the energy equation and they are enough for the calculation of pressure and velocity field. The energy equation is only engaged if the problem studied involves heat transfer or concern with temperature.

3.2.3 Simplifications of equations (Assumptions of fluid)

Because we can't use 7 unknowns and 5 equations to solve the equations, in order to find the way to solve the ship dynamics problems, the governing equations need to be simplified.

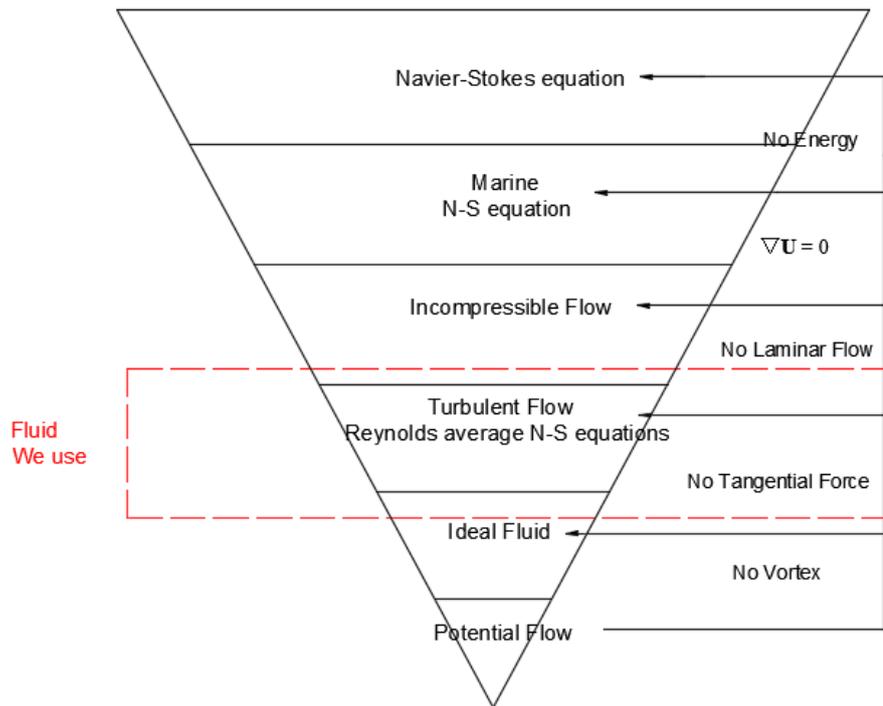


Figure3. 2. Simplification of fluid model

The simplification steps shown in figure 3.2. From top to bottom, the area of equations concerned is smaller:

1. **Marine N-S equation:** For marine CFD problems, temperature and energy are usually not taken into account, it means that in this CFD case. Energy conservation equation is not used.
2. **The Assumption of Incompressibility:** For incompressible flow such as we require for hydrodynamics, and assuming that the fluid is Newtonian and that the viscosity is constant throughout the flow, the continuity equation becomes:

$$\nabla \vec{U} = 0 \quad (3.6)$$

Combine equation 3.4 and 3.6, can get the new simplified equations:

$$\begin{aligned} \rho \frac{Du}{Dt} &= -\frac{\partial p}{\partial x} + \mu \nabla^2 u + \rho F_x \\ \rho \frac{Dv}{Dt} &= -\frac{\partial p}{\partial x} + \mu \nabla^2 v + \rho F_x \\ \rho \frac{Dw}{Dt} &= -\frac{\partial p}{\partial x} + \mu \nabla^2 w + \rho F_x \end{aligned} \quad (3.7)$$

Where substantial derivative $\frac{D}{Dt} = \frac{\partial}{\partial t} + u \frac{\partial}{\partial x} + v \frac{\partial}{\partial y} + w \frac{\partial}{\partial z}$; kinematic and dynamic viscosity coefficients $\nu = \frac{\mu}{\rho}$.

3. **Phenomenon of turbulence:** When Reynolds number $Re < 2300$, the separate layers of fluid flow still do not mix, we call this flow laminar flow. And when Reynolds number $Re > 4000$, the flow will be the chaotic vertical flow, which we called turbulent flow. In general, both these two types flow exist in flow, but in order to simplify the equations, we assume that all the fluid flow are 100% turbulent flow (In fact, 95% flow is turbulent flow in most cases), the components of flow velocity and pressure are represented as superposition of their mean values and imposed turbulent fluctuations. This is discovered by Reynolds in 1883, and the equations are well known as “*Reynolds – average Navier Stokes Equations (RANS)*”:

$$U = \bar{U} + U' \quad (3.8)$$

Where U' is named turbulent fluctuations; \bar{U} is the mean value of the velocities. This mean value can be used in N-S equations directly.

Because \bar{U} is a constant value, but fluctuated value is not constant, hence when combined equation 3.10 with 3.5 and 3.9, the equations become (x-component):

$$\rho \cdot \left[\frac{\partial \bar{U}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{v}}{\partial y} + \bar{w} \frac{\partial \bar{w}}{\partial z} \right] = - \frac{\partial \bar{p}}{\partial x} + \frac{\partial}{\partial x} \left[\mu \frac{\partial \bar{u}}{\partial x} - \rho \overline{u'u'} \right] + \frac{\partial}{\partial y} \left[\mu \frac{\partial \bar{u}}{\partial y} - \rho \overline{u'v'} \right] + \frac{\partial}{\partial z} \left[\mu \frac{\partial \bar{u}}{\partial z} - \rho \overline{u'w'} \right] \quad (3.9)$$

From the equation 3.9, we find that we get 6 more additional unknowns, plus 4 unknowns from previous, in total we get 10 unknowns at this step, but there are only 4 equations now. Reynolds stresses ($\rho \overline{u'u'}$, etc.) are consider as extra stresses that arise from the turbulent nature of the flow. And in order to solve these equations, CFD program can choose several method to solve it. The turbulence model method which we will mention after in case study part will explain how the model solves these unknowns.

3.3 Discretization Method

Discretization is a technique of conservation of general scalar transport equation into an algebraic equation that can be solved numerically. This control volume technique can be split into two parts:

1. Integrating the transport equation about each control volume.
2. Writing a discrete analog of equation.

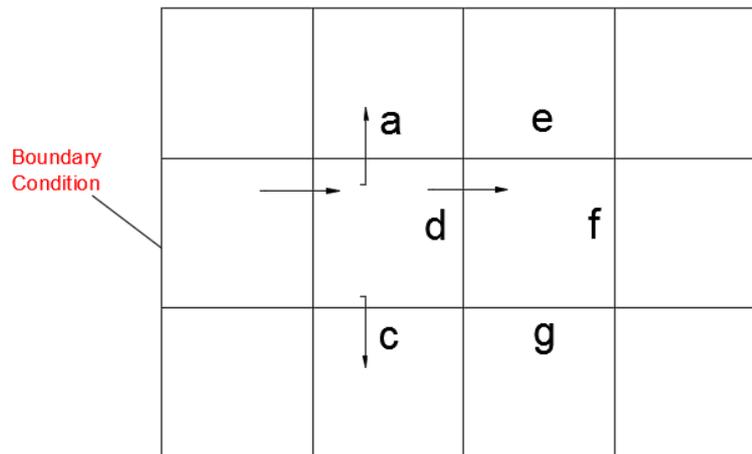


Figure3. 3. Discrete method

This analog is an algebraic form of the transport equation which use to transfer information from one mesh cell to another. In general, for hexahedral cell we use in this master thesis, each cell has six neighbors. But on the sides of the computational domain, every cell will have only five neighbors, not six. So information of one face will be missed. In order to close up the system of equations. We have to provide this face with information, this information named boundary conditions.

3.4 Ship simulation in Star CCM+

STAR-CCM+ is a new generation of CFD solver software, which is developed by CD-adapco company by using the most advanced “CCM” (computational continuum mechanics) algorithms. It is equipped with CD-adapco latest original mesh generation. STAR-CCM+ has a comprehensive suite of post-

processing tools designed to enable you to obtain maximum value and understanding from our CFD simulation. This includes scalar and vector scenes, streamlines, scene animation, numerical reporting, data plotting, import, and export of table data, and spectral analysis of acoustical data. [8]

Then how to simulate the situation that ship is sailing in the water in Star CCM+? In ITTC towing test, naval architects towing the ship in the deep-water test pool. But in this master thesis, we use another to solve the problem: build a huge region to simulate the environment which contains both water and air, then we fix the ship in a reverse flow of the fluid flow without winds in the region, waves and currents. Shown in figure 3.4.

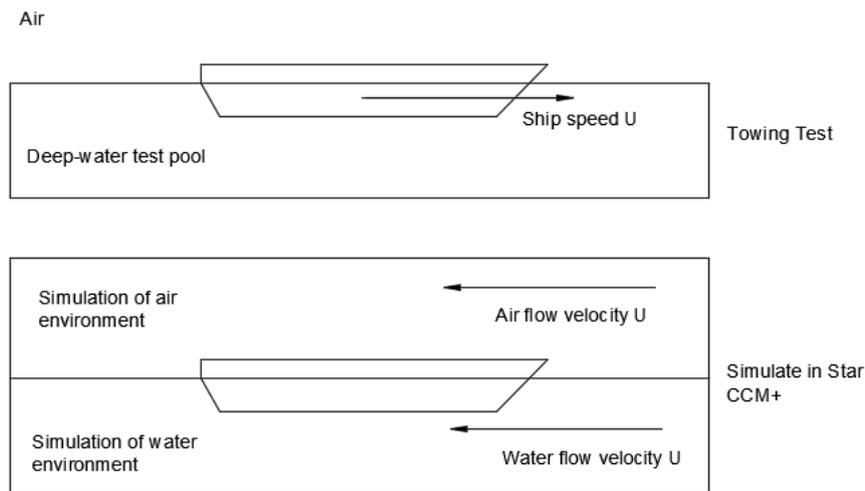


Figure3. 4. Simulation method in Star CCM+

Should be noted that this method is only use to the simulation that ship is moving in a constant velocity. If ship has an acceleration, there will be added mass and that will be different between these two method.

STAR-CCM+ has a comprehensive suite of post-processing tools designed to enable you to obtain maximum value and understanding from your CFD simulation. This includes scalar and vector scenes, streamlines, scene animation, numerical reporting, data plotting, import, and export of table data, and spectral analysis of acoustical data.

4 CASE STUDY

This chapter describes the models and cases what we want simulate in this master thesis, shows the setup parameters before simulation and explain the reasons that why we want to use these parameters.

4.1 Model Description

Japan Bulk Carrier (JBC) Model

JBC (Japan Bulk Carrier) is a capesize bulk carrier equipped with a stern duct as an energy saving device. This type of bulk carrier designed by the National Maritime Research Institute (NMRI), Yokohama National University and Ship Building Research Centre of Japan (SRC), contain a ship hull, a duct and a rudder. The shape of JBC is a traditional large merchant ship, has a bulbous bow and a single screw stern.

Towing tank experiments are planned at NMRI, SRC and Osaka University, which include resistance tests, self-propulsion tests and PIV measurements of stern flow fields. The hull design and measurements were conducted with the support of Class NK as part of the Class NK joint R&D for Industry Program. So far, there is not any full scale ship exists.

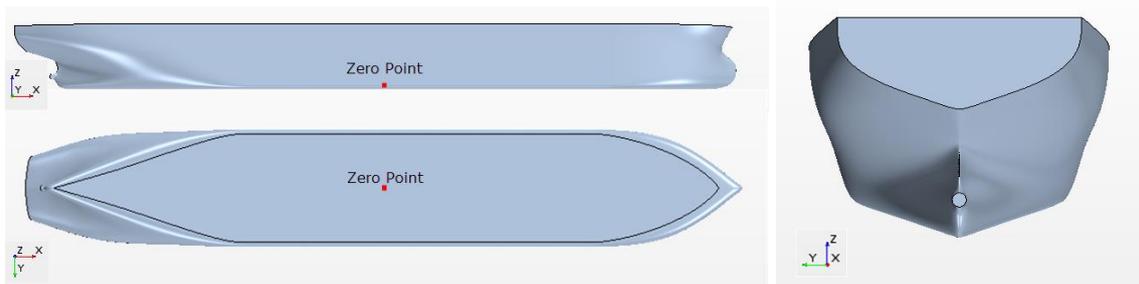


Figure4. 1. The JBC Model

In this CFD simulation, we simulate the condition without the ship appendix (Propeller, duct and rudder). Just calculate the influenced by ship hull in both model scale and full scale ship. A geometry of model scale ship is provided by “Tokyo 2015 - A Workshop on CFD in Ship Hydrodynamics”. This is a IGES file format which define the surface of JBC ship model. Before setting up the main simulation, it is recommended to perform surface “remeshing” step in Star CCM+, in order to fix possible hull surface flaws and improve overall surface quality.

The ship hull model contain five parts: Deck, hull, hull bottom, shaft_tube_end and transom. The model and full scale ship data shown in table 4.1:

		Full scale	Model scale
Proportion		1	40
Length between perpendiculars	LPP (m)	280	7
Length of waterline	LWL (m)	285	7.125
Maximum beam of waterline	BWL (m)	45	1.125
Depth	D (m)	25	0.625
Draft	T (m)	16.5	0.4125
Displacement volume	∇ (m ³)	178369.9	2.787029688

Wetted surface area w/o ESD	SW (m ²)	19556.1	12.2225625
Wetted surface area of ESD	SE (m ²)	745.2	0.46575
Block coefficient (CB)	$\nabla/(LPP*BWL*T)$	0.858	0.858
Midship section coefficient (CM)		0.9981	0.9981
LCB (%LPP), fwd+		-2.5475	-2.5475
Vertical Center of Gravity (from keel)	KG (m)	NA	
Metacentric height	GM (m)	NA	
Moment of Inertia	K _{xx} /B	NA	
Moment of Inertia	K _{yy} /LPP, K _{zz} /LPP	NA	
Design Speed	V _{design} (knots)	14.5	14.5
Design Speed	V _{design} (m/s)	7.4588	1.17934

Table4. 1. The dimensions of JBC ship

Ship Position	Full scale (m)	Model Scale (m)
Positive x-direction (Bow)	145.96	3.649
Negative x-direction (Stern)	-145	-3.625
Positive y-direction (Portside)	22.5	0.5625
Negative y-direction (Starboard)	-22.5	-0.5625
Positive z-direction (Ship deck)	25.22	0.6305
Negative z-direction (Ship bottom)	0	0
Design Draught	16.5	0.4125

Table4. 2. Position of ship

We also define the ship coordinate in Star CCM+'s coordinate system for both model and full scale.

4.2 Set up parameters

4.2.1 Main stage

Before we start to calculate the cases, we need to setup all the parameters that necessary for the simulation. The main stage consists of:

1. Geometry
 - 1.1 Import geometry
 - 1.2 Create domain
 - 1.3 Region appointment
2. Mesh
 - 2.1 Specify mesh type
 - 2.2 Specify mesh size
 - 2.3 Build volumetric control
 - 2.4 Automatic mesh generate
3. Physics
 - 3.1 Fluid condition

3.2 Equations determined: Reynolds – average Navier-Stokes equations (RANS); Unsteady flow; Turbulent model.

3.3 Set boundary condition of simulation region

3.4 Give initial information to mesh cells

4. Run simulation

4.2.2 Domain Description

The range of domain

Ship viscous-flow computations typically have two fixed boundaries: the ship surface and the (still) water surface, in some cases, there is always a symmetry plane. And these surface plane surrounds a closed region around the ship that use to simulate the CFD problem, which we call it “Domain”.

Domain is enclosed by the boundaries. In general boundary conditions include inlet, outlet and exterior boundary, where approximate boundary conditions have to be defined.

It is noteworthy that these boundaries plane of the domain has to be placed sufficiently far from the ship to minimize the effect of the location of these boundaries on the solution. For the inlet and outlet boundary either the uniform (undisturbed) flow is usually imposed, and in that case these boundaries should be located 1-2 L_{PP} away from the hull. The figure 4.2 show how much number what we set in full scale simulation.

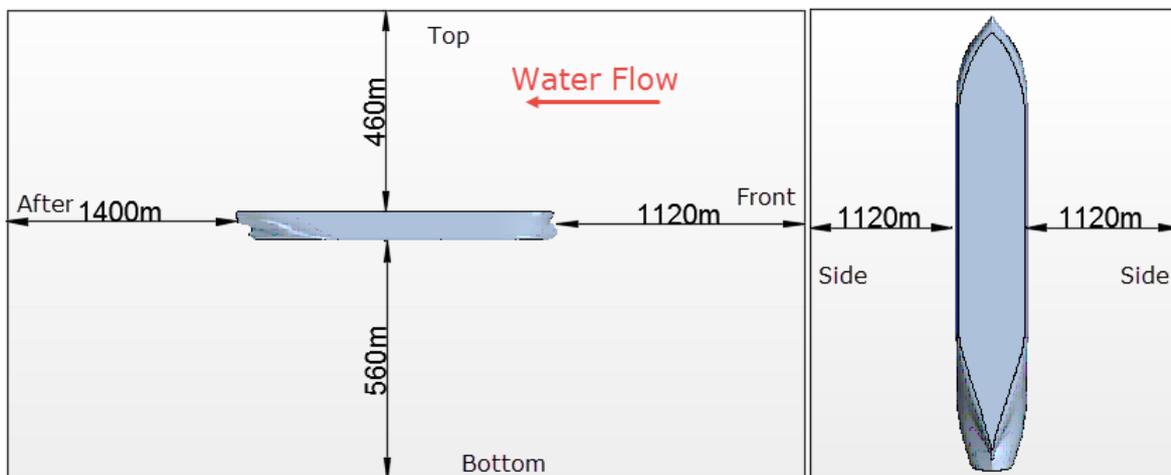


Figure4. 2. Domain of simulation

Boundary	Full scale	Model Scale	Multiple(of L_{pp})
Front (In front of ship)	1120m	28m	4
After (After ship)	1400m	35m	5
Side (Portside)	1120m	28m	4
Side (Starboard)	1120m	28m	4
Top (From ship deck)	460m	11.5m	1.643
Bottom (From ship bottom)	560m	14m	2
			$L_{pp} = 280m$

Table4. 3. Domain of model scale and full scale

Table 4.3 presents the domain size of both the model scale and full scale. In order to minimize the effect from boundary, the length around ship domain is at least 4 times than the L_{PP} ; The length after ship is 5 times because we want to observe the generate wave after ship. The smallest length is the distance between ship deck and the top boundary: just 1.642 times, this is because the effect of air is very small compare to the water, in order to save calculation time, we reduce the distance of this.

Half Domain

CFD simulation process is quite a big work for computer. In order to guarantee the accuracy of results, we also need to build huge amounts of mesh cells. Because of this, the calculation always takes a lots of time.

JBC is a symmetrical single stern ship. Theoretically, for towing simulations, we can try use half of the domain (and ship) with symmetry plane along the CP plane (XOZ). If the results is similar with the whole ship simulation, using half model can save half of calculation time.

The half ship model is cut directly from the origin model: All of the domain parts from middle plane of the ship are replaced by a symmetry plane. Shown in figure 4.3.

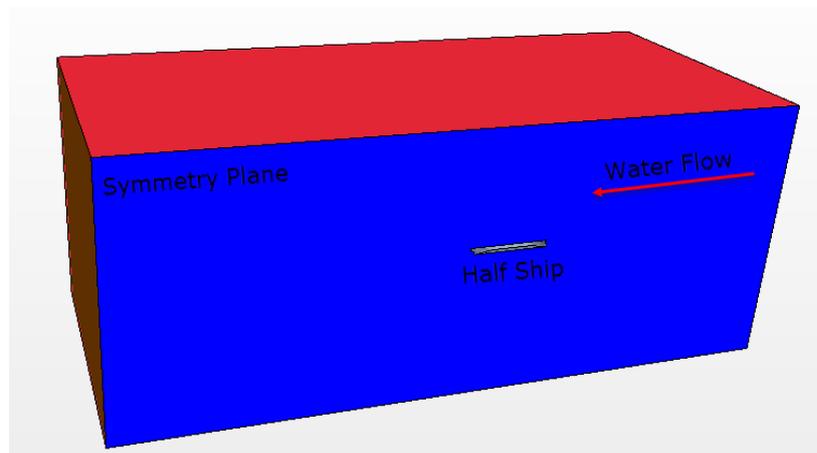


Figure4. 3. Half domain

4.2.3 Generate Mesh

4.2.3.1 Mesh Selected

Computation mesh is a discrete geometrical representation of computation domain where the flow simulation problem is solved. Computation domain has to be restricted by boundaries of one type or another depending on problem setup and on functions to be performed by these boundaries in the solution. Many (while not all) meshing methods begin the process of mesh generation from building a surface mesh on the aforementioned domain boundaries. Surface mesh consists of two-dimensional planar or curvilinear elements called faces. A face is comprised of vertices and edges. From the surface mesh a volume mesh is built consisting of three-dimensional elements called cells. A cell is comprised of vertices, edges and faces. [1]

As well as faces, mesh cells should also meet a number of criteria that are designed to ensure appropriate mesh quality. Such criteria and their mathematical definitions may vary somewhat from code to code, and not all of them are applicable to all cell types. There are several types of mesh cell: tetrahedron,

hexahedron, pyramid and prism/wedge cells. Most usually use mesh type is hexahedron, which is shown in figure 4.4

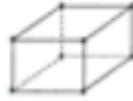


Figure4. 4. Hexahedron mesh cell

In these CFD simulation, we use Automatic Meshing Technology to build the mesh cells. This is STAR-CCM+'s single integrated process provides the fastest, most automatic route from complex CAD to CFD mesh. Advanced automatic meshing technology generates either polyhedral or predominantly hexahedral control volumes at the touch of a button, offering a combination of speed, control, and accuracy. For problems involving multiple frames of reference, fluid-structure interaction and conjugate heat transfer, STAR-CCM+ can automatically create conformal meshes across multiple physical domains.

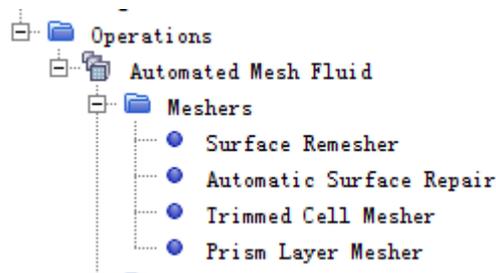


Figure4. 5. Automated mesh technology

For the region that need to consider about the huge turbulent, such the region around the ship hull and free surface region. We build some volumetric control that use smaller size of mesh cells to ensure the accuracy. Shown in figure5.6.

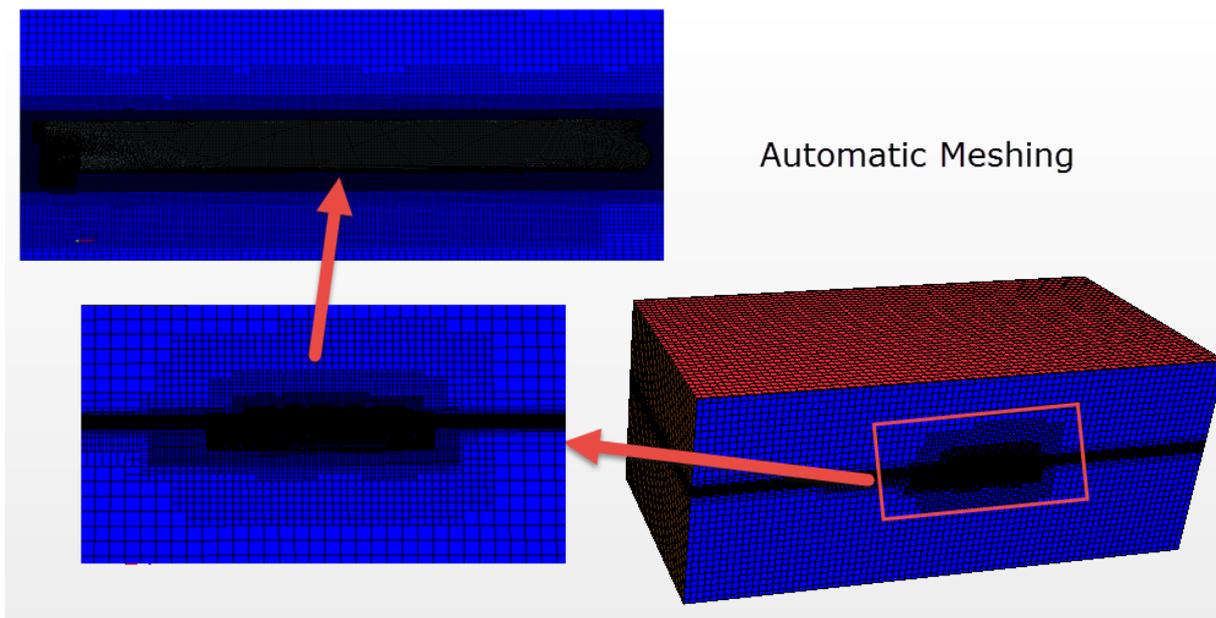


Figure4. 6. Mesh of domain

Near-wall treatment

An important part of mesh generation for accurate CFD simulation is the near-wall region, or extrusion-layer mesh. STAR-CCM+ automatically produces a high-quality extrusion layer mesh on all walls in

the domain. In addition, we can control the position, size, growth-rate, and number of cell layers in the extrusion-layer mesh.

We know that, turbulent flows are significantly affected by the presence of walls. When flow is very close to the wall, viscous damping reduces the tangential velocity fluctuations. Toward the outer part of the near-wall region, however, the turbulence is rapidly augmented by the production of turbulence kinetic energy due to the large gradients in mean velocity. It means, when the flow close to the wall (ship hull surface), the local Reynolds number will be very low and there will be the laminar flow. But in fact, in CFD simulation, we assume that the flow is 100% percentage turbulence.

Near wall treatment is a method that the turbulence models are modified to enable the viscosity affected region to be resolved with a mesh all the way to the wall, including the viscous sub-layer. Turbulence models ought to valid throughout the near-wall region. Shown in figure 4.7. This approach is capable to resolve the physical of the flow directly without any empirical models. So it is very physical and accurately. But the disadvantage is it will generate more mesh cells, and cost a lot of time to calculate.

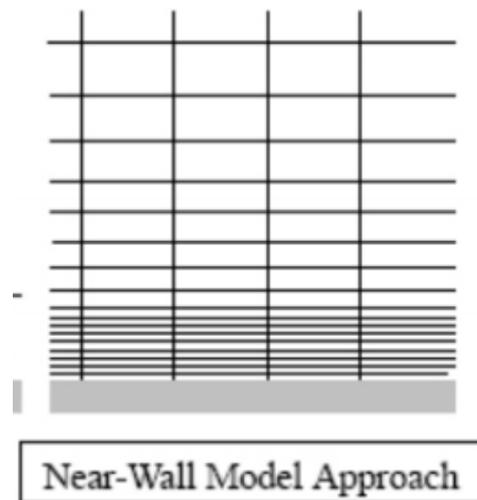


Figure4. 7. Near-wall prism mesh around ship surface

Y^+ number

After near-wall treatment, we need to know what is Y^+ . Y^+ is a special value that can give us indication that our mesh around the ship is good enough. Y^+ is defined as Reynolds number Re , the equation is:

$$Y^+ = \frac{Vt}{\gamma}$$

Where velocity is the local velocity at the cell; and t is the distance from control point of the cell to the ship surface. Shown in figure 4.8:

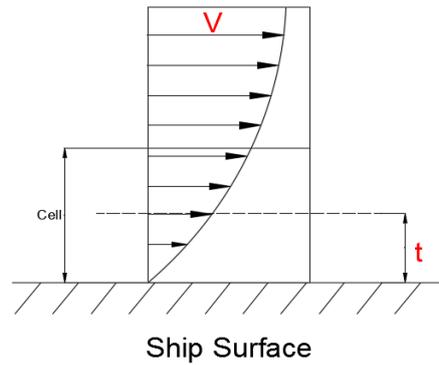
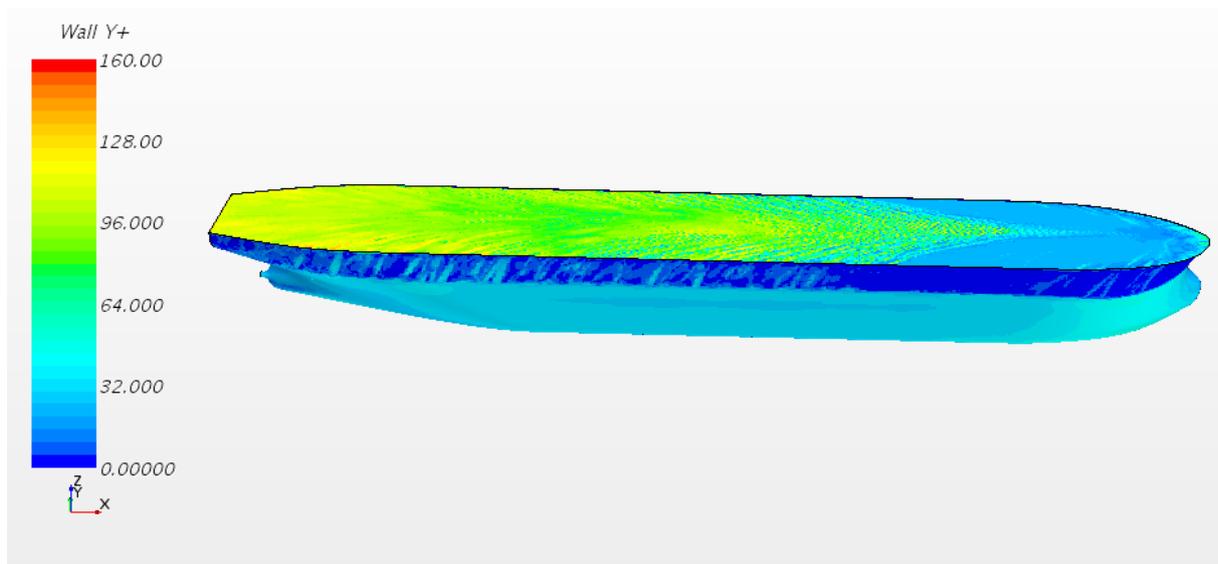


Figure4. 8. Y+

In general, the Y+ number should keep in range 30-500 and so that can keep the result high accuracy. In this thesis, we use the all y+ function in Star CCM+, which contains both high and low y+ conditions. The range we want keep is wall functions: $30 < y+ < 100$ (target 40-50) on ship hull; Figure 4.9 shows the Y+ number in the condition “Full model scale ship in $Fr = 0.1423$ (Design velocity)”

Figure4. 9. Wall Y+ number in Model scale ($Fr = 0.1423$)

Prism Layers:

“Prism Layers” is the way that reduce the Y+ number around ship. They are small thickness layers of mesh. The total thickness equation is:

$$h_w = 2y = \frac{y^+}{LRe \sqrt{\frac{C_f}{2}}} \quad (4.1)$$

At full scale Re numbers, boundary layer becomes thinner and near-wall velocity profiles become fuller, which needs more cells to resolve them accurately.

Mesh cells

Figure 4.4 shows the auto-generated meshes in the simulation region. In the place that need higher accurate such ship hull and free surface, we use volumetric control method to build more details meshes. In table 2, the mesh settings for all simulations are exhibited.

	Ship status	Fr	Mesh basic size	Number of prism layers	Thickness of prism layers	Mesh cells number
Model scale	Half	0.0785	0.203 m	5	0.045675 m	2794083
	Half	0.0981	0.203 m	5	0.03045 m	2949867
	Half	0.1178	0.203 m	5	0.015225 m	3002787
	Half	0.1374	0.203 m	5	0.015225 m	3002787
	Half	0.1423	0.203 m	5	0.015225 m	3002787
	Full	0.1423	0.203 m	5	0.015225 m	5881132
	Half	0.1472	0.203 m	5	0.015225 m	3002787
	Half	0.1669	0.203 m	5	0.015225 m	3002787
Full scale	Full	0.0785	8.12 m	10	0.19082 m	6495589
	Full	0.0981	8.12 m	10	0.19082 m	6495589
	Full	0.1178	8.12 m	10	0.19082 m	6495589
	Full	0.1374	8.12 m	10	0.19082 m	6495589
	Half	0.1423	8.12 m	10	0.19082 m	6495589
	Full	0.1423	8.12 m	10	0.19082 m	6495589
	Full	0.1472	8.12 m	10	0.19082 m	6495589
	Full	0.1669	8.12 m	10	0.19082 m	6495589

Table4. 4. The mesh setting for model and full scale case

4.2.4 Physics setting

After set the model and the mesh, we need to set the physical parameters that use to define the simulation region. Figure 4.10 shows the example of physical settings we set in some of the simulation,

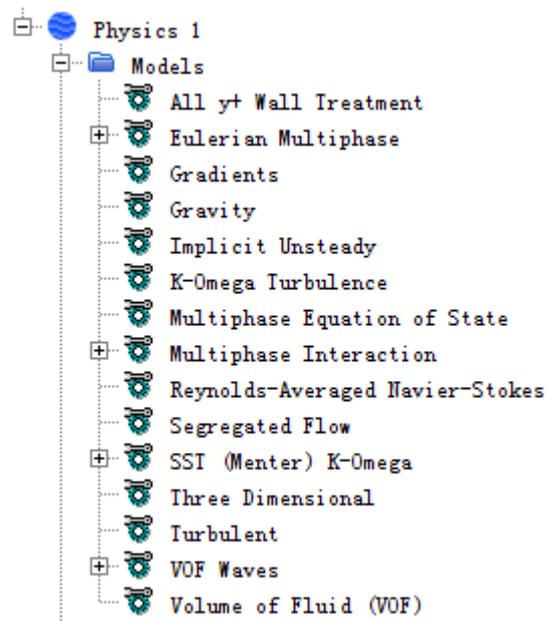


Figure4. 10. Physical settings

4.2.4.1 The Pressure-Based Segregated Algorithm

In CFD, the pressure-based solver we choose is a solution algorithm where the governing equations are solved sequentially. Because the governing equations are non-linear and coupled, the solution loop must be carried out iteratively in order to obtain a converged numerical solution.

In the segregated algorithm, the individual governing equations for the solution variables are solved step by step. Each governing equation, while being solved, is "decoupled" or "segregated" from other equations, hence its name. The segregated algorithm can save the computer memory, since the discretized equations need only be stored in the memory one at a time. However, the solution convergence is relatively slow, inasmuch as the equations are solved in a decoupled manner. Figure 4.11 shows the whole steps of this algorithm how to progress. [1]

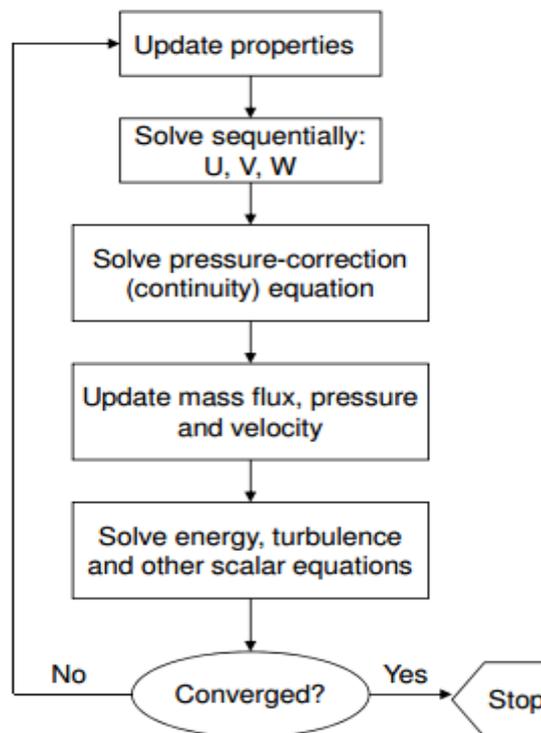


Figure 4. 11. The pressure-based segregated algorithm

The other way in Star CCM+ is pressure velocity coupling method, when use this method to calculate, the software usually solves transport equations not with the actual value pressure but with some estimated value. On the next stage, pressure value is corrected while pressure velocity coupling algorithm. We often use the segregated algorithm because we can only handle one equation at one time.

4.2.4.2 Eulerian Multiphase

The simulated environment is the standard freshwater conditions without any winds, waves or currents. Water and the air is still. In Star CCM+, we defined the environment parameters in Eulerian multiphase definition phase, some basic parameters are set below.

Water density, ρ_w (kg/m ³)	999.1
Air density ρ_A (kg/m ³)	1.205
Kinematic viscosity for water, ν (m ² /s)	0.000001107
Dynamic viscosity for water, μ (Pa-s)	0.001106
Kinematic viscosity for air, ν (m ² /s)	0.0000153949
Dynamic viscosity for air, μ (Pa-s)	0.0000185508
Acceleration of gravity (m/s ²)	9.81

Table4. 5. Basic values of the environment

4.2.4.3 Turbulent model

In order to simulate the turbulence, there are several method and models provided by CFD help us solving the turbulent problems. In this thesis, what we choose to use are the model: “*SST k – ω Turbulence Model*” and “*k – ϵ Turbulence Model*”

k – ϵ Turbulence model: This is the most common method that use to modelling the effect of turbulence. The assumption is that all liquid is isotropic. The calculation of added eddy viscosity from RANs can be approached in a number of ways, but the most commonly used method is that developed for the k – ϵ . The equations of 6 unknowns eddy viscosity μ_t and etc are transfer to the equation of k and ϵ [9]:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (3.12)$$

Where k is the turbulent kinetic energy per unit mass; and ϵ is the rate of dissipation of the turbulent kinetic energy per unit mass. C_μ is a constant number. Instead of 6 addition unknowns, k – ϵ turbulence model change them to the new unknowns k and ϵ . And at the same time, we also get two addition equations here: Transport equation for turbulent kinetic energy k and Transport equation for dissipation rate ϵ .

At this step, we get 6 unknowns: k, ϵ , surface pressure p and velocities in 3-directions. Also we have 6 equations: one continuity equation, 3-directions momentum conservation equations and equations for k and ϵ . CFD solvers can use segregated algorithm to solve these equations.

k – ω Turbulence model: An alternative two – equation turbulence model, which is use to the above is to attempt to calculate each of the 6 Reynolds stresses directly through the solution of further transport equations for each component. k – ω turbulence model is used as a closure for the Reynolds-averaged N-S equations (RANS equations). The model attempts to predict turbulence by two partial differential equations for two variables, k and ω , with the first variable being the turbulence kinetic energy (k) while the second (ω) is the specific rate of dissipation (of the turbulence kinetic energy k). [10]

SST k- ω Turbulence model: One of the main problems in turbulence modeling is the accurate prediction of flow separation from a smooth surface. Standard two-equation turbulence models often fail to predict the onset and the amount of flow separation under adverse pressure gradient conditions. SST k- ω turbulence model is a two-equation eddy-viscosity model. The meaning of SST is shear stress transport (SST) formulation. In the inner parts of the boundary layer, the model can directly usable all

the way down to the wall through the viscous sub-layer by using a $k-\omega$ formulation, hence the SST $k-\omega$ model can be used as a low-Re turbulence model without any extra damping functions. Compare to $k-\omega$ turbulence model, the SST formulation also switches to a $k-\epsilon$ behaviour in the free-stream and thereby avoids the common $k-\omega$ problem that the model is too sensitive to the inlet free-stream turbulence properties. [11]

These models have shown to be able to give accurate predictions in ship hydrodynamics, especially certain versions of the $k-\omega$ model and SST $k-\omega$ model and are by far the most applied ones (80% of the submissions for the Gothenburg 2010 Workshop). And Compare to the $k-\epsilon$ turbulence model, $k-\omega$ turbulence model has a more sensitive result and can get a more accuracy answers.

4.2.4.4 Interface Capturing

Turbulent is clearly the unsteady flow in the simulation. In this simulation, the ship is placed in a domain that has both air and water. Water has a free surface between air and water, at which such effects as phase change and surface tension can be neglected. But the boundary conditions of the two kinds – kinematic and dynamic – apply on this free surface. The way to define the free surface of the water in this thesis is “*Interface capturing method*”. These methods solve the transport equations on a pre-defined mesh which covers the whole domain including water and air. The position and shape of the free surface is given in computing the fraction of each cell located near the interface that is partially filled. There are several different approaches, what we use here is the VOF (Volume of Fluid) algorithm.

VOF (Volume of Fluid) algorithm

The VOF model is based on the assumption that the phases of a multi-phase flow do not mix. Therefore, in each cell the sum of the volume fractions of all phase equals to one. Considering a flow consisting of N_q phases and denoting the volume fraction of the q -th phase $\alpha_q = \frac{V_q}{V}$ one can than write:

$$\sum_{q=1}^{N_q} \alpha_q = 1$$

By capture the fraction of each cell can define the water surface position. VOF algorithm allow us to save a lot of computational time in the simulation, but not so accurate. This is the reason that we build a volumetric control in the near free surface and want the cells as small as possible in the free surface region, especially in the z -direction.

In star CCM+, we also define the fluid flow velocities in this step, the “VOF waves”, what we defined consist of the height of the free surface plane (16.5 m for full scale ship, known as the draught of the ship) and the velocities of water and air. This “VOF waves” will become some field functions that we can set them to boundary conditions and initial information directly later.

In addition, if we use VOF algorithm, there will generate one more equation and unknown for the volume fraction. So at last we get 7 equations.

4.2.4.5 Boundary condition (BC) and Initial Information

For the cells that near the boundary, they need some initial information: boundary conditions of region. Boundary conditions is a prescribed value is defined for the variable of interest at some known points on the boundary. From the point of view of physical properties addressed in the boundary conditions, it is customary to distinguish the following two kinds:

1. *Kinematic boundary conditions that deal with velocity;*
2. *Dynamic boundary conditions that deal with pressure.*

The boundaries that define and separate fluid zones can be of various types which depend on the problem setup and functions to be performed by the boundaries in the solution. The most common types are represented by walls, inlet boundaries, exit boundaries, symmetry boundaries, periodic boundaries and interface boundaries. In their turn, these types allow for different boundary conditions to be imposed.

Walls represent the impenetrable surfaces that bound the fluid.

Velocity inlet – used to define the velocity and scalar properties of the flow at inlet boundaries;

Pressure outlet – sets up a static pressure value at the exit boundary;

Symmetry boundaries allow one to take benefit of physical flow symmetry, in order to reduce the size of computation domain in the simulation and thus save memory and time. A solution obtained with a symmetry plane boundary is identical to the solution that would be obtained with a twice larger domain whose second half is a mirror image of the mesh in the first half about the symmetry plane.

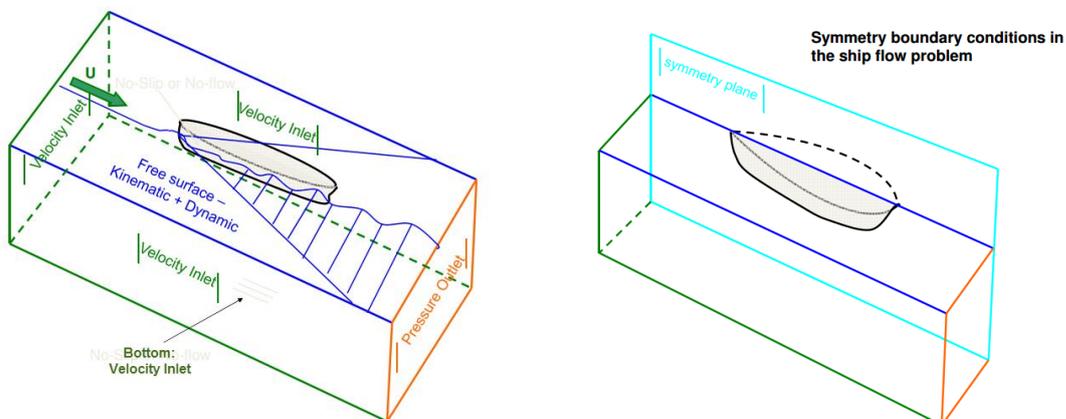


Figure4. 12. The boundary conditions of our region and half region

(Marine CFD for Engineering Applications Dr. Dmitriy Ponkratov, Senior Consultant, Technical Investigation Department, London, UK)

4.2.4.6 Initial Condition

Initial condition is the previous values that have to be set up in every cell of the mesh. Initial condition should be as close as possible to boundary conditions.

Because we have used the Eulerian multiphase to define the medium of the fluid, use VOF waves defined the velocity of fluid and the free surface. In this simulation, both boundary condition and initial conditions are defined by the “field function”, and use “VOF waves” as the field functions:

1. *Velocity of VOF waves for inlet boundary condition*
2. *Pressure of VOF waves for outlet boundary condition*

VOF Damping length

The VOF Waves model includes a VOF Wave damping capability. This means that a VOF wave can be damped in the vicinity of selected boundaries to reduce wave oscillation near those boundaries. Activating the damping introduces vertical resistance to the vertical motion. This setting is the distance from the VOF wave damping-enabled boundary at which the damping starts.

From the results we get from the later part, we find that the pressure resistance coefficient C_p is very sensitivity to wave damping parameters, In our full scale simulations with the JBC ship we used VOF Wave Damping Length = 520m and 13m for model scale. Which is roughly at the middle of the distance from ship Central Plane (CP) to the Side boundary.

4.2.4.7 Time step

Choice of the time step

In CFD, the Courant–Friedrichs–Lewy (CFL) condition is a necessary condition for convergence while solving certain partial differential equations (RANs) numerically by the method of finite differences. And time step of physical setting influence this CFL numbers. Time step means the real physical time for each calculation iteration step in CFD. As a consequence, the time step must be less than a certain time, otherwise the simulation will produce incorrect results. [12]

For one-dimensional case, the CFL has the following form:

$$C = \frac{u \Delta t}{\Delta x} \leq C_{\max} \quad (4.2)$$

Where the dimensionless number is called the Courant number, in this equation, u is the magnitude of the velocity; Δt is the time step; Δx is the length interval (whose dimension is length). The value of C_{\max} changes with the method used to solve the discretised equation, especially depending on whether the method is explicit or implicit. If an explicit (time-marching) solver is used then typically $C_{\max} = 1$. In our simulation, it means that for most region of our simulation, the CFL number must less than 1, or the results will not correct enough. Figure 4.13 is one example of our CFL number plots for case “Model scale _ Fr = 0.1423”.

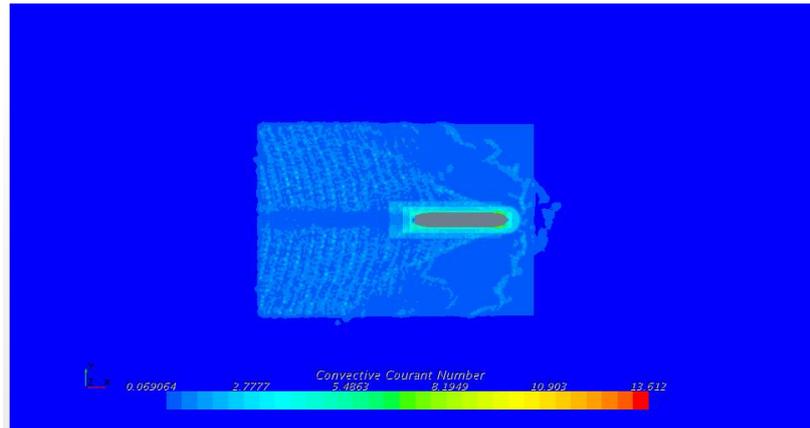


Figure4. 13. CFL number plot of simulation

In explicit solvers the time step is chosen to satisfy the CFL condition or to resolve the flow features of interest, whatever results smaller. Usually the CFL condition is more demanding than the flow requirements. In implicit solvers the time step is decided by the flow features. As a rule of thumb: For standard pseudo-transient resistance computations, use $\Delta t = 0.005 \sim 0.01 L/U$. The choice of time step will also depend on the complexity of the turbulence model. For Reynolds stress turbulence models it is more appropriate to use $\Delta t = 0.001 \sim 0.0025 L/U$. This also requires a larger number of iterations to obtain reasonable convergence. In more unstable problems, as those with low Froude number, a smaller time step may be needed. Notice that naturally transient problems will not reach a steady-state solution. For this research case, we use the

$$\Delta t \approx 0.005 \sim 0.01 \cdot \frac{L}{U}$$

By using the equation we try to calculate the time step of the simulation when ship sailing in fastest speed. For model scale, we use 0.05s. And for full scale we use 0.16s.

In order to increase the accurate of the results, also decrease the variables, even in the low speed simulation, we use the same time step number with the 0.05s in model scale and 0.16s in full scale.

4.2.4.8 Measurements

At last of physical setting part, the value we need to measure in these simulations should be clearly, contain:

a) Ship resistance coefficient:

1. Total resistance coefficient C_t (Model scale and full scale)
2. Friction resistance coefficient C_f (Model scale and full scale)
3. Pressure resistance coefficient C_p (Model scale and full scale)

b) Wave patterns

c) Wake field pictures

d) Mean wake fraction

In addition, we need to know how to measure the mean wake fraction on the propeller plane at the aftship: Set the measure cartesian propeller cylindrical at the aft ship, which can see the figure 2.4. Then collect the numbers of velocities on this measure panel, and then calculate the average number of this

velocities to calculate the mean wake. Table 4.6 shows the dimensions of the propeller that help us building the measure propeller cylindrical.

Even the model in this research simulation has no propeller, but in order to

Propeller diameter	D (m)	8.12	0.203
Propeller center, long. location (from FP)	x/LPP	0.985714	0.985714
Propeller center, vert. location (below WL)	z/LPP	0.040414	0.040414
Propeller center, long. location (from FP)	x (m)	275.99992	6.899998
Propeller center, vert. location (below WL)	z(m)	11.31592	0.282898
Propeller center, long. location (in ship c.s.)	x (m)	135.99992	3.399998
Propeller center, vert. location (in ship c.s.)	z(m)	5.18408	0.129602
Propeller rotation direction (view from stern)		clockwise	clockwise

Table4. 6. The dimensions of propeller

5 RESULTS

The measurement results read from the simulation cases are shown in this chapter, which consists of: the scale effect of ship resistance, wave patterns and wake field between model scale ship and full scale ship. We also mix some discussions in this part that relate to the results closely.

5.1 Test cases description

The cases we simulated are “ship sailing in calm water at a constant speed” in this thesis, or we can say we simulate the ITTC ship towing test for both model and full scale ships. The ship model is named “Japan Bulk Carrier”, it is a single screw stern bulk carrier model and provided by “Tokyo 2015 Workshops”. The model doesn’t need to consider about the rudder, propeller and ESD. Because the center of gravity and the moment of inertias are unknown, we don’t need to consider the movement of ship (sinkage and trim), what we simulate is just a fixed ship model, this condition is write as FRZ0 what we will use blow.

1. Ship speed calculation

We need to simulate several cases that ship sailing at different speed for both model scale and full scale ship: 8knots, 10knots, 12knots, 14knots, 14.5knots, 15knots, and 17knots. The design speed of JBC is 14.5 knots so for most of comparison we select the cases at this speed. In the Star CCM+, we only can input the velocities that use the unit “meter per second (m/s)”. Before simulation, we need to transform it by ourselves.

Velocities for full scale ship is easy to calculate, just use unit conversions method to transform. But for model scale ship, we need to use Froude number equations to convert because for model and full scale ship, we keep the Fr number equivalent which we have mentioned in the background part. The process is:

$$\text{Model scale ship:} \quad Fr_m = \frac{V_m}{\sqrt{gL_m}} \quad (5.1)$$

$$\text{Full scale ship:} \quad Fr_s = \frac{V_s}{\sqrt{gL_s}} \quad (5.2)$$

$$Fr_m = Fr_s \quad (5.3)$$

$$\text{Velocity of model scale ship:} \quad V_m = \sqrt{\frac{L_m}{L_s}} \cdot V_s \quad (5.4)$$

2. Test cases

Table 5.1 and 5.2 show all the cases we need to done during the whole simulation phase: 9 cases for model scale ship and 8 cases for full scale. The table also shows how many steps and physical time we calculated for each case. The physical time here means the time that the ship sailing in the water what we simulate. The convergence time number means the real physical time in CFD simulation that the results get converged. We find that, with increasing speed of the ship, the required convergence time is correspondingly delayed for both model and full scale ship.

Model Scale									
Case	1.1	1.2	1.3	1.4	1.5	1.6	1.7	1.8	1.9
Attitude	FR _{z0}								
Ship	Full	Full	Half						
Steps	26000	26000	14000	13000	10000	10000	11000	11000	11000
Physical Time	250s	250s	140s	120s	100s	100s	110s	110s	110s
Results convergence time	40s	No	40s	40s	40s	40s	40s	60s	70s
Turbulence Model	SST k- ω	k- ϵ	SST k- ω						
Speed	14.5	14.5	14.5	8	10	12	14	15	17
Fr	0.1423	0.1423	0.1423	0.0785	0.0981	0.1178	0.1374	0.1472	0.1669
Flow Velocity	1.1793	1.1793	1.1793	0.6506	0.8133	0.9760	1.1386	1.2200	1.3826
	4	4	4	7	4	1	7	1	7
							LPP =	7	m

Table5. 1. Test cases for model scale

Full scale									
Case	2.1	2.2	2.3	2.4	2.5	2.6	2.7	2.8	
Ship type	Full	Half	Full	Full	Full	Full	Full	Full	
Attitude	FR _{z0}								
Steps	20800	24500	12500	12500	12500	12500	15500	20800	
Physical Time	400s	400s	400s	400s	650s	400s	500s	600s	
Convergence time	350s	No	300s	300s	300s	350s	400s	500s	
Turbulence Model	SST k- ω								
Ship Speed	14.5	14.5	8	10	12	14	15	17	
Fr	0.1423	0.1423	0.0785	0.0981	0.1178	0.1374	0.1472	0.1669	
Input Flow Velocity	7.45880	7.45880	4.11520	5.14400	6.17280	7.20160	7.71600	8.74480	
							LPP =	280	m

Table5. 2. Test cases for full scale

5.2 Empirical Value

Because there is no physical experiment in this master thesis project, even the model scale tests are simulated by using CFD method, in order to verify if the results of CFD calculations are correct, some empirical data that calculated by using the empirical formula is needed. These equations are provided by ITTC model scale towing test.

ITTC towing tank tests aim to measure the resistance and other ship hydrodynamics values by using model towing test. And from the experimental data, ITTC summarizes some empirical equations to get empirical data. What we use here are the resistance coefficient equations. In fact, only friction resistance can be calculated by using these equations, which is related to the Reynolds number Re . Residual resistance is linked to Froude number Fr , and it has no empirical equations. [6]

The following equations are applied for resistance test:

Froude Number:

$$Fr = \frac{v}{\sqrt{gL}} \quad (5.5)$$

Where v is ship speed, g is gravitational acceleration, L is the ship length between perpendiculars. Fr defines how fast the ship is.

Reynolds Number:

$$Re = \frac{VL}{\nu} \quad (5.6)$$

V is ship speed, L is the ship length between perpendiculars, ν is water kinematic viscosity. The Reynolds number is defined as the ratio of inertial forces to viscous forces, this number influence the friction resistance of ship.

Resistance coefficients:

$$C_R = 2R_R/(\rho SV^2) \quad (5.7)$$

Where S is the wetted surface area that is normally calculated from the body plan to the still waterline and ρ is the water density. V is ship speed. $C_T = C_T(Re, Fr)$ is total resistance coefficient, C_P is pressure resistance coefficient, C_F is friction resistance coefficient. Ship resistance coefficient defines how big resistance ship suffers.

From ITTC (1957) model-ship correlation line, we can get the empirical formula of friction resistance coefficient:

$$C_F = 0.075/(\log_{10}Re-2)^2 \quad (3.5)$$

This equation is use to verify if our results from CFD program are right in this master thesis.

5.3 Residuals:

Residual plot shows the difference in the value between the current interaction and the previous one, it indicates how far the present approximate solution is away from perfect conservation (balance) of mass and momentum. The values it measured are the every solution of the RANS equations and turbulence models equations.

In this thesis, most of our cases use SST $k-\omega$ turbulence model as the equations solver. In this situation, the residuals will contains 7 values: one continuity equation; three momentum convergence equations in X, Y, Z directions; T_{ke} and T_{dr} equations (For SST $k-\omega$); and one equation for water (this variable comes from the VOF waves equation). For $k-\epsilon$ turbulence model, the variables in residual are the same except T_{ke} and T_{dr} are the variables for $k-\epsilon$ equations. Figure 5.1 and 5.2 show the residual plots for the simulation “Model scale ship (full ship in $Fr = 0.1423$) _Using SST $k-\omega$ turbulence model” and “Model scale ship (full ship in $Fr = 0.1423$) _Using $k-\epsilon$ turbulence model”

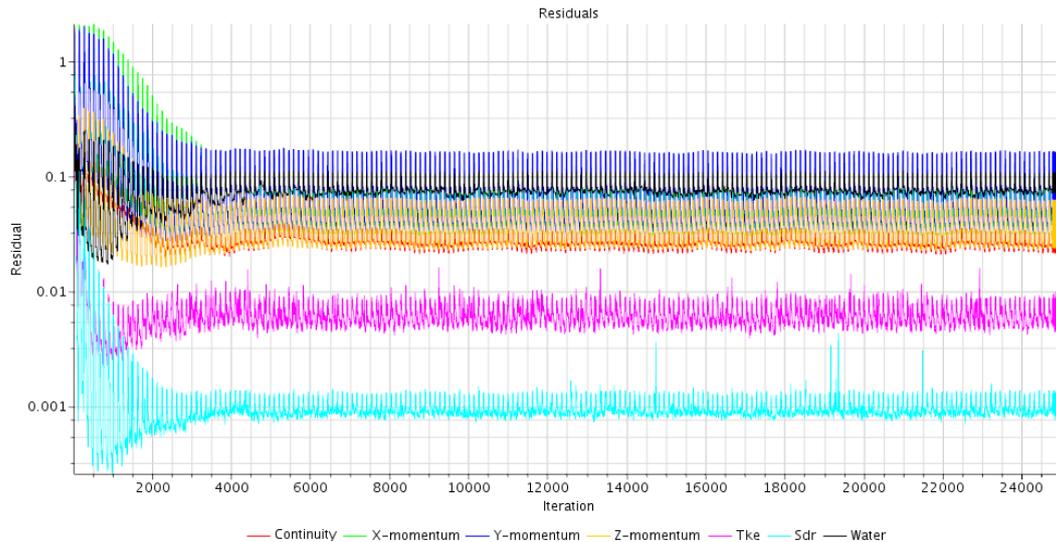


Figure5. 1. Residuals for model scale _ Fr =0.1423_ SST k- ω turbulence model

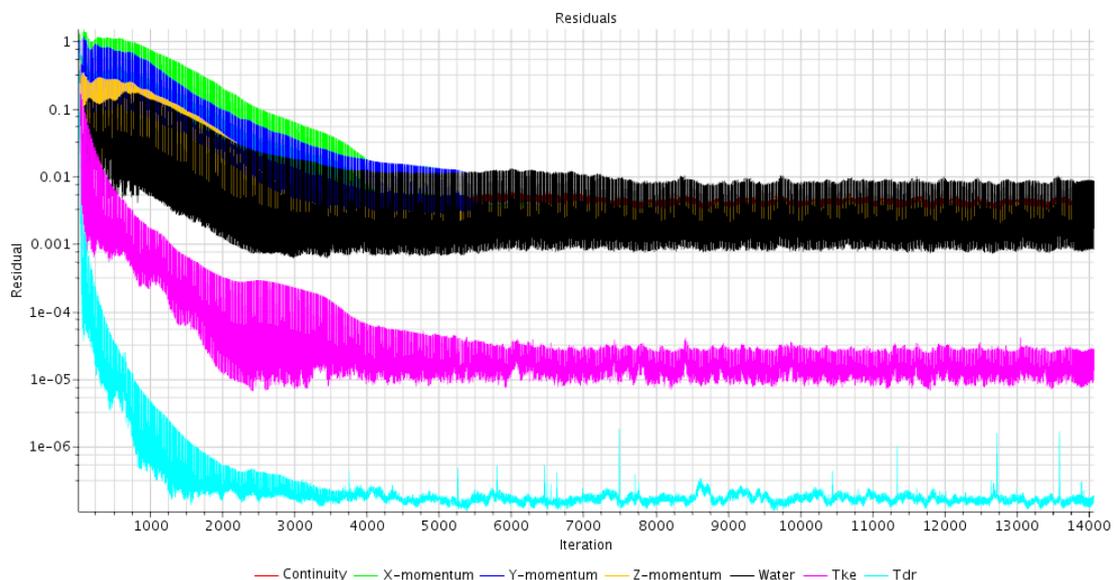


Figure5. 2. Residuals for model scale _ Fr =0.1423_ k- ϵ turbulence model

The status of residuals indicates the convergence of results. If all the residuals go down with every interaction, it means that the results are a good convergence. The practical convergence is obtained when all the values are below 0.01. But in this thesis, because there are vortices after the ship, in most conditions, residuals cannot reach 0.01. In fact, the results always have small fluctuations at the end, which we will discuss in the discussion part later.

Sometimes, because of the large time step or mesh, at the beginning of the simulation, the residuals may rise to more than 10,000 and then gradually decrease. This situation occurred in the full-scale simulation, which can be shown in the appendix. But this will not influence the final results of the simulation because the calculation process is just the process that the CFD program tries to approach the correct answer gradually, even though there are some very strange values during calculation, but it will find the right answer at last if we set all the parameters correctly.

Besides of residuals, another indication of a good convergence is behavior of one of the key parameters. For this case, when resistance keeps the same level at every interaction step.

5.4 $k-\omega$ turbulence model & $k-\epsilon$ turbulence model (Verification)

Before we start to simulate all the test cases, we need to guarantee that the results we calculate from CFD program are correct. Except contrast with the empirical data, another method is tried to use different mesh types or solve methods.

Mesh types we use most in this simulation are hexahedron type mesh. This type of mesh is easy to transfer information to neighbourhood and helpful for reducing the calculation time. So if we use other mesh types will greatly increase the calculation time, in this thesis we try to use two turbulence models to measure the results: SST $k-\omega$ turbulence model and $k-\epsilon$ turbulence model. Comparison of all the results between these two methods to see if the results are credible, and try to find the better solution.

1. Ship resistance coefficient

Even all the measured values are needed to compare, but the resistance coefficients are the main standard that we measure if the results are correct due to two reasons: 1). These values can be read directly; 2). We have some empirical and EFD values to compare. The empirical values are calculated by using 1957 ITTC correlation line of model ship. EFD values are from the values published in the website of "Tokyo 2015 Workshop". The comparison condition what we choose is the model scale ship sailing in design velocity 14.5 knots, which $Fr = 0.1423$. Results are shown in table 5.3.

Model scale(Full ship); $Fr = 0.1423$ (Design speed)				
Methods	SST $k-\omega$	$k-\epsilon$	Difference (%)	Empirical
C_f	0.0031341	0.0030403	2.99	0.00316
C_p	0.000932	0.0010618	13.9	-
C_t	0.004066	0.0041022	0.89	0.00426
Empirical equation: $C_f = 0.075/(\lg Re-2)^2$ EFD value: $C_t = 0.00426$ (Ship is not fixed)				

Table5. 3. Compare with two method and EFD value

From the table 5.1, we can see that even the total resistance coefficient C_t value is quite closed, but pressure coefficient has 13.9% error between these two methods. And when use SST $k-\omega$ turbulence model as equations solver, the C_f value is much closer than empirical date 0.00316. In addition, the EFD value of total resistance of ship is in the condition that ship is free to heave and pitch, and that is the reason why it has a larger number.

2. Wave pattern

Wave pattern is another result that we need to measure. These pictures show the generated waves by sailing ship, we can compare with the wave patterns of these two methods to determine which one is the better method to use. And the results is obviously, in figure 5.3, some very nice diffusion waves on both sides of the ship are observed, which are consistent with our knowledge. But in figure 5.4, the wave patterns by using $k-\epsilon$ method calculate, are quite strange. These may happen because use the unsuitable mesh or physical setting for $k-\epsilon$ turbulence model solver, we will discuss it in discussion part. More

attempt can be carried on to find the reason why this problem happened, but in order to ensure that the following simulations proceed smoothly, we choose SST k- ω turbulence model as the equations solver at last.

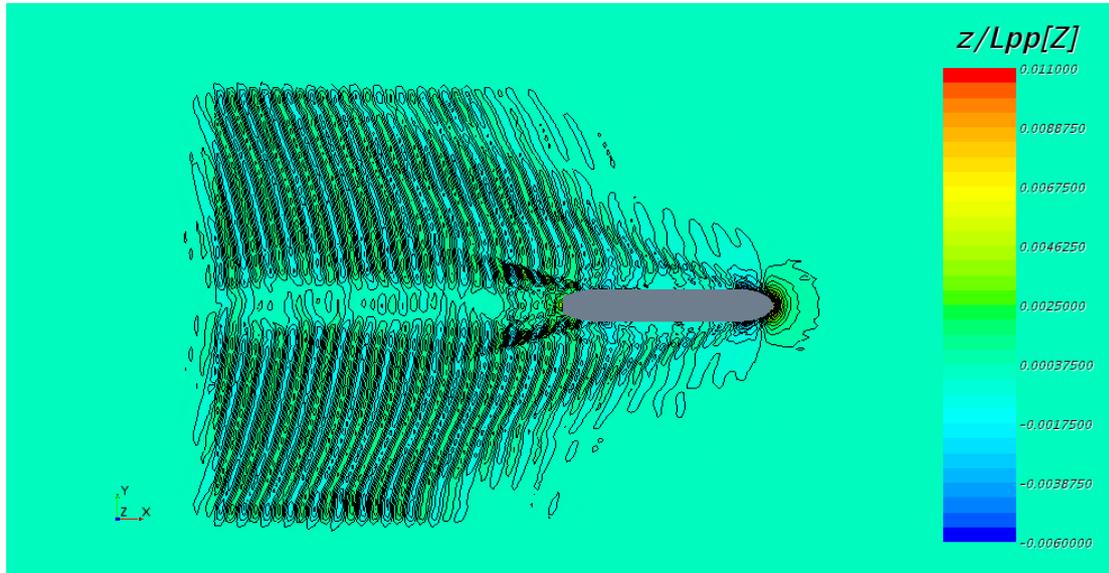


Figure5. 3. Wave patterns of ship (SST k- ω turbulence model)

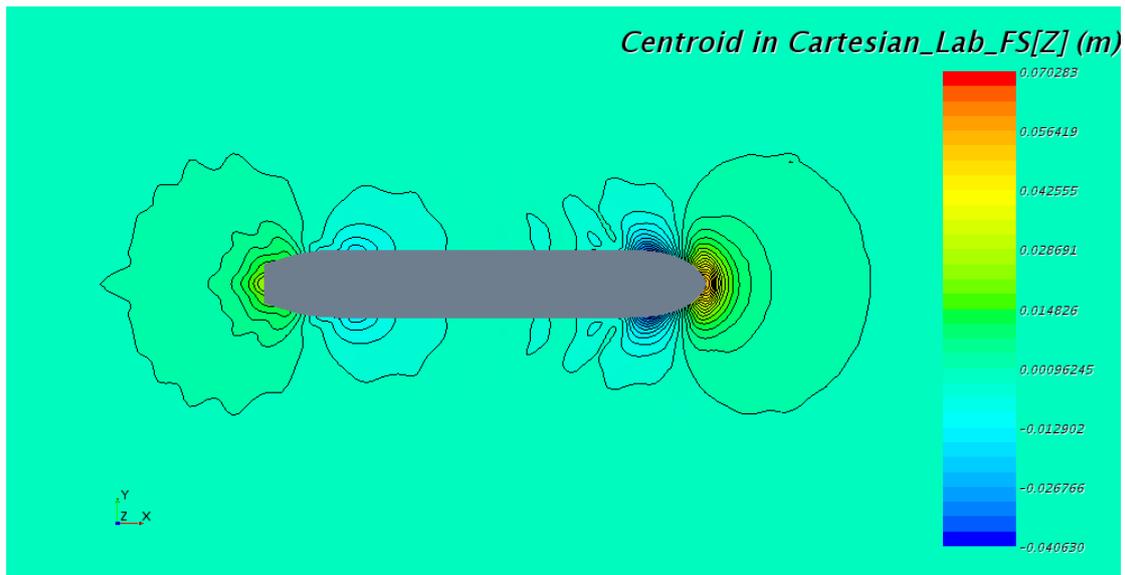


Figure5. 4. Wave patterns of ship (k- ϵ turbulence model)

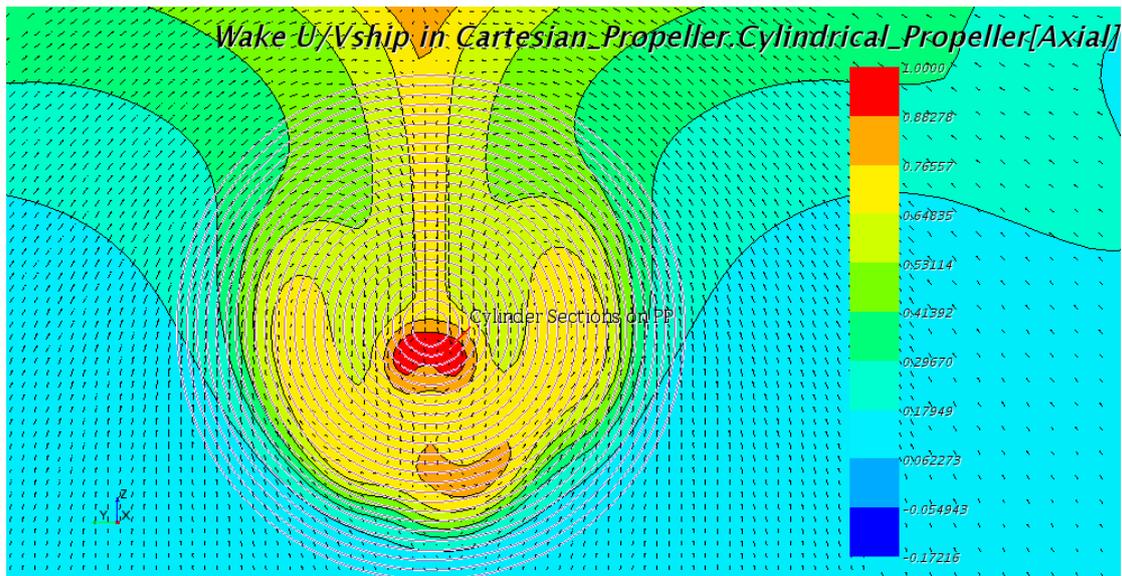
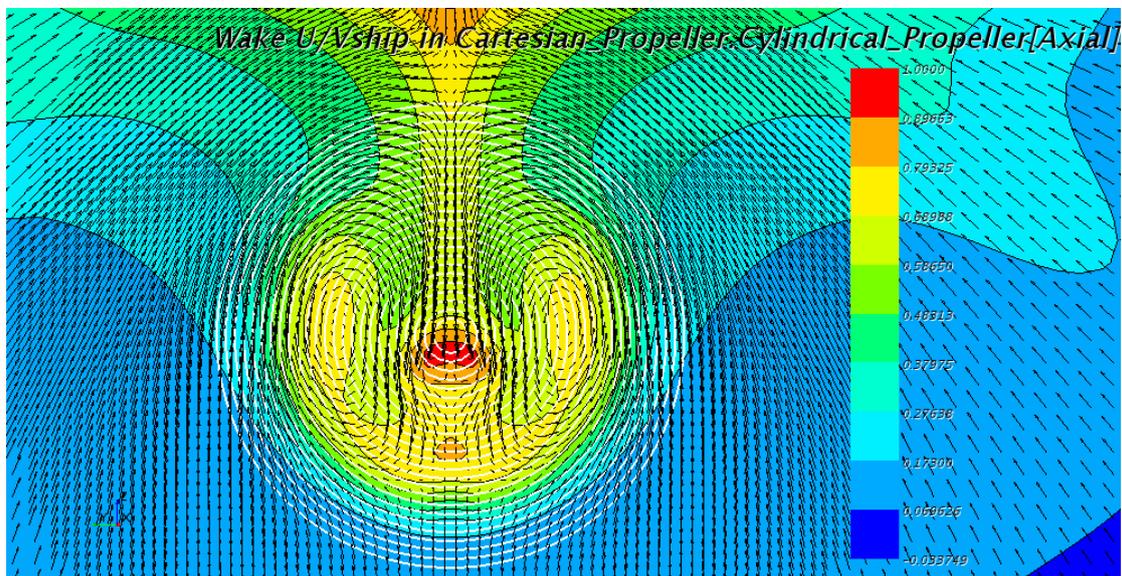
3. Nominal wake fraction

Wake fraction is the main measurement we want to get in this master thesis, so the quality of this values determined which method we choose. Table 5.4 shows the mean nominal wake number in the area of propeller plane:

Turbulence model	SST k- ω	k- ϵ
Mean wake fraction	0.4896	0.4899

Table5. 4. Mean wake value in the propeller plane

Results show that the results of these two conditions are almost the same. But figure 5.5 and 5.6 shows the different between these two method, when we use SST k- ω turbulence model, the shape of wake on the propeller plane after stern is asymmetric, while it's get a symmetrical plot when we use k- ϵ turbulence model as the turbulent equations solver.

Figure5. 5. Wake in propeller plane (SST k- ω turbulence model)Figure5. 6. Wake in propeller plane (k- ϵ turbulence model)

4. Conclusion

Consider all the factors mentioned above:

1. The distribution of resistance is different, but the total resistance is the same.
2. Wave patterns are quite different, when use SST k- ω turbulence model it's fit our knowledge.

3. Mean wake fraction is similar, the shape of the wake after ship stern also is the same.

The conclusion is in all the following simulation we use SST k- ω turbulence model as the equations solver.

5.5 Full ship and half ship

These ship test cases are quite big work. For each of the case it may cause lots of time to calculate. In order to ensure all the simulation can be done, we need to in ensuring the accuracy of the results as the premise to minimize the computation time.

The reason that the simulation costs lots of time to calculate is there are nearly 6 million cells in the simulation region in each case. The computer needs to calculate all the equations in these cells for every step. This will use lots of computer memories. The way that solves this problem is trying to reduce the number of mesh cells.

Of course we can't change the mesh type and size, what we have set already to keep the Y^+ number in the range of 30 – 100 around ship hull. The solution to achieve this goal is: cut away half of the whole region from the middle XOZ plane and just simulate half of the ship, set the middle plane as a symmetry plane, which can be shown in figure 4.3. And in the export results phase, use this symmetry plane to get a complete result.

This operation will reduce half amounts of mesh cells and save more than 2 days calculation time for each case. But the prerequisite for complete this operation is:

1. Model is completely symmetrical;
2. Friction resistance is the dominate resistance;
3. The vortex after ship stern are not mixed from each side of ship

5.5.1 Model scale

1. Streamlines

In Star CCM+, streamlines are a family of lines that are instantaneously tangent to the velocity vector and the magnitude of the flow. These show the direction a massless fluid element will travel in at any point in time. By observing the streamlines, we can know how the water flow around a ship hull and after ship.

The streamlines of water flow for the full model scale ship and half model scale ship are shown in figure 5.7. The case we choose is “Model scale _ SST k- ω turbulence model _ Fr =0.1423”.

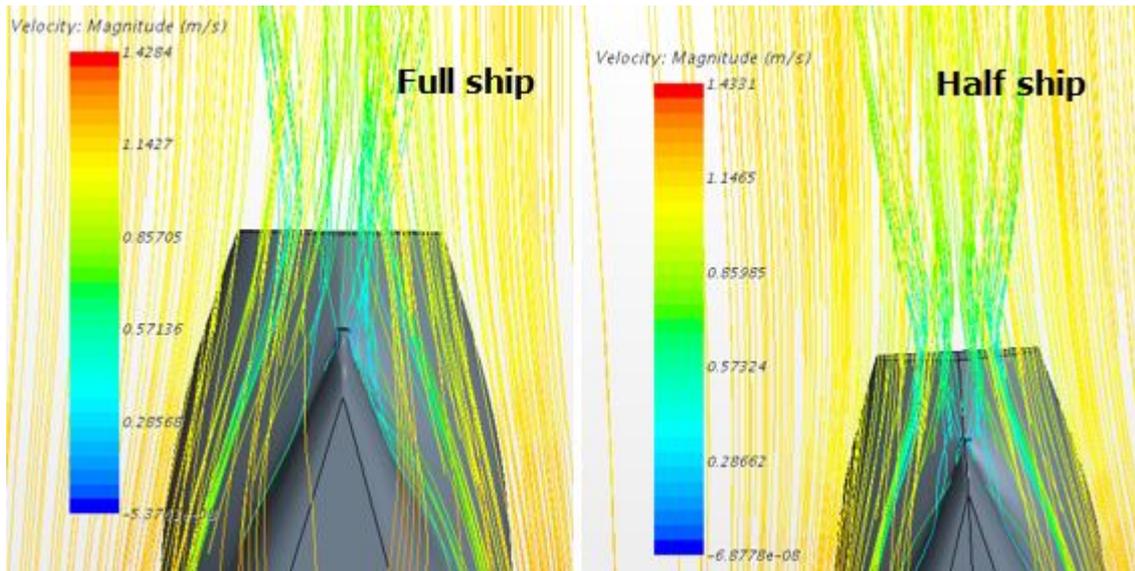


Figure5. 7. Streamlines of full ship (Model scale)

Compare with the streamline in figure 5.7, we can see that even has difference, the vortex are not mixed after full ship stern, and the flow directions are similar with the half ship streamline. Which we can say we may use the half ship to simulate in model scale ship. But the final conclusion should be drawn after comparing the measurements.

2. Ship resistance coefficient & wake

Table 5.5 shows the comparison of ship resistance coefficient of full and half ship. Mesh cells of half ship simulation nearly decreased 3 million compare to the full ship. We keep all the other parameters same includes the time step, which is quite related to the mesh setting.

Model scale; SST k- ω ; Fr = 0.1423(Design speed)				
Ship Type	Full ship	Half ship	Difference (%)	Empirical Data
Mesh Cells	5,881,132	3,002,787	-	-
Time step	0.05s	0.05s	-	-
C_f	0.003134	0.003129	0.17	0.00316
C_p	0.000932	0.000936	0.42	-
C_t	0.004066	0.004066	0	-
Empirical equation: $C_f = 0.075/(\lg Re-2)^2$				

Table5. 5. Comparison of ship resistance coefficient of full and half ship (Model scale)

The values in the table indicated that quite small difference between full ship and half ship: total resistance is nearly the same; we can observe that the friction resistance is 0.17% larger than half ship, and 0.42% less in pressure resistance. The reason of this maybe because after stern part, full ship's water flow still flow to one side of the ship cause by inertia, the flow slightly to the right, this may cause the change of the distribution of ship resistance. But in the half ship simulation, the water flow is totally symmetrical. This effect also can be seen in the comparison of the wake field, which is shown in figure 5.8.

We also observe that the flow velocity around the ship of full ship is slightly smaller than half ship from figure 5.4. By using Reynolds number equation and 1957 ITTC friction empirical equation we can know that a smaller flow velocity has a larger friction resistance.

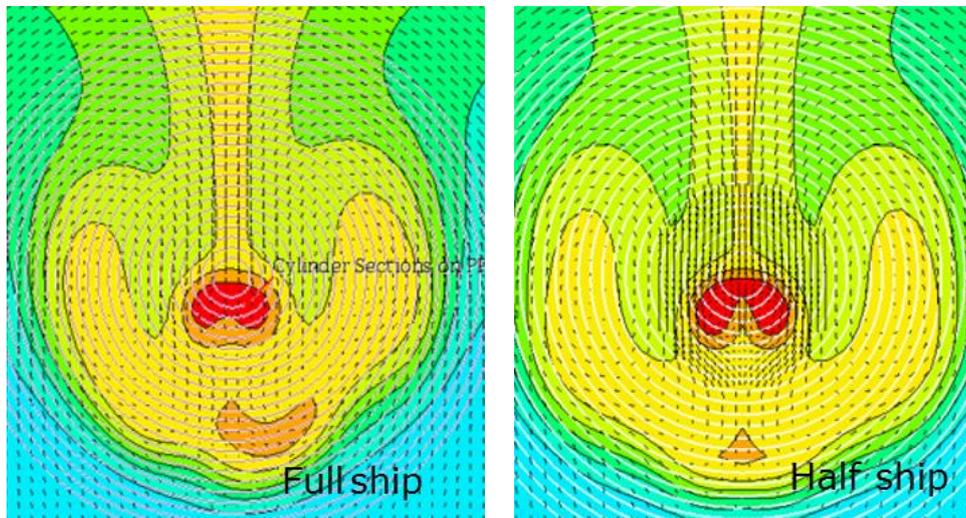


Figure 5.8. Comparison of the wake field of full ship and half ship

And another important measurement is the mean value of the wake fraction on the propeller plane, of course it will have difference because the wake field is already changed. But if the difference is in the acceptable range, we just say we can use half ship to simulate. Table 5.4 shows the comparison of the mean wake fraction.

Ship type	Full	Half	Difference (%)
Mean wake fraction	0.4896	0.519	5.6%

Table 5.4 Comparison of mean wake fraction

In general, for model scale ship simulation, there is no much different between full ship and half ship, in order to save the calculation time, we choose half ship to simulate.

5.5.2 Full scale

1. Streamlines

As same as the model scale ship, we try to test and verify if the full scale cases also can use half ship to save the calculation time. Streamlines of full ship and half ship are shown in figure 5.9. The case we choose is “Full scale _ SST k- ω turbulence model _ Fr =0.1423”. At this Froude number ship is sailing in the design speed, which is 14.5 knots.

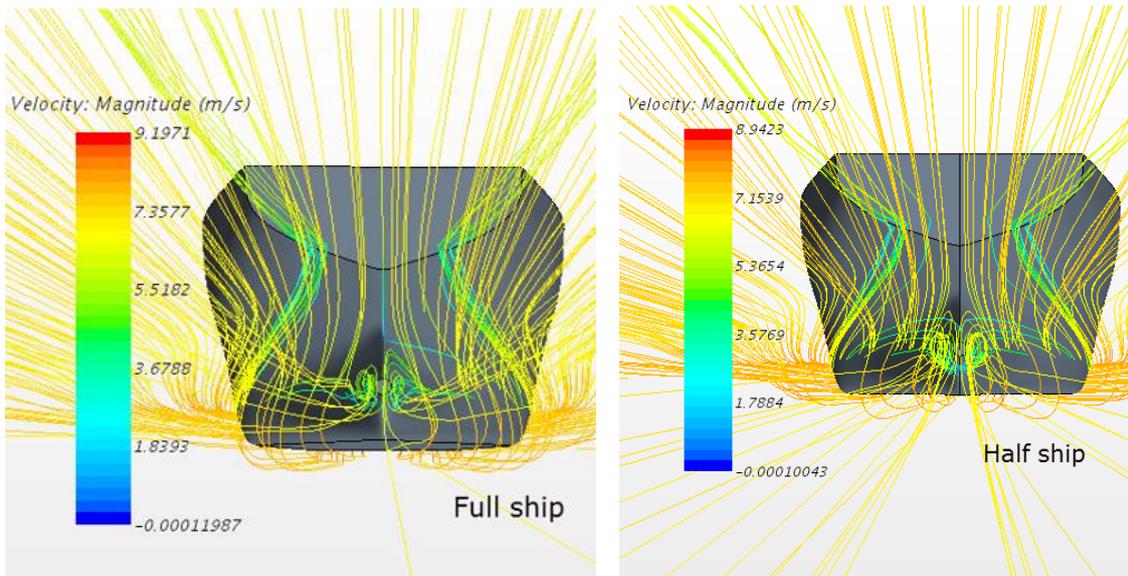


Figure5. 9. Comparison of streamline of full and half (Full scale)

From the figure we notice that in full scale ship, compare to the model scale the asymmetry of the flow after ship stern is more obvious. But the vortex at the aftship still not mixed.

2. Ship resistance coefficient & Residuals

Table 5.6 indicates the Comparison of ship resistance coefficient of full and half ship for full scale cases

Full scale; SST k- ω ; Fr = 0.1423(Design speed)				
Ship Type	Full ship	Half ship	Difference (%)	Empirical Data
Mesh Cells	6495589	3085864	-	-
Time step	0.16s	0.08s	-	-
C_f	0.001415	0.001413	0.14	0.00142
C_p	0.000412	0.000415	0.72	-
C_t	0.001827	0.001828	0.05	-
Converges	Yes	No	-	-
Empirical equation: $C_f = 0.075 / (\lg Re - 2)^2$				

Table5. 6. Comparison of ship resistance coefficient of full and half ship (Full scale)

In fact, even all the results are very similar, but if we choose half ship and half region to simulate, by using the same calculation steps and simulate physical time, the ship resistance coefficient can't get converged. Shown in figure 5.10. The residuals plot in figure 5.11 also indicates that the residuals start to get higher than 1 in the future steps. These indications show that if we want to get a converged result, maybe we need to speed much more time than we imaged. This is for us to cut computing time did not help. In order to get the accurate result, for full scale ship, we use full ship and domain to calculate.

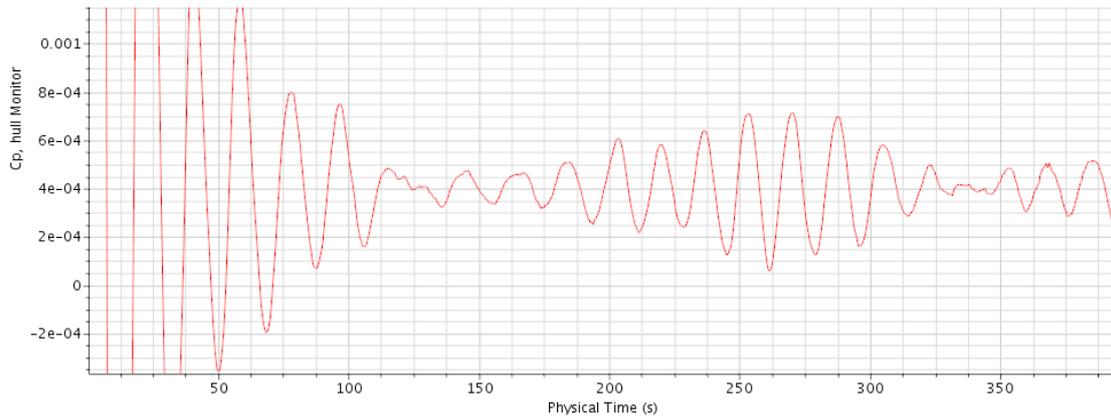


Figure5. 10. Cp for half ship case (Full scale)

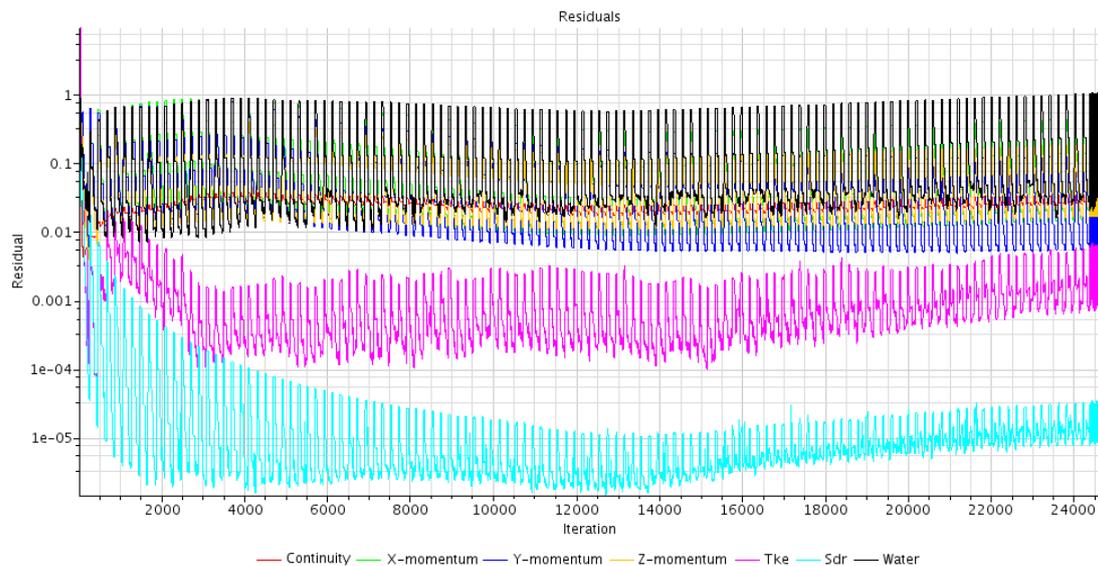


Figure5. 11. Residuals for half ship case (Full scale)

5.6 Comparison of ship resistance coefficient

5.6.1 Resistance coefficient for model scale ship

The scale effect of the ship resistance coefficient is one of the most important parts that we focus on in this master thesis. As we mentioned in the background part, ship resistance always contains two parts: friction resistance and residual resistance. Friction resistance is due to the viscosity between water flow and ship hull, this type of force is related to the Re ; the residual resistance is another type of resistance which is due to the generated wave of ship, this type of force is related to the Fr number. In order to intuitive to read the magnitude of resistance, we use the force coefficient to explain them:

$$C_F = 2R_F / (\rho S V^2) \quad (5.1)$$

For C_p and C_t values, because even though at the last of the simulation, pressure resistance coefficient and total resistance coefficient still have a stable fluctuation. Shown in figure 5.12.

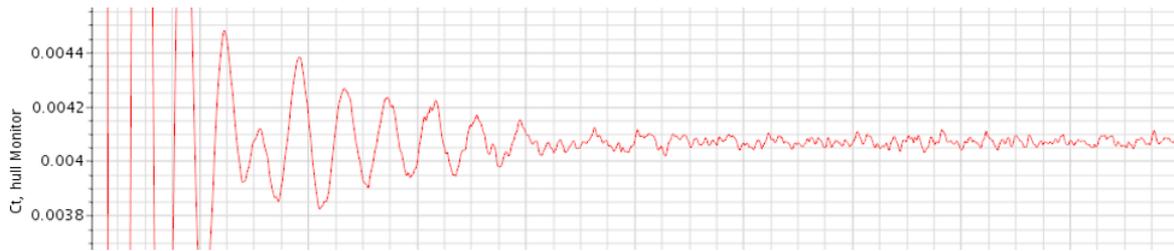


Figure5. 12. Cp number of model scale ship (Fr = 0.1423)

The way we solve this problem is capture 200 values of C_p and C_t at the end of the plots. In the region, the results have get converged, we collect the data and calculate the average value of them.

Table 5.6 shows all the resistance coefficient of the model scale ship at the speed from 8 knots to 17 knots, when 14.5 knots is the designed velocity. The comparison C_f value is calculated by using ITTC empirical equations. Where L_{PP} is 7 meters for model scale ship.

Model scale fixed ship							
Ship Speed	8	10	12	14	14.5 (Design)	15	17
Velocity(m/s)	0.6507	0.8133	0.9760	1.1387	1.1793	1.2200	1.3827
Fr	0.0785	0.0981	0.1178	0.1374	0.1423	0.1472	0.1669
Re ($\times 10^{-6}$)	4.11	5.14	6.17	7.20	7.46	7.71	8.74
Expectation of C_f ($\times 10^3$)	3.52	3.38	3.27	3.18	3.16	3.14	3.07
$C_f (\times 10^3)$	3.58	3.36	3.25	3.15	3.13	3.11	3.03
Difference of C_f (%)	1.58	0.53	0.63	0.86	0.96	1.02	1.43
$C_p (\times 10^4)$	9.53	9.48	9.78	9.17	9.36	9.45	10.45
$C_t (\times 10^3)$	4.53	4.31	4.23	4.07	4.07	4.05	4.07
Percentage of C_f	79.0%	78.0%	76.9%	77.4%	76.9%	76.7%	74.3%
$Re=(V*L_{PP})/NU$					L_{PP}	7	m
					Water density, PO	999.1	kg/m ³
Expectation of $C_f = 0,075/(lgRe-2)^2$					Kinematic viscosity, NU	1.107E-6	m ² /s
					Dynamic viscosity, MU	0.001106	Pa-s

Table5. 7. The resistance of model scale ship

From table 5.7, we can see that the friction resistance results calculated by CFD program are quite close to the empirical number, where the maximum difference is 1.58%. Compare to the pressure resistance, friction resistance dominate the large proportion of the total resistance. We summarize all these results and plot them in a picture, shown in figure 5.13.

We also notice that, the total resistance coefficient is nearly equal the sum of C_f and C_p directly, this means the direction of the pressure resistance and friction resistance is the same in these cases.

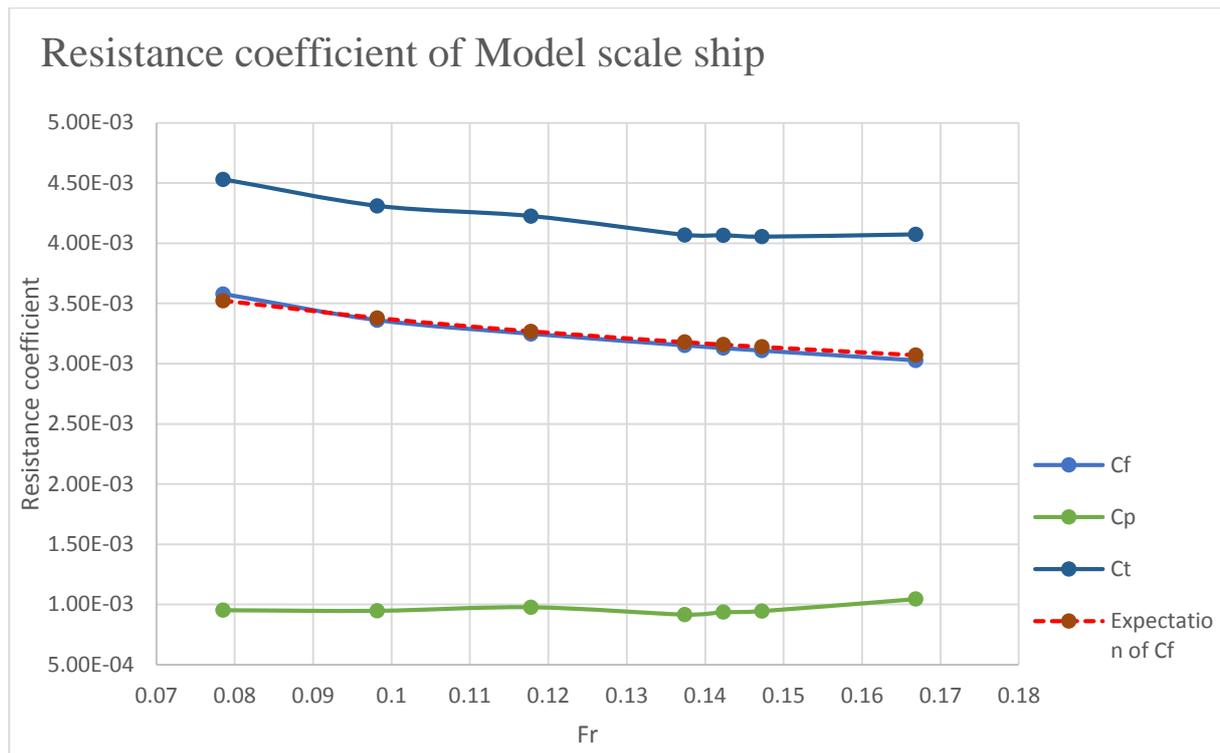


Figure 5.13. Resistance coefficient of model scale ship

From the plot, we notice that the C_f number is decreased with the increase of the Reynolds number. And it can use a smooth curve to connect each measurement point. This phenomenon consistent with our knowledge learned.

It is noted that the difference between calculated number and empirical number of friction resistance coefficient. When the ship velocity is slow, which is smaller than 10 knots, the calculated C_f number is larger than the empirical one; but when speed increased, the calculated values are smaller than empirical C_f values. This phenomenon also happens in the full scale ship cases, which can be seen in figure 5.9. We guess the reason is when the water velocity increased, because the vortex generation and flow separation at aft ship part, the pressure resistance start to occupy more and more number of components in total resistance, at the same time, the proportion of the friction resistance is decreased, so the real value of friction resistance is smaller than expect when ship speed is fast.

5.6.2 Resistance of full scale ship

1. Expectations of full scale ship resistance coefficient

Theoretically, we can use the results from model scale: for friction resistance coefficient, we can calculate it by using empirical equations; for pressure resistance coefficient, because the Froude number is the same, the pressure resistance coefficient should be the same also compare the model scale ship. In fact, this is the way how ITTC towing test estimate the full ship resistance. More detail is given in figure 5.14. For the same Fr number model scale and full scale ships, because the velocities are different, the friction resistance that related to the Reynolds number is changed, $C_{f1} > C_{f2}$; but the pressure resistance coefficient, C_{p1} should equal to the C_{p2} . And the total force coefficient equals the C_f plus C_p .

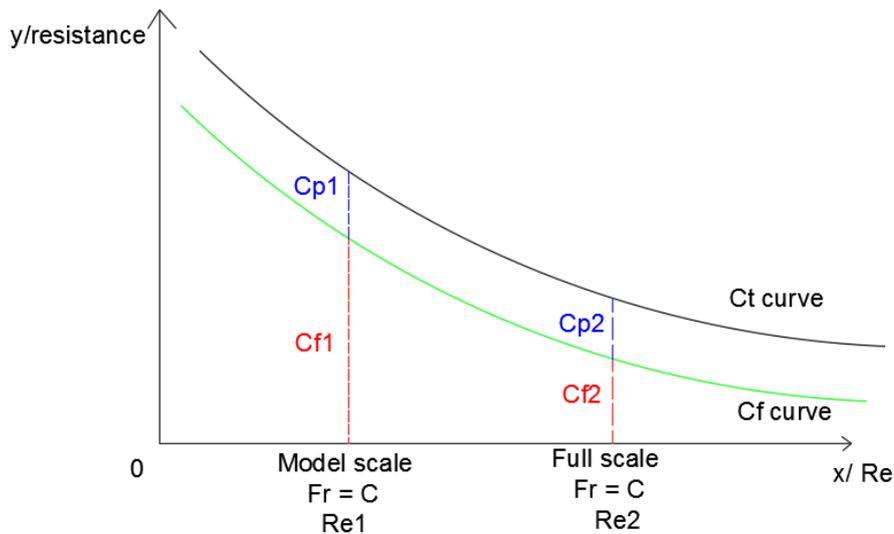


Figure5. 14. The resistance coefficient

2. Resistance of full scale ship

In fact, the real results we get is quite different compare with our expectation. The detail values can be seen in table 5.8. And we also use these results build a plot of the resistance coefficient of the full scale ships, shown in figure 5.15.

Full scale fixed ship							
Ship Speed	8	10	12	14	14.5	15	17
V(m/s)	4.1152	5.1440	6.1728	7.2016	7.4588	7.7160	8.7448
Fr	0.0785	0.0981	0.1178	0.1374	0.1423	0.1472	0.1669
Re ($\times 10^{-9}$)	1.04	1.30	1.56	1.82	1.89	1.95	2.21
Expectation of Cf ($\times 10^3$)	1.52	1.48	1.45	1.42	1.42	1.41	1.39
Cf ($\times 10^3$)	1.57	1.51	1.46	1.42	1.41	1.41	1.38
Difference of Cf (%)	3.26	1.91	0.67	0.06	0.15	0.33	0.69
Expectation of Cp ($\times 10^4$)	9.53	9.48	9.78	9.17	9.36	9.45	10.45
Cp ($\times 10^4$)	4.65	4.80	4.50	3.80	4.12	4.10	5.24
Difference of Cp (%)	51.2	49.4	54	58.6	56	56.6	49.8
Ct ($\times 10^3$)	2.04	1.99	1.91	1.80	1.83	1.82	1.90
$Re=(V*LPP)/\nu$					LPP	280	m
					Water density, ρ	999.1	kg/m ³
Expectation of Cf = $0,075/(\lg Re-2)^2$					Kinematic viscosity, ν	0.00000	m ² /s
					Dynamic viscosity, μ	0.00110	Pa-s

Table5. 8. Resistance coefficient of full scale ship

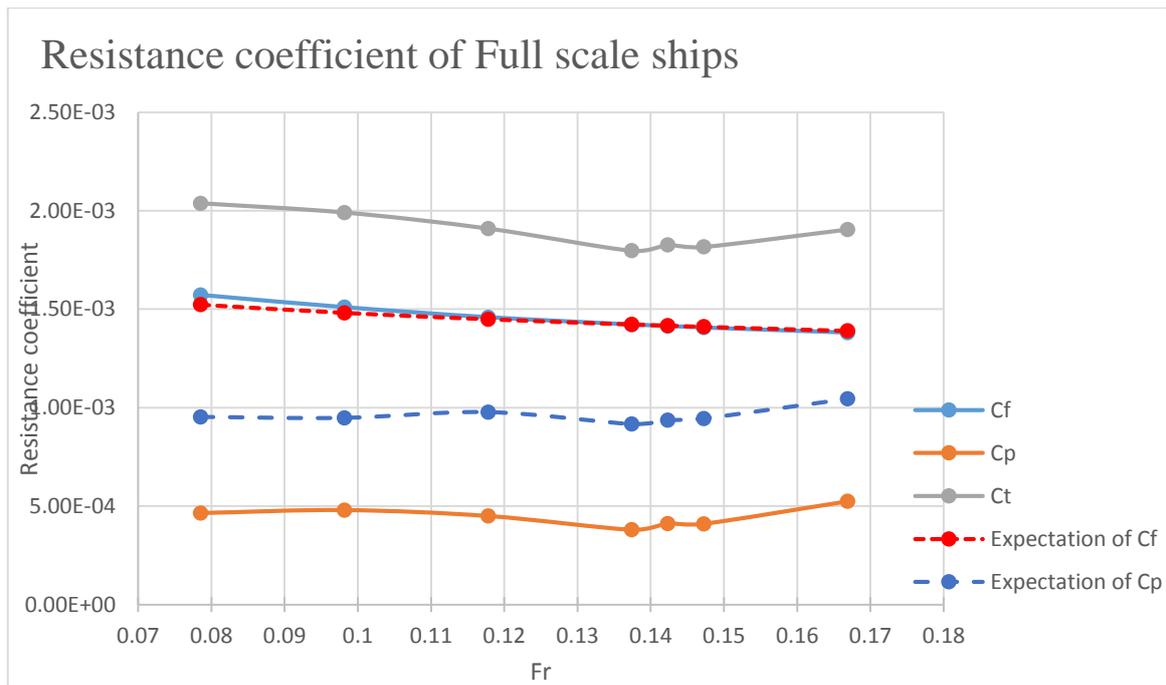


Figure 5.15. Resistance coefficient of full scale ships

From the table 5.8, we can see that the friction resistance coefficient is significantly decreased with the increased of Reynolds number, the difference of C_f is very small, which is in the acceptable range.

But for pressure resistance coefficient, the difference is quite big. The actual pressure resistance coefficients are nearly half smaller than what we expect, from 49.4% to 58.6%. This means the method that use ITTC towing test to estimate the full scale ship resistance does not fit this Japan bulk carrier. The reason for this result maybe complex, the most possible reason is due to the reduction of a large flow separation and vortex formation domain at the aftship. Which we can compare the streamlines at the aftships of model scale ship and full scale ship in the figure 5.4a and 5.6. Less boundary layer separation and smaller vortex happened at the aftship of the full scale ship. More detail about this question we will discuss in the discussion part.

5.6.3 Scale effect of resistance coefficient

By using the results summarized in table 5.8, we build a plot of the full scale and model scale, shown in figure 5.16.

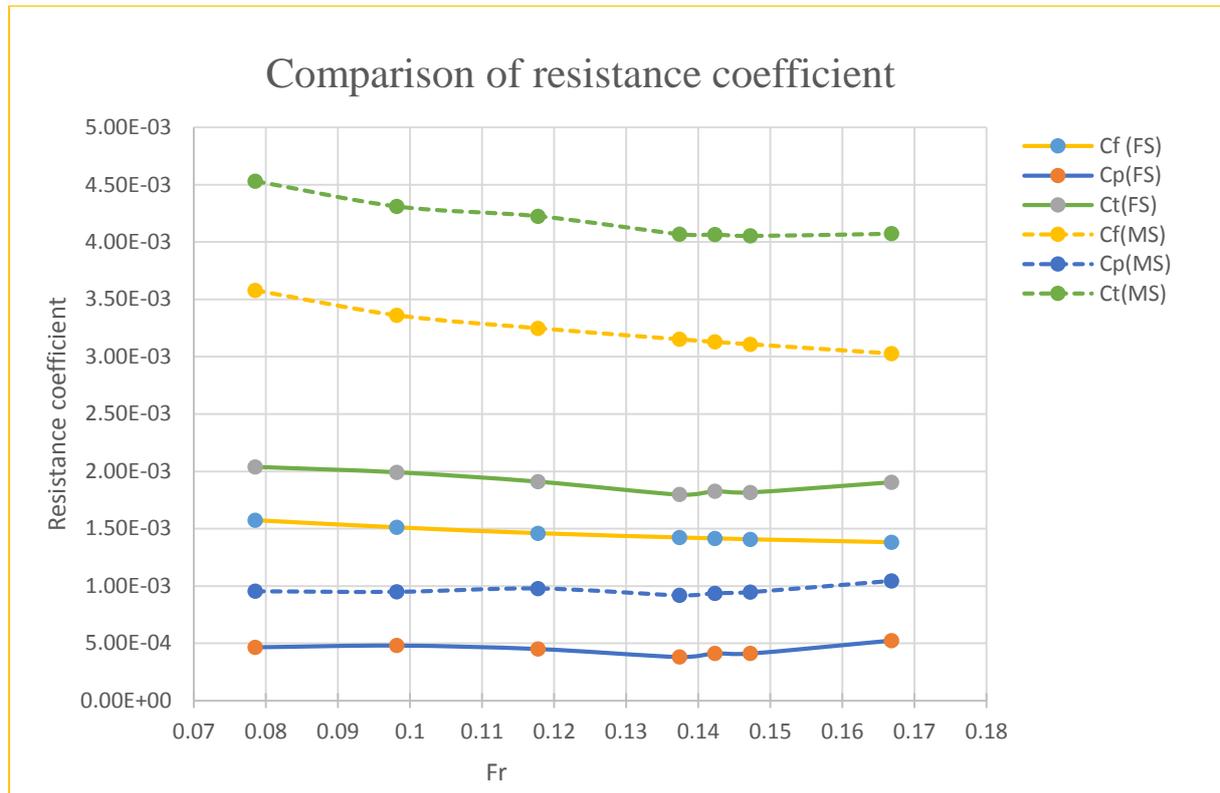


Figure 5.16. Comparison of resistance coefficient of model and full scale

In figure 5.16, the resistance coefficient of the model scale ship are presented in dotted line. Because the scale effects, the Cf number which related to the Reynolds number has a very significant difference, this difference comes from the increased flow speed around ship, the near-wall velocity profiles become fuller and the boundary layer become thinner, this will cause the decrease of the friction resistance coefficient.

Cp number also changed between model scale and full scale. The pressure resistance coefficient of the full scale ship are nearly half of the number compare with the model scale. This is due to the reason that the reduction of a large flow separation and vortex formation domain at the aftship. But the shape of the curve of the Cp is quite the same. We can sum up some laws of this, which we will discuss later in chapter 6.1.

In conclusion, because the scale effect, both the Cp and Cf are different between model scale and full scale. The way use towing test to estimate the full ship resistance is suitable for this Japan bulk carrier.

5.6.4 The distribution of resistance

The distribution of the ship resistance is depends on the speed of the ship, or we can say depends on the Froude number. In ship design area, we call the ship that sailing in $Fr < 0.2$ the slow speed ship; $Fr = 0.2 \sim 0.3$ called middle speed ship and $Fr > 0.3$ called high speed ship. In our case, ship's Fr is in the range of $0.65 \sim 1.38$, it's a typical low-speed ship. In general, for low speed ship, friction resistance is dominating 70% -80% of the total resistance, and the rest are the viscous pressure resistance and wave

making resistance. With increasing speed of the ship, more waves and vortex are made by ship, and the proportion of friction resistance coefficient will decreased.

We sum up the results of the proportion of friction resistance for both model scale and full scale ship, shown in figure 5.17.

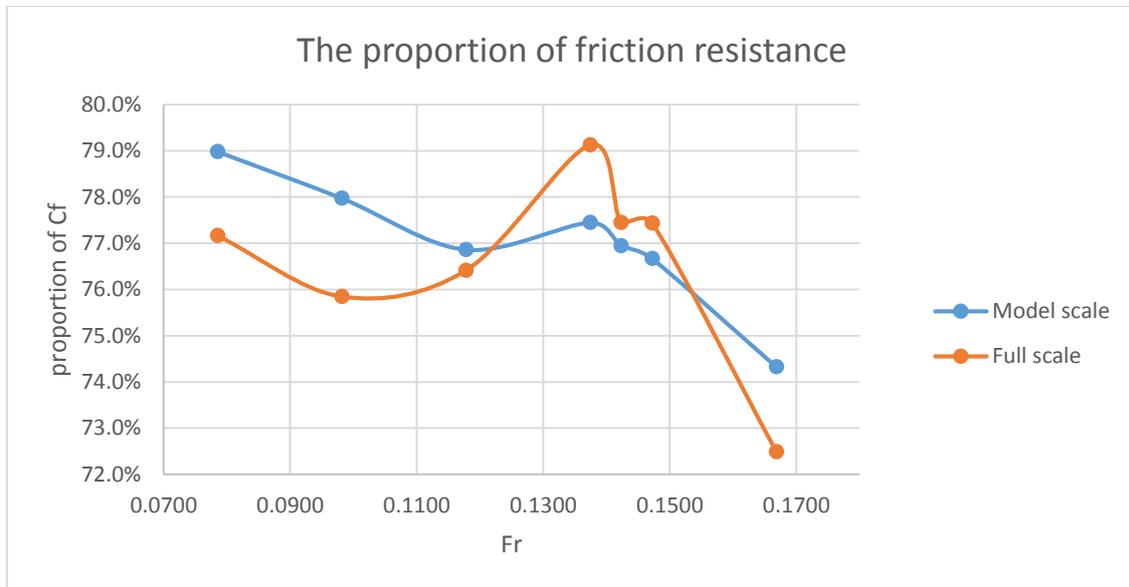


Figure5. 17. The proportion of friction resistance

The figure show that for both model scale and full scale ship, the proportion of friction resistance curve follow the law: go down, reach the peak of lowest place then go up, reach the highest place around the ship design speed and then go down again. This is because near the design speed area, pressure resistance reach it smallest value, at this time, even the friction resistance is still decreasing, but the pressure resistance decreases faster. So the proportion of friction resistance increases in this area.

5.7 Wave Patterns

The wave pattern that is left behind by a ship at sea, these waves are generated by the sailing ship and consists of a system of waves that around the hull the longitudinal wave, and another is interwoven with a system of transverse waves. The two systems advance with the boat so as to get stationary relative to it.

One of the many successes of the theory of dispersive waves is the explanation of the distinctive wave patterns formed by ships in relatively deep water. Such patterns are always the same and are referred to as Kelvin ship waves. These waves are proving that the envelopes of them stands at a fixed angle of 19.5 degrees and have a characteristic feathered pattern. Figure 5.18 display the components of the ship generated wave pattern. [13]

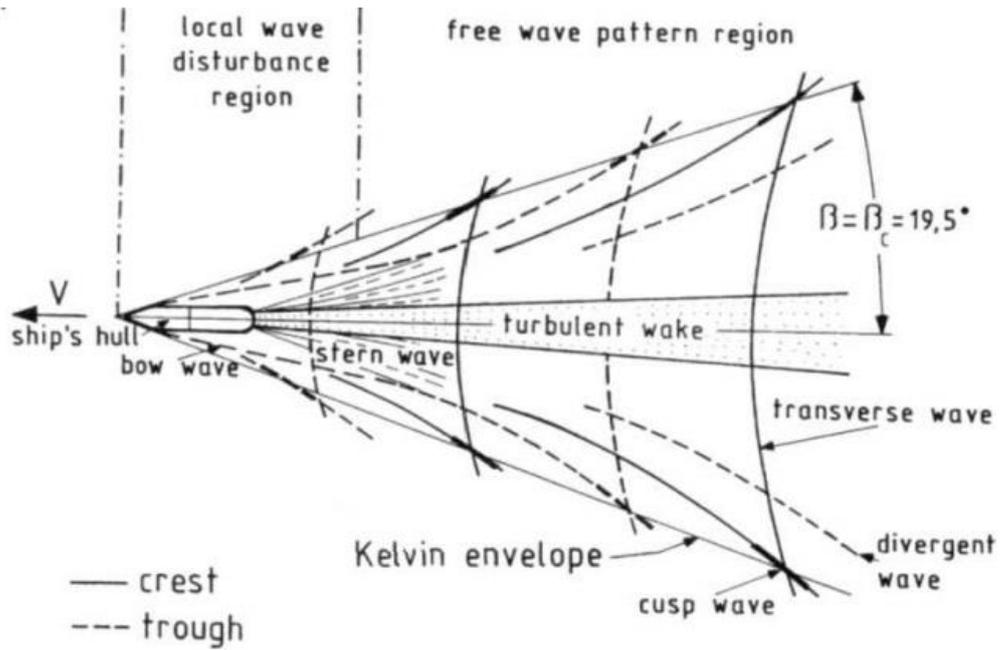


Figure5. 18. The components of wave pattern

(UNIVERSITY OF LJUBLJANA FACULTY FOR MATHEMATICS AND PHYSICS
 DEPARTMENT FOR PHYSICS KELVIN SHIP WAVES Student: KSENJA MAVER Advisor:
 Professor RUDI PODGORNIK Ljubljana, 2004)

Figure 5.16 shows the wave pattern we get from our cases. The cases choose are “Model scale ship _ Fr = 0.1423 (Design speed)” and “Full scale ship _ Fr = 0.1423 (Design speed)”.

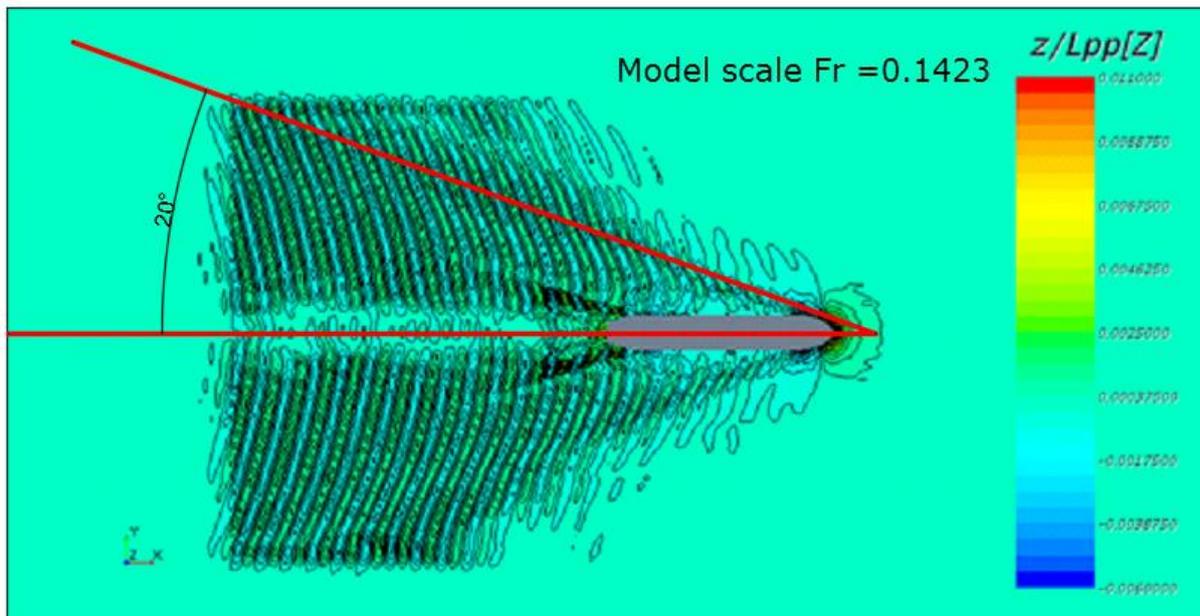


Figure5. 19. Wave patterns for model scale ship (Fr = 0.1423)

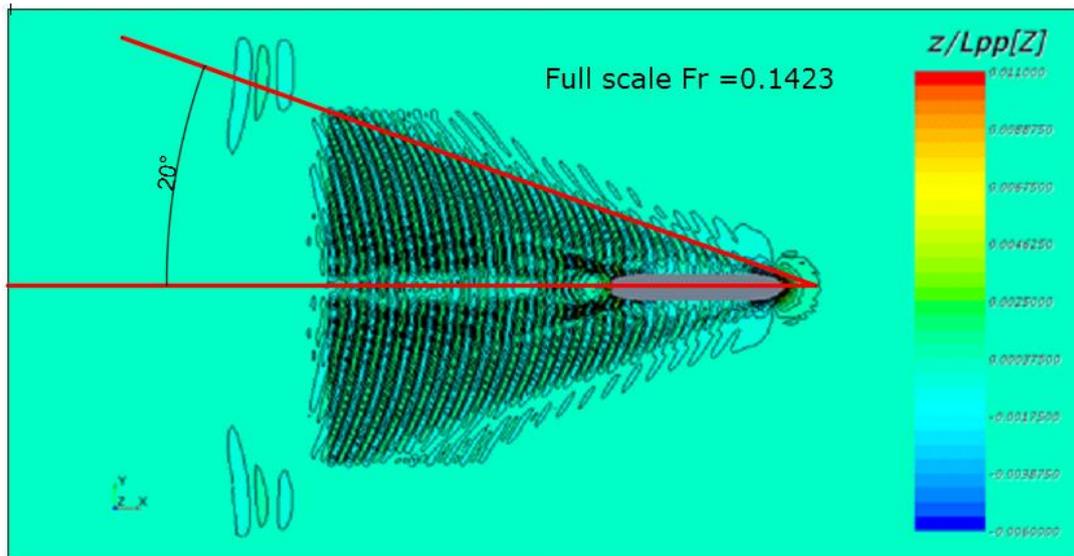


Figure5. 20. Wave patterns for full scale ship ($Fr = 0.1423$)

From the picture 5.19 and 5.20. We can see that the degree of wave patterns for both of model scale ship and full scale ship is nearly 20 degrees. Which we can say they are fit the Kelvin ship waves. As can be seen, the CFD calculation can reflect the hull Waves bow and stern wave system quite well for these cases, and the features of transverse wave and longitudinal waves can get a good description

5.7.1 The scale effects of wave patterns

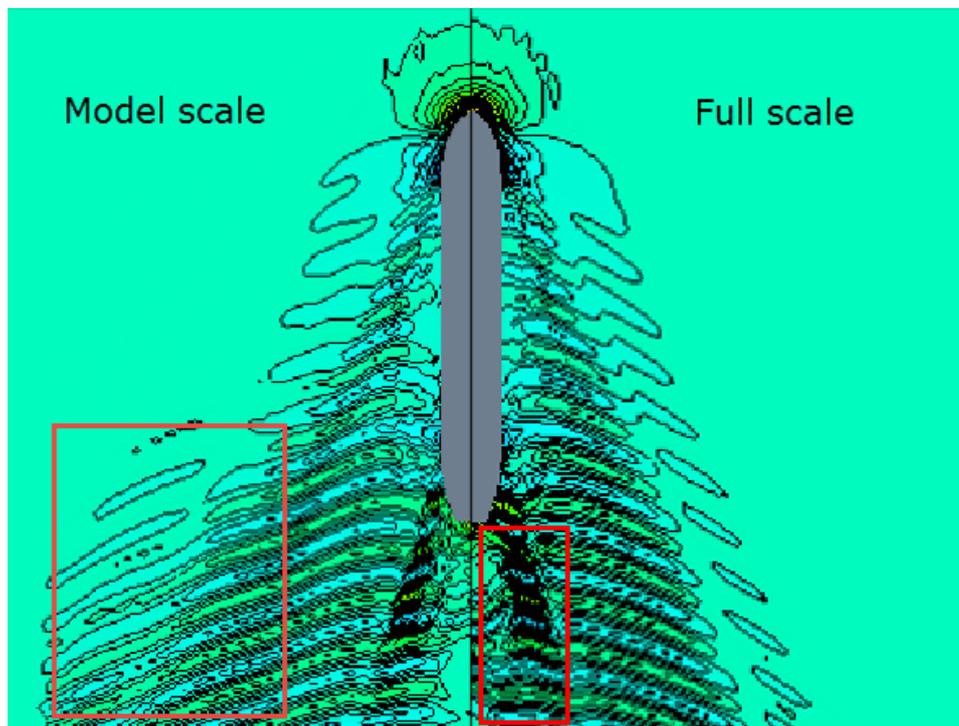


Figure5. 21. Comparison of wave patterns

Figure 5.21 shows the comparison of wave patterns between model scale and full scale ship. From figure 5.20 we have known that the degree of the wave pattern is nearly seen. In figure 5.21 we can see this phenomenon more intuitive. The shape and the wave length is also the same between model scale and full scale.

The difference come from free wave pattern region and turbulent wake region, which has been marked by using red blocks in figure 5.21. The difference in turbulent wake region is because the reduction of a large flow separation and vortex formation domain at the aftship. In wake field part we also can see the difference between model and full scale. The difference happens in free wave pattern region is we will discuss in discussion part 6.3.

5.8 Wake Field

5.8.1 Comparison of wake field on the propeller plane

Nominal wake is ship velocities deduction happened in the place that behind the ship hull and in front of the propeller. The flow around the ship and in front of propeller are affected by the presence of ship hull, the potential and viscous nature of the boundary layer around the ship contribute to the development of the wake, and the result is: the advanced speed of the water through the propeller plane is usually less than the ship speed. In general, naval architects always expect huge number of the wake fraction because if helpful for increasing the ship propulsion efficiency.

Figure 5.22 and 5.23 shows the wake field on the propeller plane at the aftship of model scale and full scale respectively. The cases choose are “Model scale _ Fr =0.1423” and “Full scale _ Fr = 0.1423”.

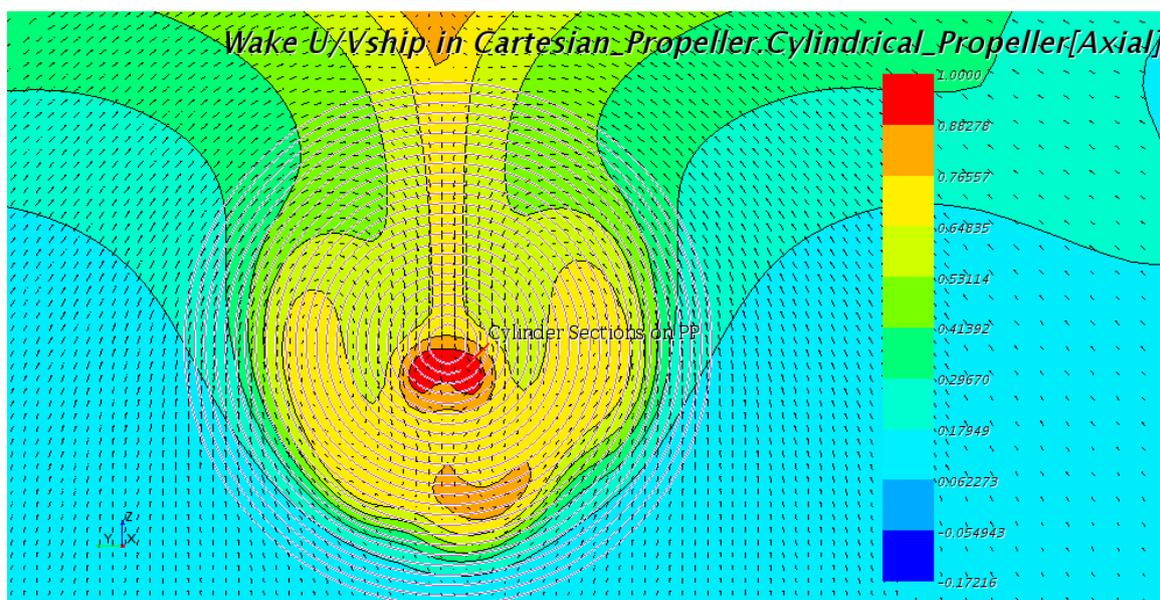


Figure5. 22. The wake field on the propeller plane of model scale ship (Fr = 0.1423)

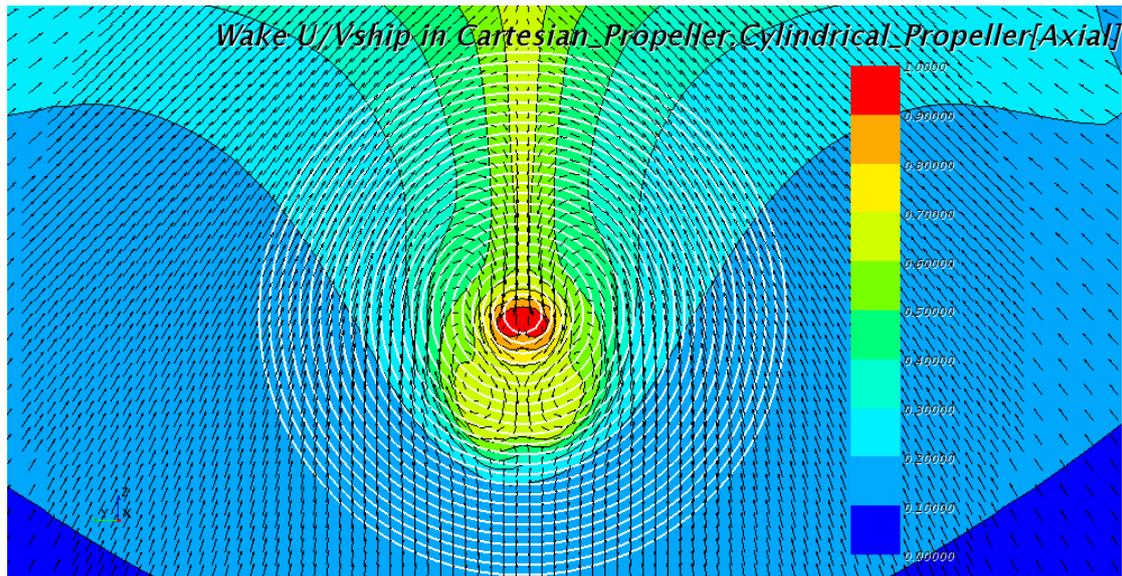


Figure5. 23. The wake field on the propeller plane of full scale ship ($Fr = 0.1423$)

From the picture, we can find the huge difference between the model scale and full scale ship. For model scale ship, the wake field almost occupied the whole propeller plane, and more than half proportion of the wake fraction is larger than 0.5. In fact this value is not actually happening in the real ship, for single screw merchant ship, the wake fraction's range always is 0.2 – 0.3. In the figure 5.23, for full scale ship, only small region has large wake numbers in the middle of the propeller plane, and the wake field of full scale ship has a much smaller wake fraction.

In addition, another information we can capture from figure 5.22 and 5.23 is: the arrow in the picture, which shows the tangential velocity of the water flow, this information describe how the scale effect happened in wake field, which we will discuss in chapter 6.2.

5.8.2 The mean value of nominal wake fraction

In Star CCM+, we can export the advanced flow velocity on the measure plane, shown in figure 5.24.

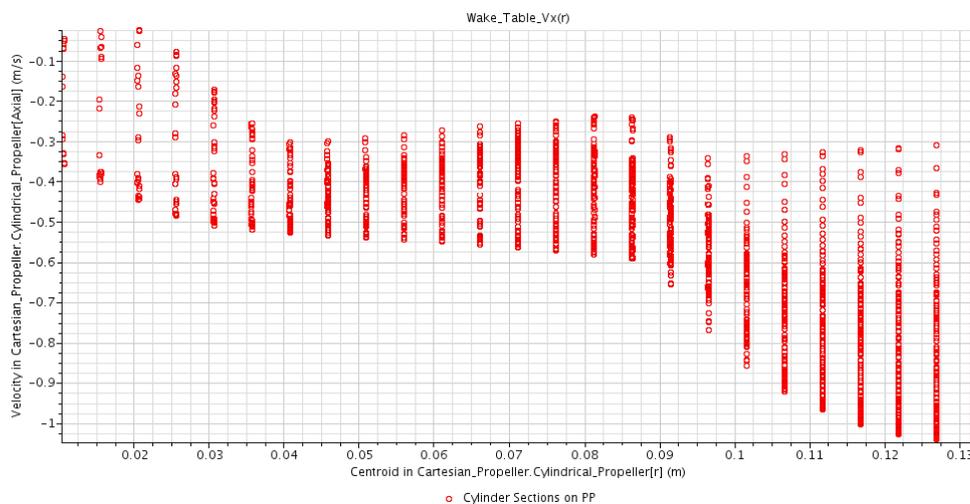


Figure5. 24. Velocities on the propeller

The x – axis of the plot in figure 5.24 means the radius of propeller. By observe this plot can learn the distribution of the wake on the propeller. Then we collect the data on this plot. Average nominal wake fraction is the mean value of the wake fraction of these data.

Speed (knots)	8	10	12	14	14.5	15	17
Fr	0.0785	0.0981	0.1178	0.1374	0.1423	0.1472	0.1669
Model scale	0.6736	0.5658	0.535	0.5157	0.5194	0.5145	0.5089
Full scale	0.316	0.314	0.312	0.3099	0.31	0.31	0.3048

Table5. 9. Average wake fraction

Table 5.9 indicates the mean wake fraction value, and the plots of these data presented in figure 5.25.

The conclusion are:

1. There is a huge scale effect of wake fraction number of model scale and full scale ship. The average wake fraction of model scale ship is much bigger than the full scale ship.
2. With increasing Fr number, the wake fraction is decreasing for both model and full scale ship.

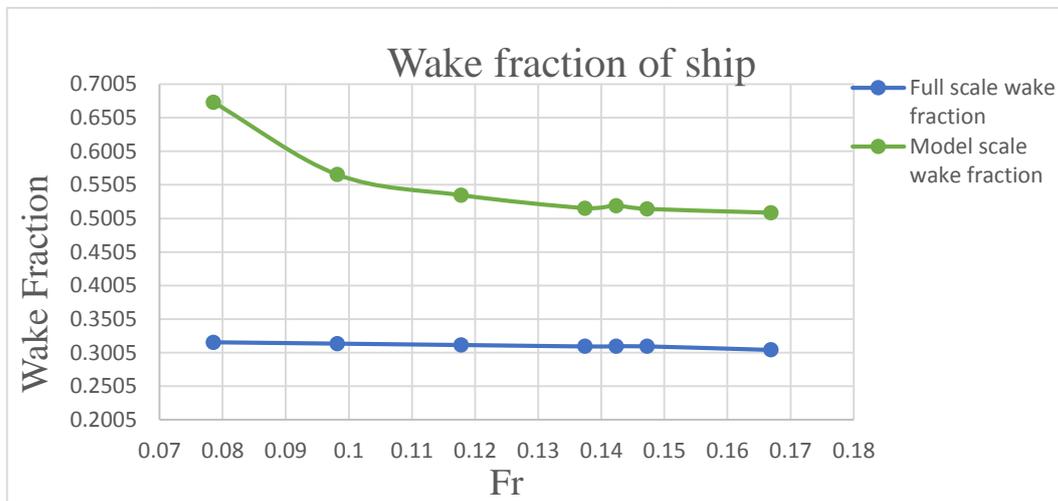


Figure5. 25 Mean value of wake fraction for all cases

6 DISCUSSION

6.1 *C_p influenced by bulbous bow*

In this part, we discuss the trend of resistance coefficient. In previous chapter, we notice that the friction resistance coefficients curve are quite fit what it should be look like: with the Re increasing, the Cf number decreased smoothly. But for C_p curve, the situation become different. Shown in figure 6.1

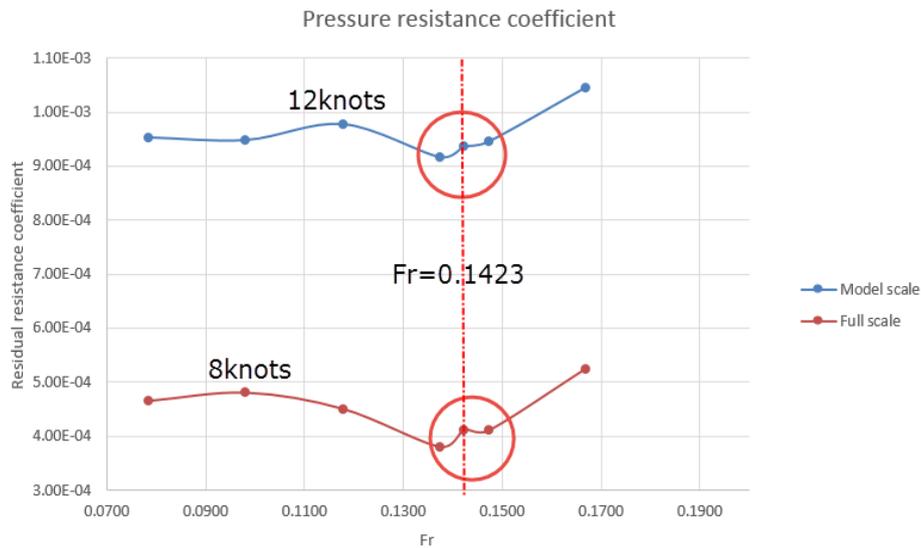


Figure6. 1. Pressure resistance coefficient

From figure 6.1, we can find that, at the beginning of the C_p curve, the pressure resistance coefficient go up and then reach the peak of the highest place: 8 knots for full scale and 12 knots for model scale; And then, the C_p number start to sharp drop until get the lowest point around the design speed (Fr = 0.1423); After this, the C_p number start to raise once again until the largest speed of the ship.

In fact, this trend of the C_p curve is fit our knowledge about the pressure resistance coefficient. When a ship has a bulbous bow, the real pressure resistance of the ship is going up and down again and again with the speed increased, which can be indicated in figure 6.2.

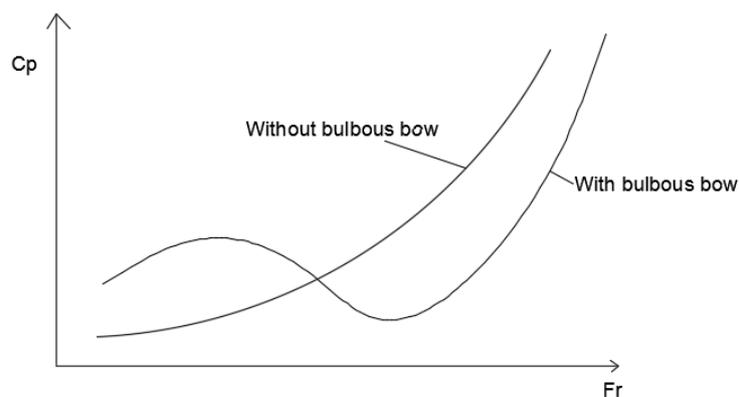


Figure6. 2. The curve of the C_p with/without bulbous bow

A bulbous bow is a protruding bulb at the bow of a ship at the underwater part. The bulbous bow modifies the way the water flows around the hull, reducing drag and thus increasing speed, range, fuel efficiency, and stability. [16]

This is no surprise that the naval architects set 14.5 knots as the design velocity: total resistance equals the sum of the C_f and C_p . C_f value is decreasing with the speed increasing, and the C_p number get the lowest value around $Fr = 0.1423$. From figure 5.15, we can clearly see that the JBC ship get the smallest total resistance coefficient when the ship is sailing in 14.5 knots.

6.2 The causes of scale effects

While the difference between friction resistances of model scale simulations and full scale simulations are suitable with our ship dynamics knowledge, the scale effect of pressure resistance coefficient C_p is far from our expected. In this part, we want to discuss the reason that influences the pressure coefficient and how the scale effect bring about.

Boundary Layer & Friction resistance coefficient

A boundary layer is the layer of fluid in the immediate vicinity of the ship surface where the effects of viscosity are significant. The thickness of the boundary layer is generally defined as approximately 99% of the vertical distance from the outer flow velocity at the surface of the object, which is shown in figure 6.3a. Only the water in the boundary layer consider the viscosity between water and ship surface. When the Reynolds number increased, the boundary layer become thinner, which we can see from equation 6.1, which describe the thickness of the boundary layer for turbulent boundary layers:

$$\delta \approx 0.382x / Re_x^{1/5} \quad (6.1)$$

Where x is the distance downstream from the start of the boundary layer; Re is the Reynolds number. When the boundary layer become thinner, the number of water that need to consider the viscosity is also reduced, and the friction resistance coefficient is decreasing at the same time.

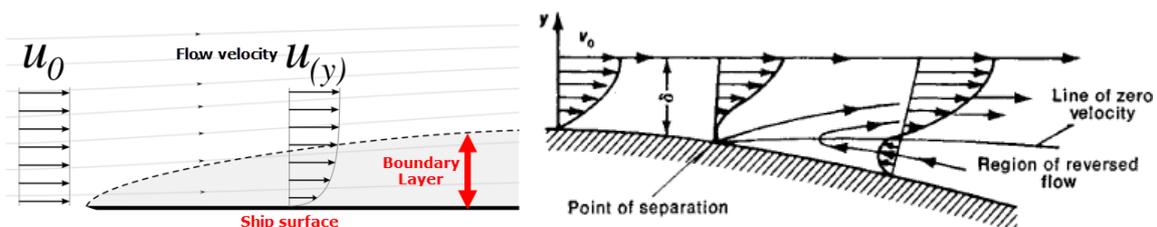


Figure 6. 3. a) Boundary layer b) Flow separation

Flow separation & Pressure resistance coefficient

When velocity of fluid flow increased or the flow encounter the sharp mutations at ship hull surface, because of the reversed flow, the fluid flow start to separate from the ship-hull surface (water will not flow along the shape of ship surface), and these separated flow generate vortex at aftship, which can be seen in figure 6.3b. These separate flow and vortex will cause the pressure difference between foreship and aftship, what means, the pressure resistance coefficient C_p is increased due to the flow separation.

Scale Effects

We compare the simulation between model scale and full scale, the Re number of full scale ship is much higher than model scale. As we mentioned before, the boundary layer is thinner in full scale ship, and has a smaller C_f number compare with the model scale.

Also, because the boundary layer is thinner than the model scale, the flow separation is generally delayed in full scale and vortices are much weaker. This phenomenon can be observed in figure 6.4.

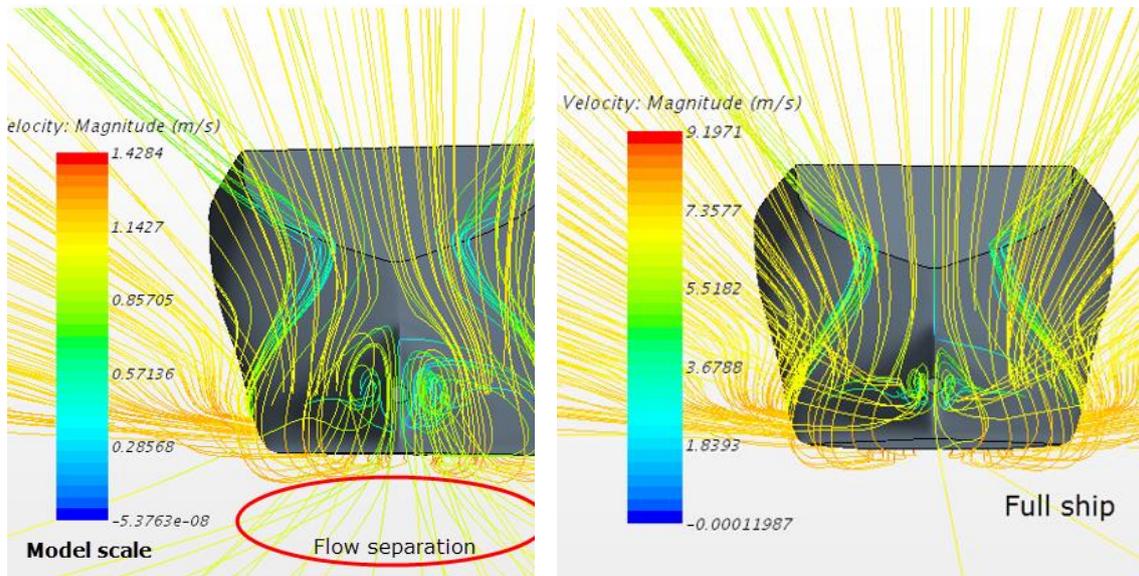


Figure6. 4. Streamline at aftship of model and full scale ship ($Fr = 0.1423$)

Figure 6.4 indicate that, for model scale ship, the water flow has a larger flow separation after ship stern, the flow from ship bottom go straight line directly but not along the ship surface. Compare with it, full scale ship's flow is along the stern surface. So we can make a conclusion here that because the full scale ship has less flow separation and vortex, the pressure resistance coefficient is of course smaller than the model scale ship. What we observed is 50% nearly.

Scale effect of the wake fraction is caused by same reason that the boundary layer flow separation. We are focussing on the wake field of figure 5.22 and 5.23 again and see the arrows in the figure. These arrows represent tangential velocity of the water flow. We find that, because the flow separation, the flow in model scale has very small tangential velocity that go straight line and flow to after ship directly. Not very much water flow to the propeller plane along the ship surface. In opposite, most the flows in full scale flow to propeller plane, a larger velocity compare to model scale of flow at full scale aftship. This is why the wake fraction of full scale ship is smaller than the model scale.

Conclusion

1. Compare to the model scale simulations, the Reynolds number of full scale ships are larger;
2. Boundary layer of full scale ship is thinner than model scale ship; Which cause friction resistance coefficient C_f lower than model scale;
3. Flow separation is generally delayed in full scale and vortices are much weaker than model scale.
4. C_p of full scale ship is smaller than model scale ship
5. Wake fraction is much smaller than model scale ship because less vortices and flow separation

6.3 Results Influenced by mesh & physical parameters

6.3.1 Friction resistance coefficient & Prism layers

Before finally determine all the parameters setting for full scale ship simulation, we performed several failure simulation that gets the really weird results. One of them is the friction resistance coefficient.

In fact, this is the most stable variable in these simulations, the change is very slightly when setting is changed. Where the greatest impact is prism layers setting on this resistance, or we can say the $Y+$ number.

$Y+$ number should be kept in the range of 30 -100, or at least 30 -500 around ship hull in these simulations. This is the suggestion we get from the Norwegian Marine Technology Research Institute (MARINTEK) and test conclusion we make after series of attempts on the prism layers setting. Otherwise the friction resistance coefficient will be more different from the expectation numbers.

6.3.2 Pressure resistance coefficient & VOF Waves Damping length

C_p is quite sensitive with VOF Waves damping length. In fact we still not figure out why this should happen. Slightly change of this number will let the pressure resistance coefficient changed in large numbers. At last we accept the suggestion from “MarineTek” that use the distance between the ship Central Plane (CP) and the Side boundary. For model scale ship, we use 13 meters and for full scale ship we use 520 meters.

6.3.3 Waves & Parameters setting

For some simulation cases, such as “Model scale _ $k-\epsilon$ turbulence model, $Fr = 0.1423$ ” case and the cases of lower speed of ship. The wave patterns generated looks wired when the others results get good number. Shown in figure 6.5.

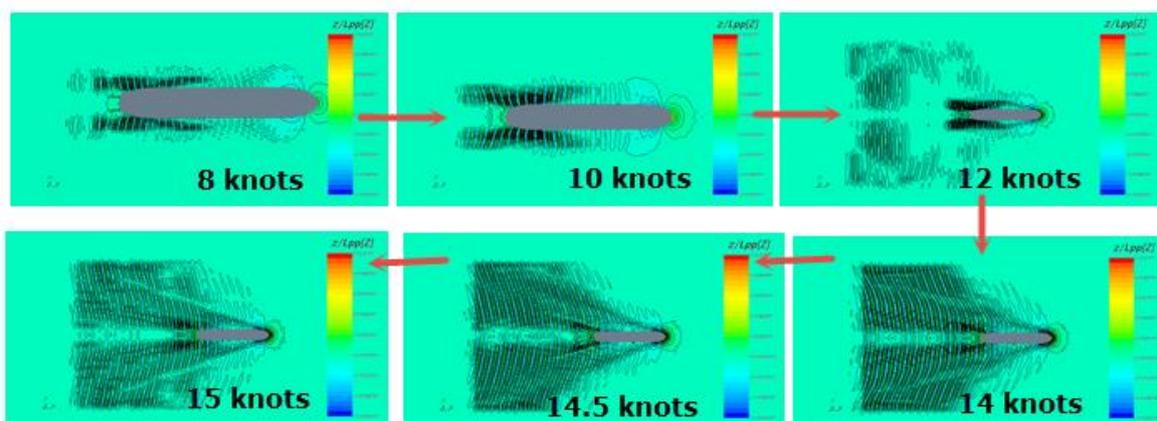


Figure6. 5. Wave patterns for different speed ships (Model scale)

From the figure we find that, with speed increasing, the wave “grows” gradually until it get a complete Kelvin ship waves pattern when ship sailing at 14 knots. We didn’t do more works to focus on this phenomenon, but we guess it should be caused by the bad physical and mesh settings, consists of:

1. The mesh size at the free surface region is not sufficient. The future work we should try more details mesh cells at the free surface region;
2. Time step is not small enough: even we keep the CFL number smaller than 1 at most of simulation region, but at the place that around the ship, the CFL is still larger than 1. The future work should try smaller time step number to simulate;
3. Turbulence models: by comparing the results difference between SST $k-\omega$ turbulence model and $k-\varepsilon$ turbulence model, we find that these parameters influence the wave patterns very much. Try more turbulence model in the future works to be seen the difference.

7 CONCLUSION

By simulate 17 different ship sailing model in Star CCM+, we basically grasp the hydrodynamic characteristics of this type Japan Bulk Carrier ship for both model and full scale, compare the results of them and try to figure out the causes of scale effects on a deeper level. In addition, we summarize some precautions for the mesh and parameters setting and simulation after experiencing several failures of simulations

During the mesh setting process, we should notice that the prism layers' setting is a quite important concerns that we need to try several times to find the best setting thickness to keep the Y^+ number around the underwater ship hull in the range of 30 -100. This number is actually can influence the accuracy of friction resistance coefficient.

“VOF waves damping length” is the parameter that impact the pressure resistance coefficient. We use the distance between the ship Central Plane (CP) and the Side boundary. On model scale ship, we use 13 meters and for the full scale ship we use 520 meters.

The scale effect for this type of ship is conspicuous, not only for the friction resistance coefficient C_f which is related to the Reynolds number, but also the pressure resistance coefficient C_p . For full scale ships, C_p number are 50% smaller than the model scale ship. The wake field we focus on in this master thesis, the differences are also huge.

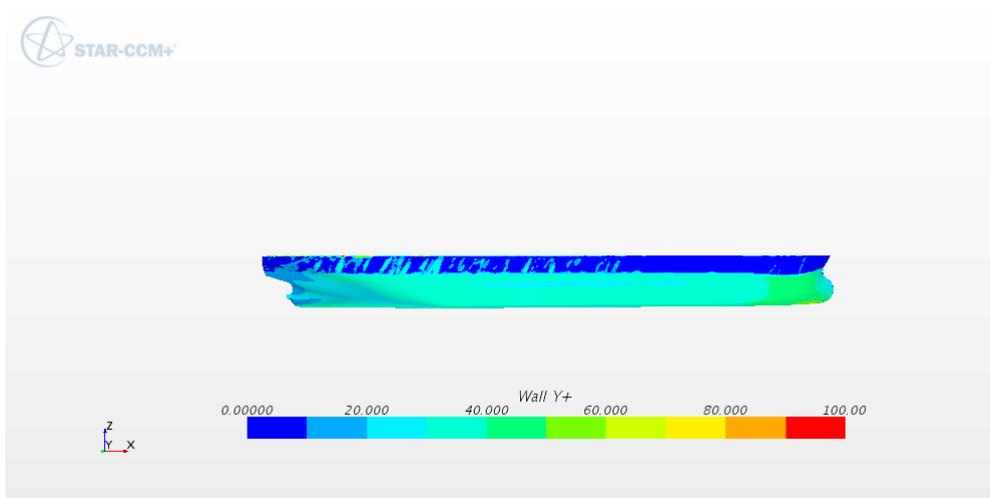
The underlying cause of the scale effect is the difference of Reynolds number between model scale simulations and full scale simulations. With increasing of Reynolds number, the boundary layer around ship surface become thinner which cause the friction resistance decreased. The thinner boundary layer cause a delayed flow separation and weaker vortex, and then, this will cause smaller C_p number of full scale ship than model scale ship. The delayed flow separation let more water flow to the propeller plane along the ship hull surface, which cause a smaller wake fraction of full scale ship than the model scale ship.

At last, we also can make the conclusion that because the big scale effect between model scale ship and full scale ship, we can't use the model towing test to measure the ship hydrodynamics of this type of ship directly. More simulation we need to be done to estimate the factor of the pressure resistance coefficient between model and full scale in the future work.

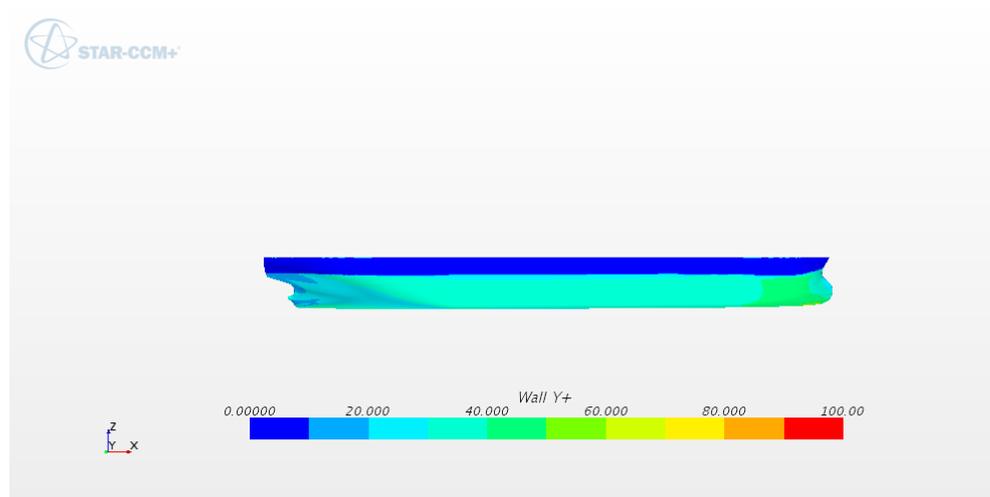
REFERENCES

- [1] *THE SCALE EFFECT ON NOMINAL WAKE FRACTION OF SINGLE SCREW SHIPS*, By Z. BENEDEK, Department of Hydraulic Machines, Polytechnical University, Budapest" (Received August 28, 1967) Presented by Prof. B. BALOGH
- [2] http://en.wikipedia.org/wiki/Computational_fluid_dynamics (September 2014)
- [3] *Experimental Fluid Dynamics and Uncertainly Assessment Methodology*, S.Ghosh, M.Muste, F.Stern, Page 3
- [4] *FIRST INTRODUCTION INTO COMPUTATIONAL FLUID DYNAMICS FOR MARINE APPLICAION*, Vladimir I. Krasilnikov
- [5] http://en.wikipedia.org/wiki/Ship_resistance_and_propulsion
- [6] *ITTC – Recommended Procedures and Guidelines, Testing and Extrapolation Methods, General Guidelines for Uncertainty Analysis in Resistance Towing Tank Tests*, Effective date: 2008
- [7] *Resistance & Propulsion MAR 2010, Presentaion of ships wake*, Rod Sampson – School of Marine Science and Technology
- [8] *The help interface of Star CCM+*
- [9] http://en.wikipedia.org/wiki/K-epsilon_turbulence_model
- [10] http://en.wikipedia.org/wiki/K%E2%80%93omega_turbulence_model
- [11] http://www.cfd-online.com/Wiki/SST_k-omega_model
- [12] http://en.wikipedia.org/wiki/Courant%E2%80%93Friedrichs%E2%80%93Lewy_condition
- [13] *UNIVERSITY OF LJUBLJANA FACULTY FOR MATHEMATICS AND PHYSICS DEPARTMENT FOR PHYSICS KELVIN SHIP WAVES*, Student: KSENJA MAVER Advisor: Professor RUDI PODGORNIK Ljubljana, 2004
- [14] http://en.wikipedia.org/wiki/Fluid_dynamics
- [15] http://www.t2015.nmri.go.jp/Instructions_JBC/Case_1.1a/Case_1-1a.html
- [16] http://en.wikipedia.org/wiki/Bulbous_bow
- [17] *BEST PRACTICE GUIDELINES FOR MARINE APPLICATIONS OF COMPUTATIONAL FLUID DYNAMICS*, Prepared by WS Atkins Consultants And members of the NSC
- [18] *On the Importance of Full-Scale CFD Simulations for Ships*, Karsten Hochkirch, FutureShip, Potsdam/Germany, Karsten Hochkirch, FutureShip, Potsdam/Germany
- [19] *Numerical simulation of propeller-hull interaction and determination of the effective wake field using a hybrid RANS-BEM approach*, Douwe Rijpkema, Bram Starke, Johan Bosschers, Maritime Research Institute Netherlands (MARIN), Wageningen, The Netherla
- [20] *PREDICTION OF EFFECTIVE WAKE AT MODEL AND FULL SCALE USING A RANS CODE WITH AN ACTUATOR DISK MODEL*, Antonio Sánchez-Caja and Jaakko V. Pylkkänen, VTT Technical Research Center of Finland

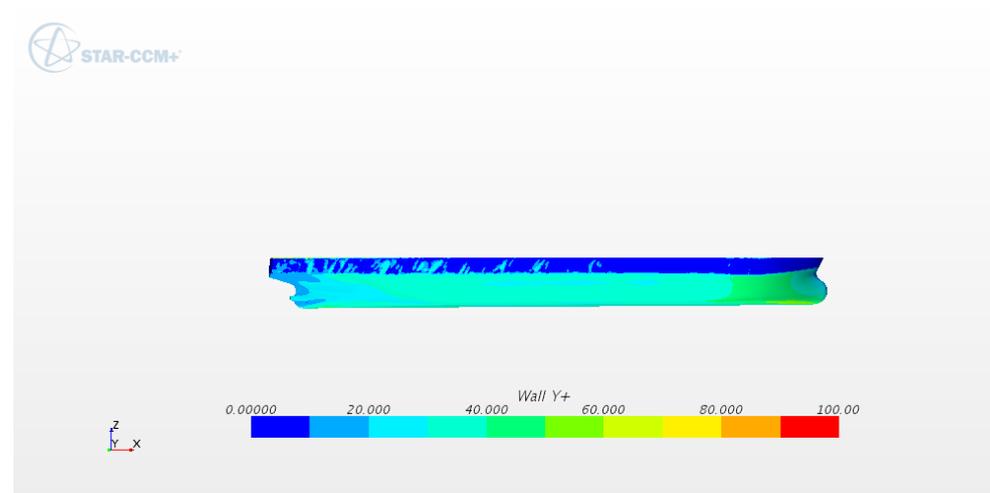
APPENDIX A1. Y⁺ NUMBER



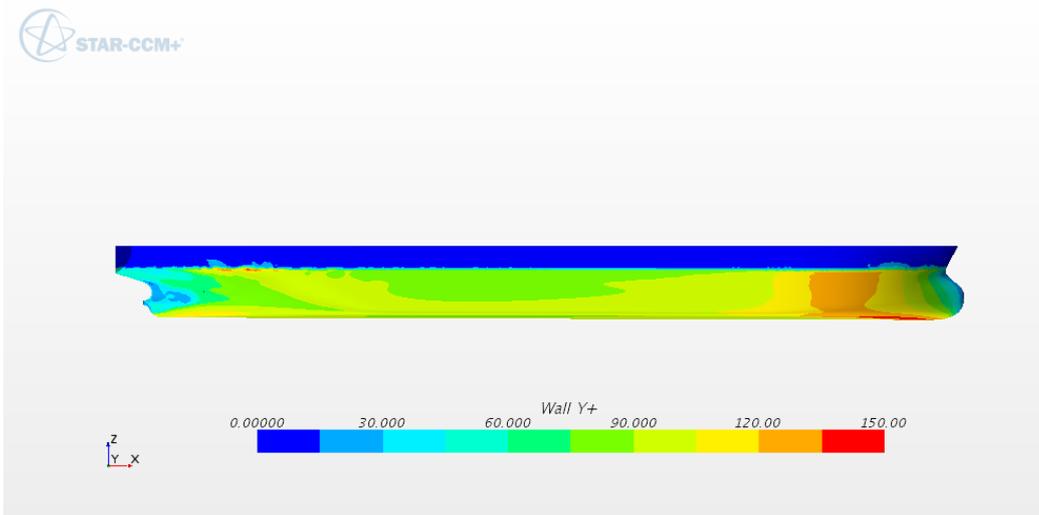
Model scale _ Fr = 0.1423 (14.5 knots)



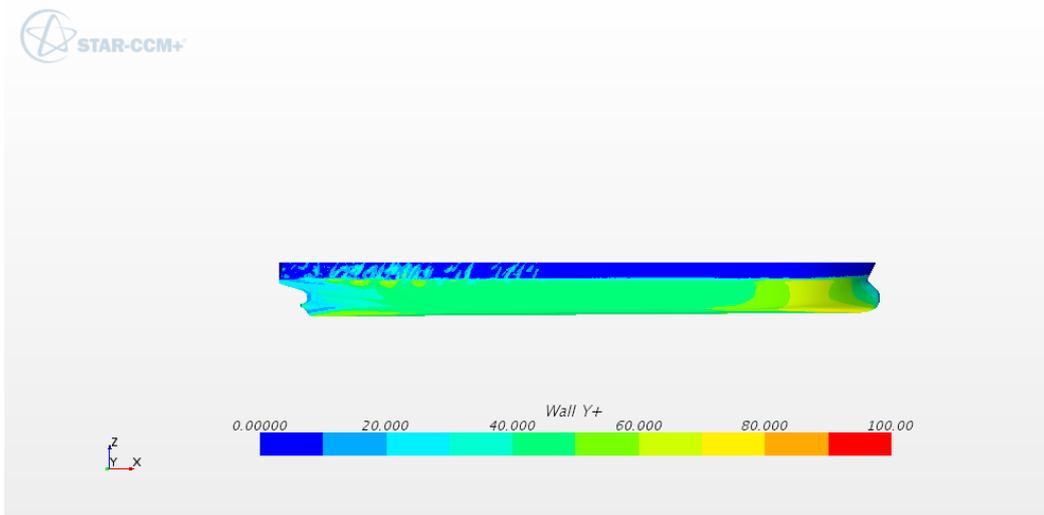
Model scale _ k-ε turbulence model _ Fr = 0.1423



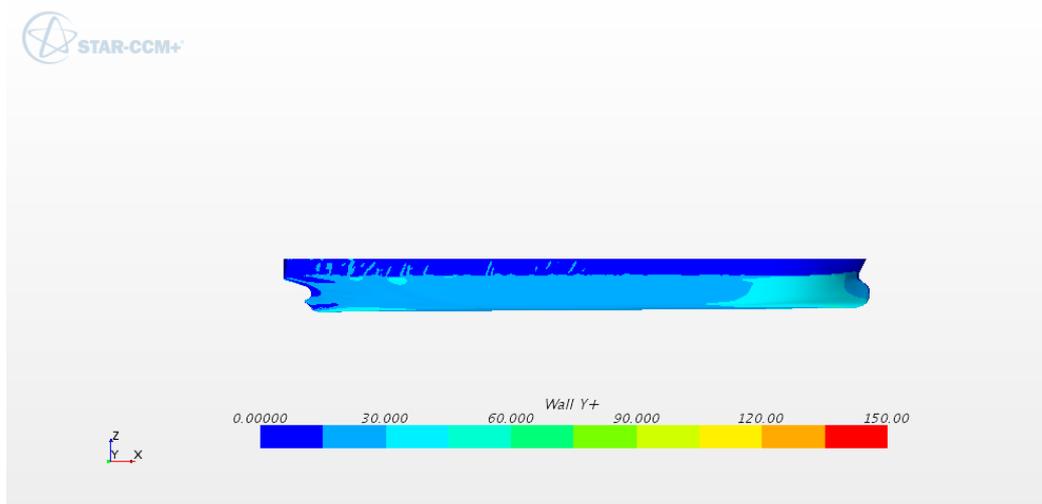
Model scale _ Half ship _ Fr = 0.1423



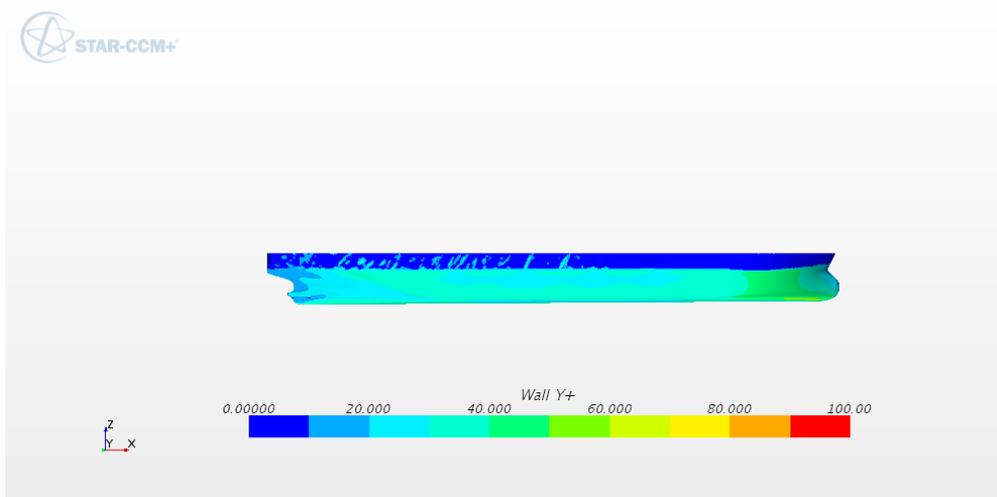
Model scale _ Fr = 0.0785 (8 knots)



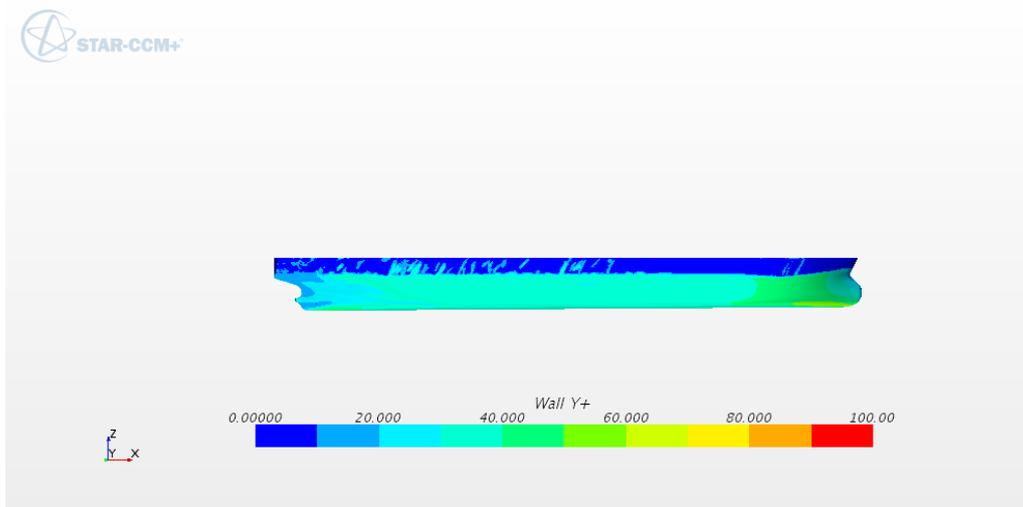
Model scale _ Fr = 0.0981 (10 knots)



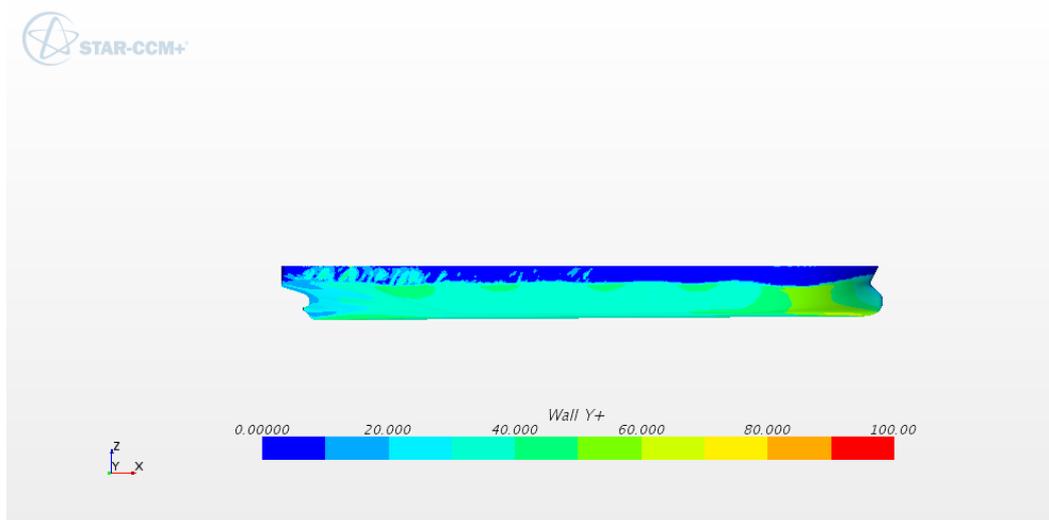
Model scale $_ Fr = 0.1178$ (12 knots)



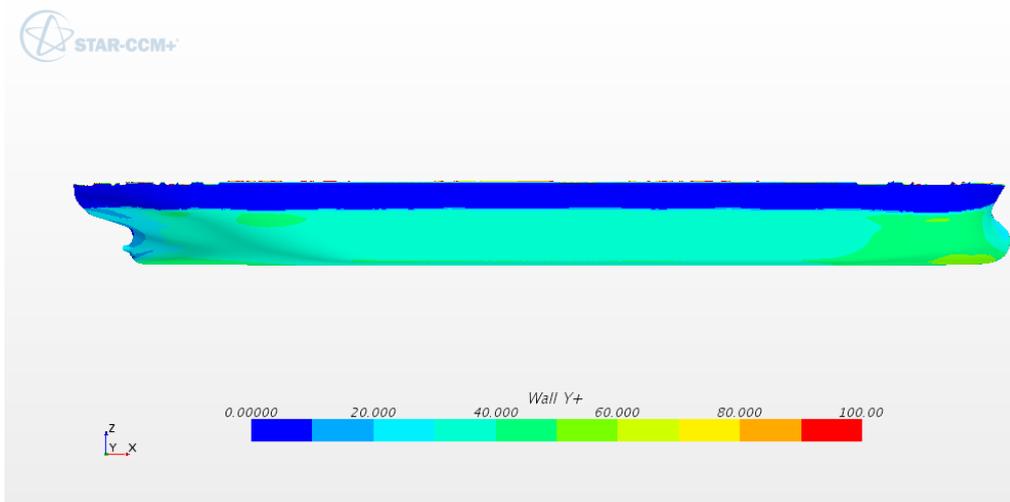
Model scale $_ Fr = 0.1374$ (14 knots)



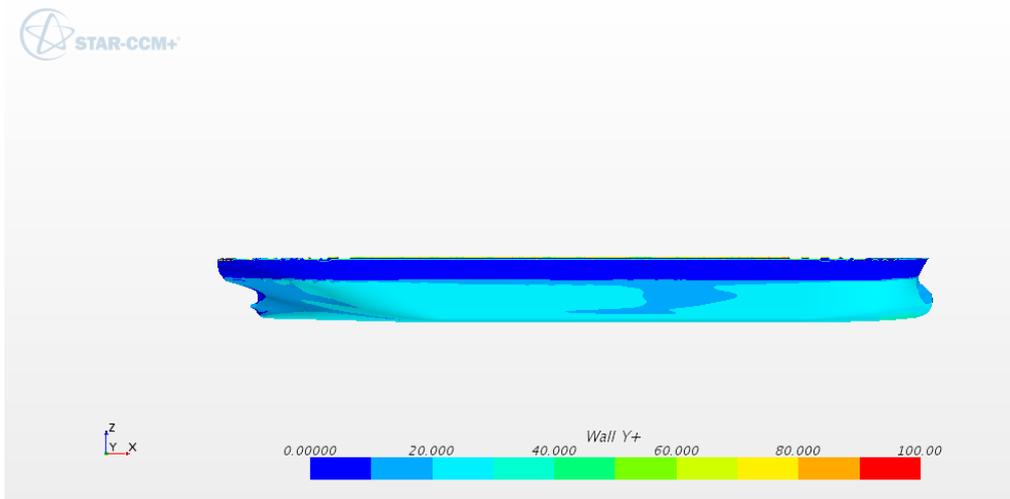
Model scale $_ Fr = 0.1472$ (15 knots)



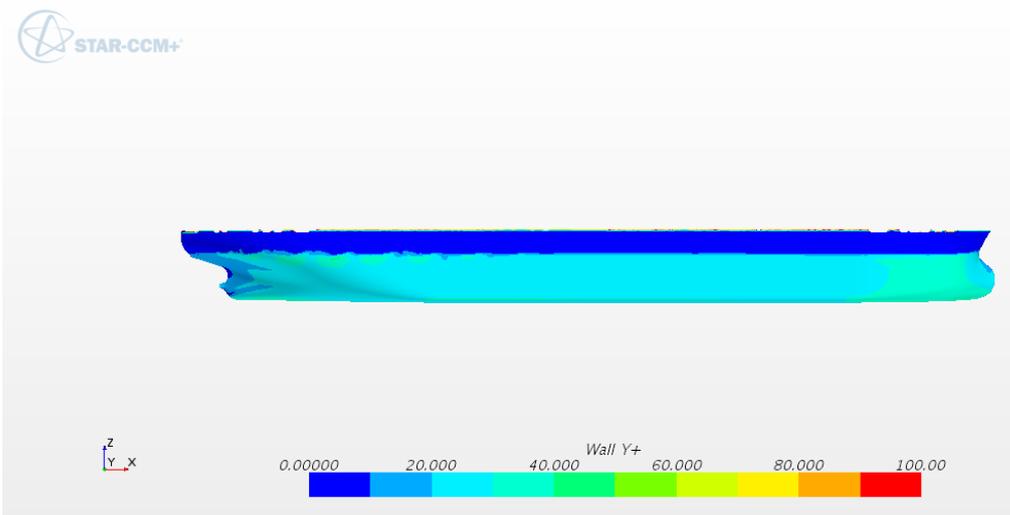
Model scale _ Fr = 0.1669 (17 knots)



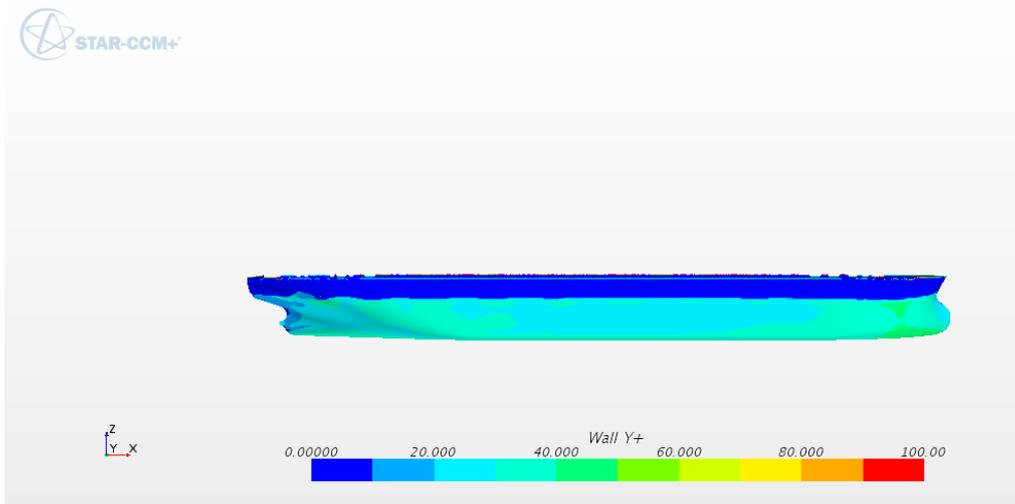
Full scale _ Fr = 0.1423 (14.5 knots)



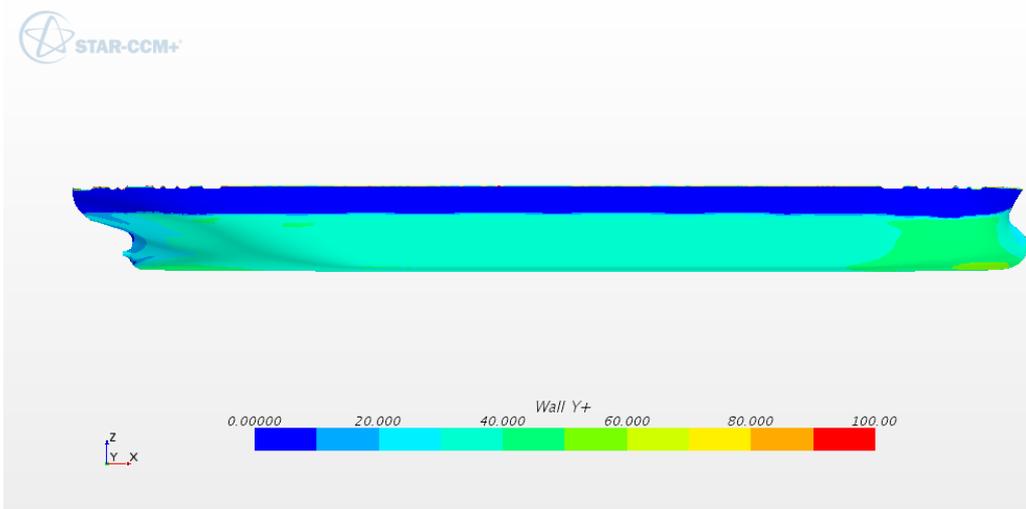
Full scale _ Fr = 0.0785 (8 knots)



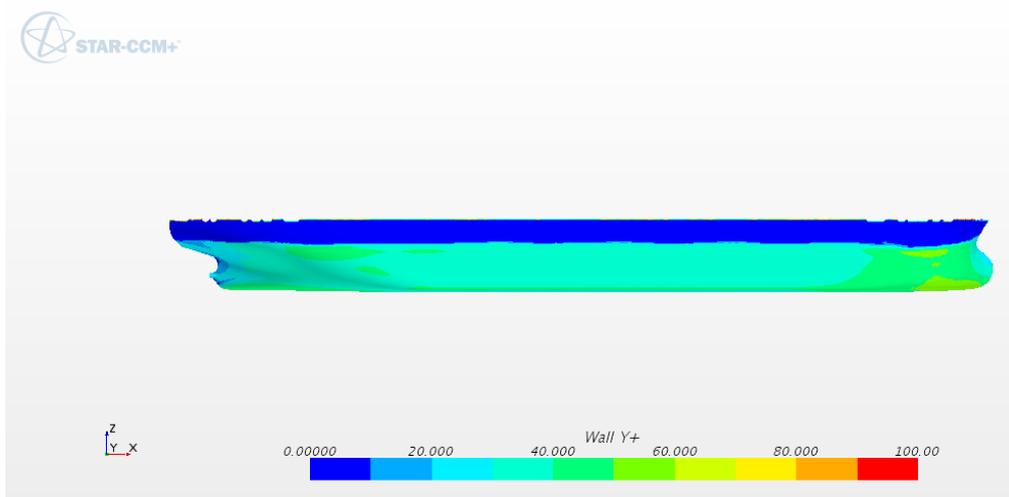
Full scale _ Fr = 0.0981 (10 knots)



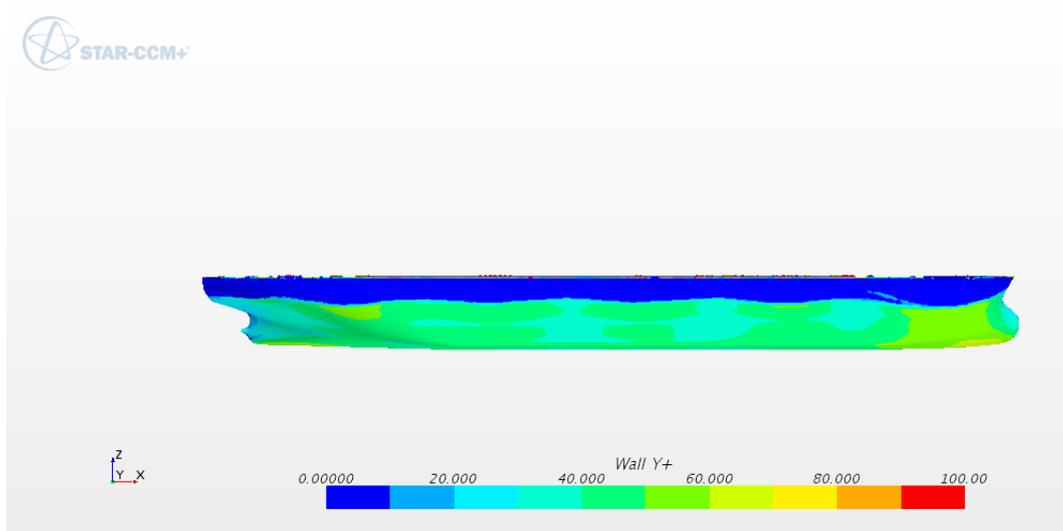
Full scale _ Fr = 0.1178 (12 knots)



Full scale _ Fr = 0.1374 (14 knots)

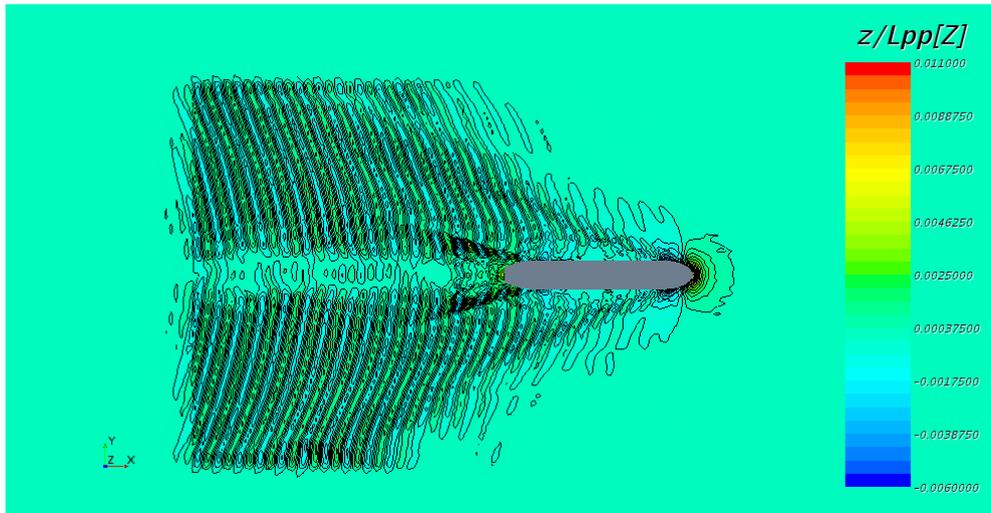


Full scale _ Fr = 0.1472 (15 knots)

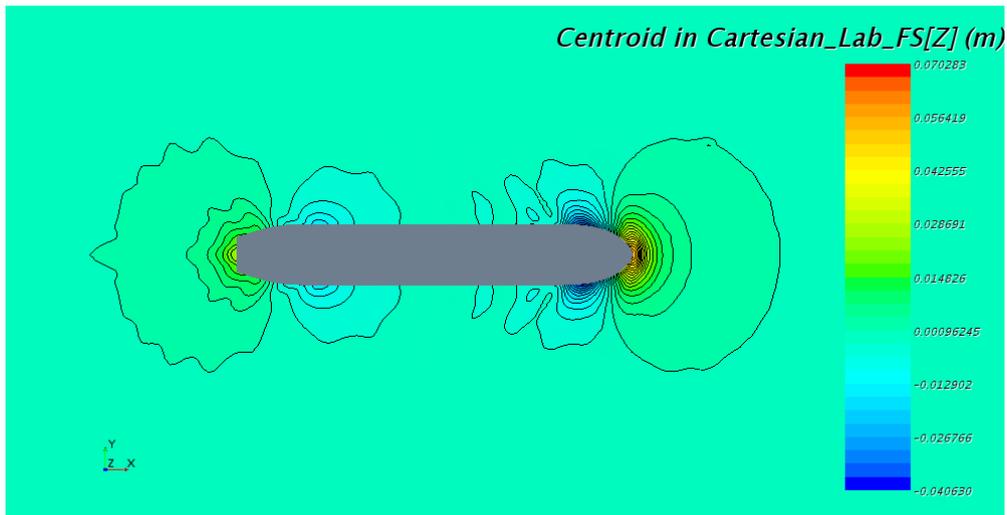


Full scale _ Fr = 0.1669 (17 knots)

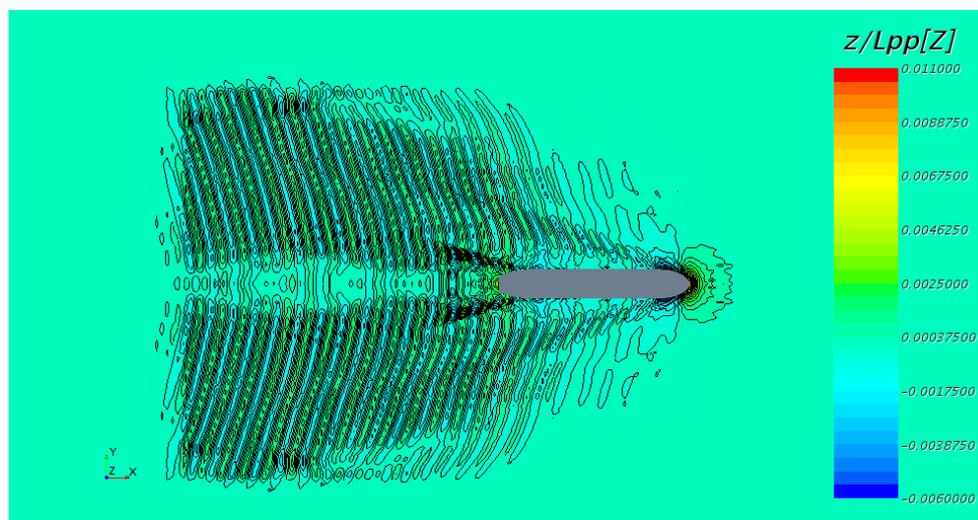
APPENDIX A2. WAVE PATTERNS



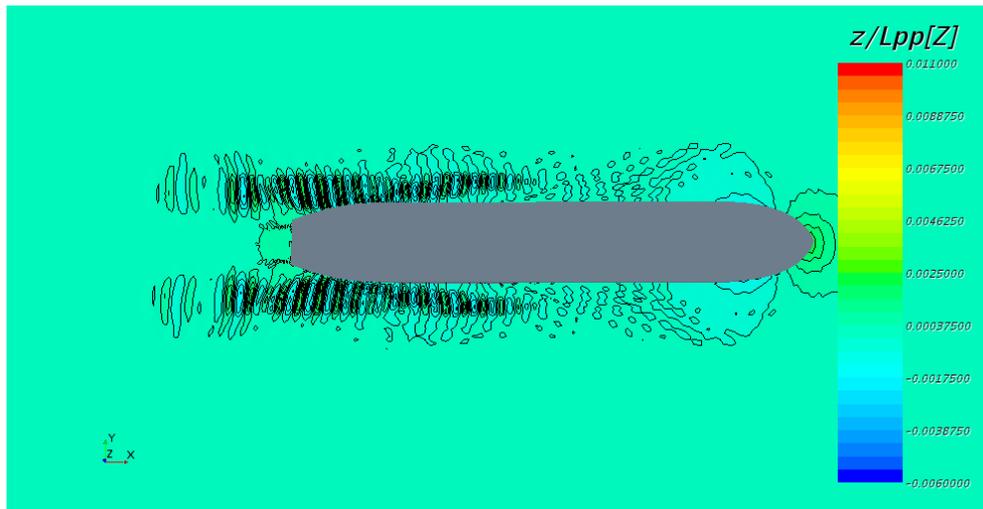
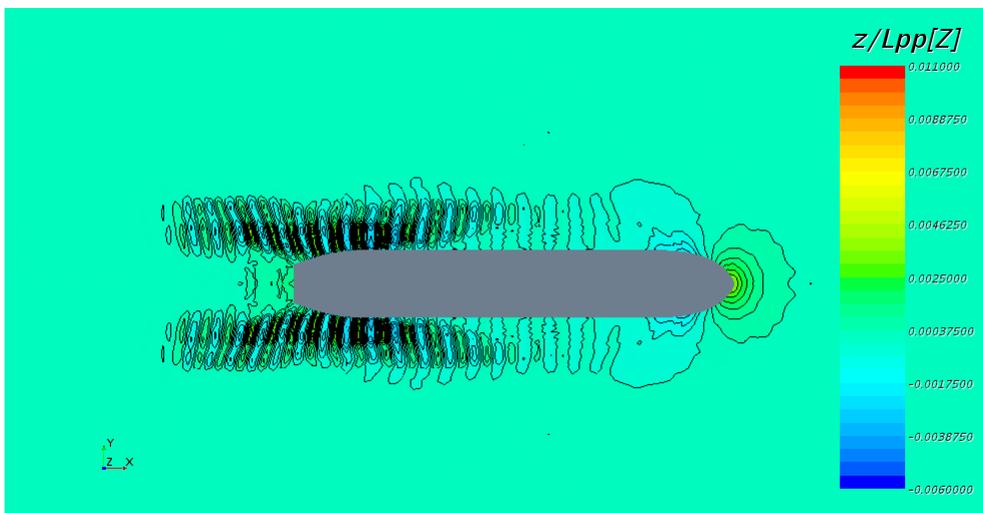
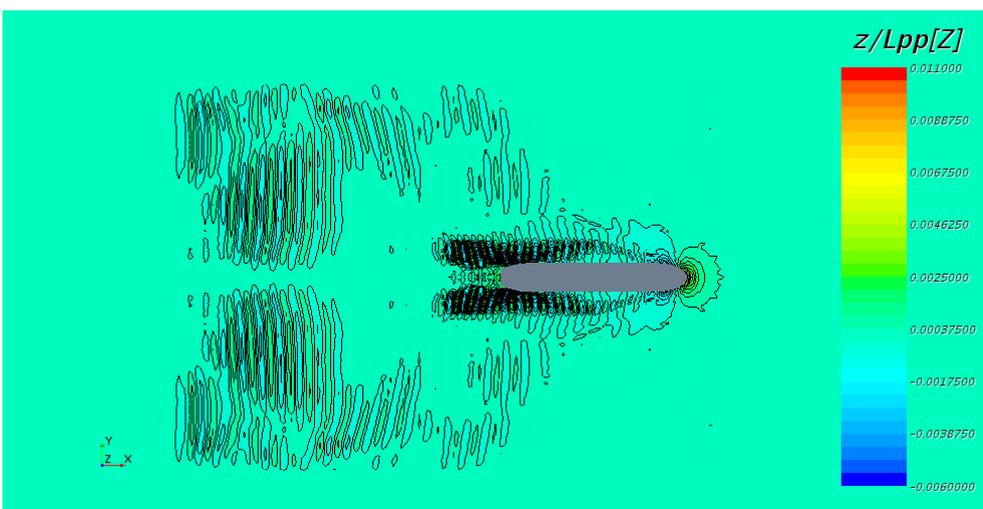
Model scale $_ Fr = 0.1423$ (14.5 knots)

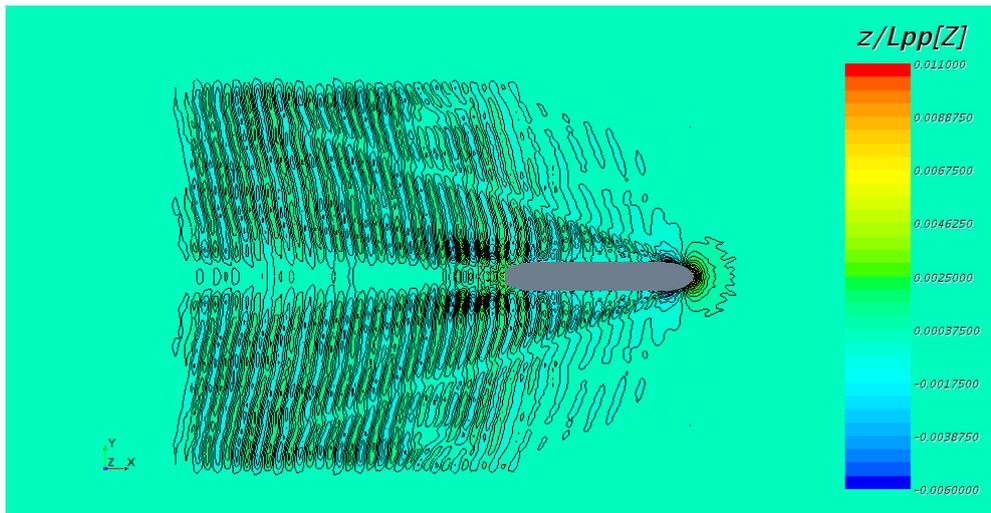
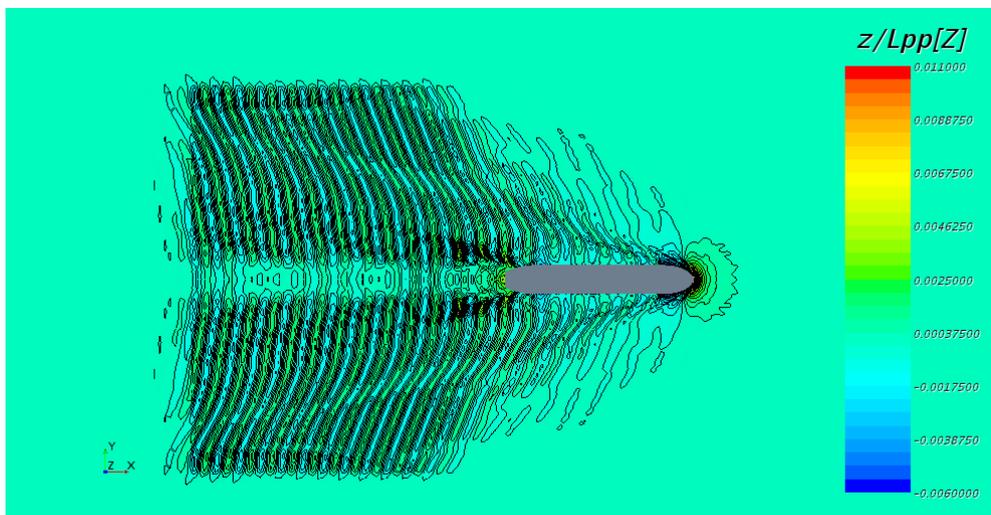
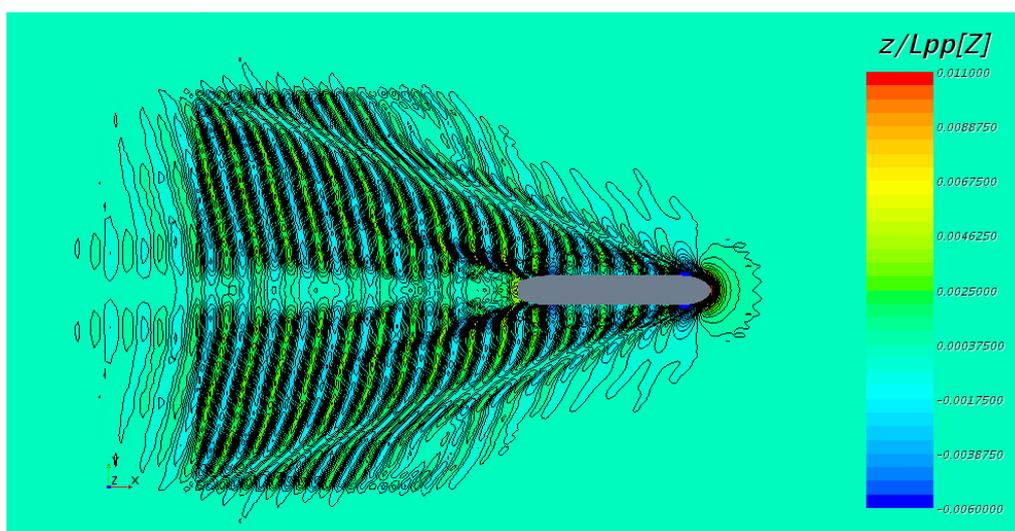


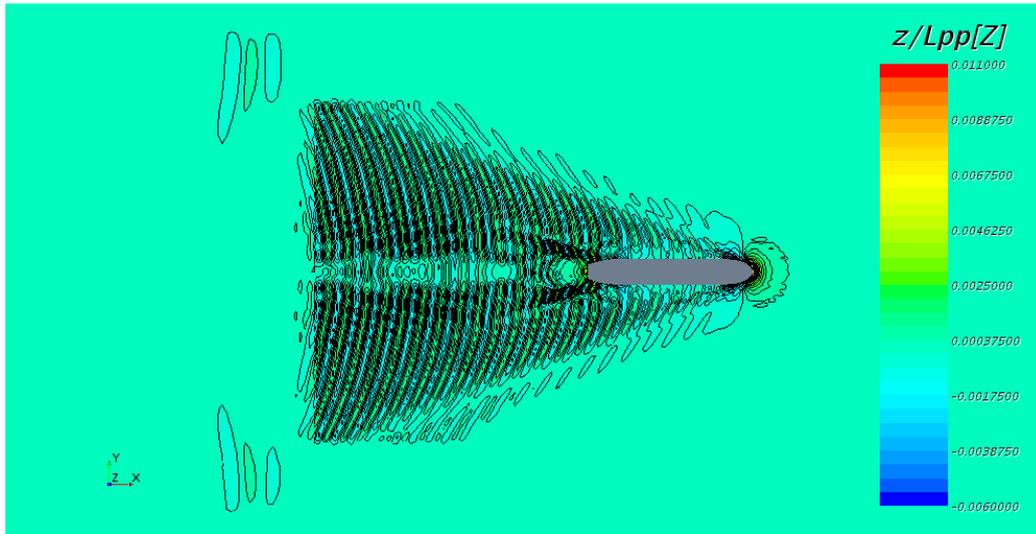
Model scale $_ k-\epsilon$ turbulence model $_ Fr = 0.1423$



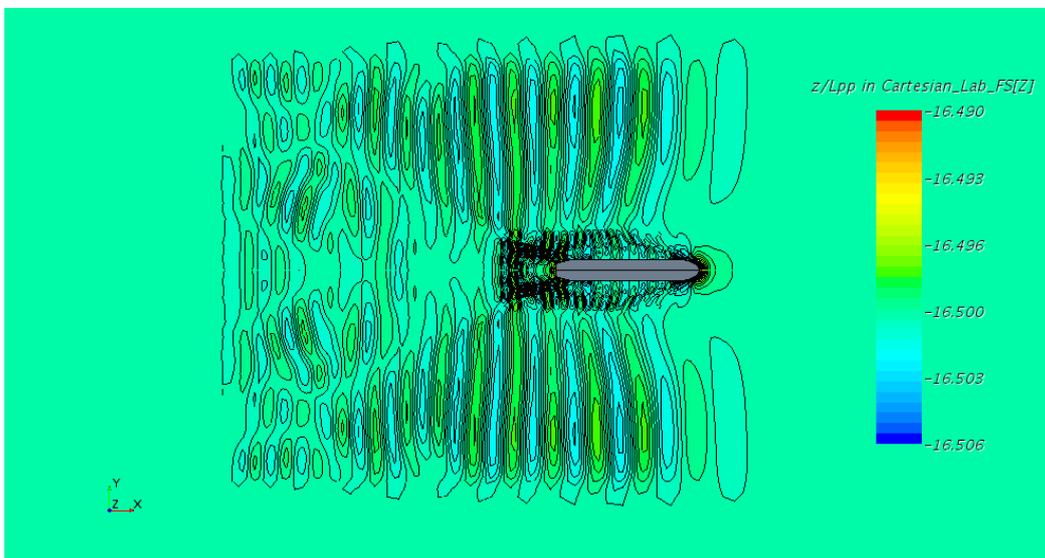
Model scale $_ Half ship$ $_ Fr = 0.1423$

Model scale _ $Fr = 0.0785$ (8 knots)Model scale _ $Fr = 0.0981$ (10 knots)Model scale _ $Fr = 0.1178$ (12 knots)

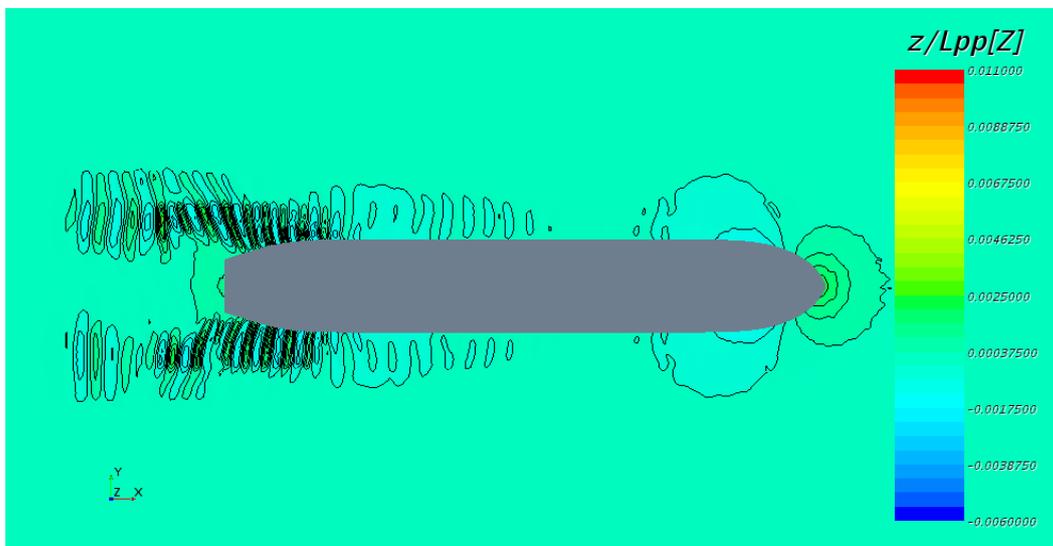
Model scale $_ Fr = 0.1374$ (14 knots)Model scale $_ Fr = 0.1472$ (15 knots)Model scale $_ Fr = 0.1669$ (17 knots)



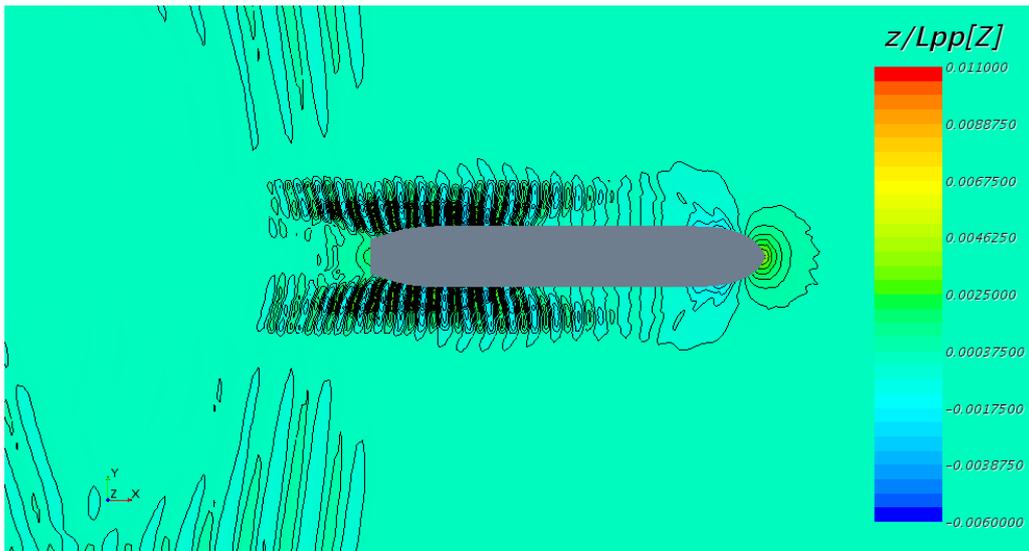
Full scale _ Fr = 0.1423 (14.5 knots)



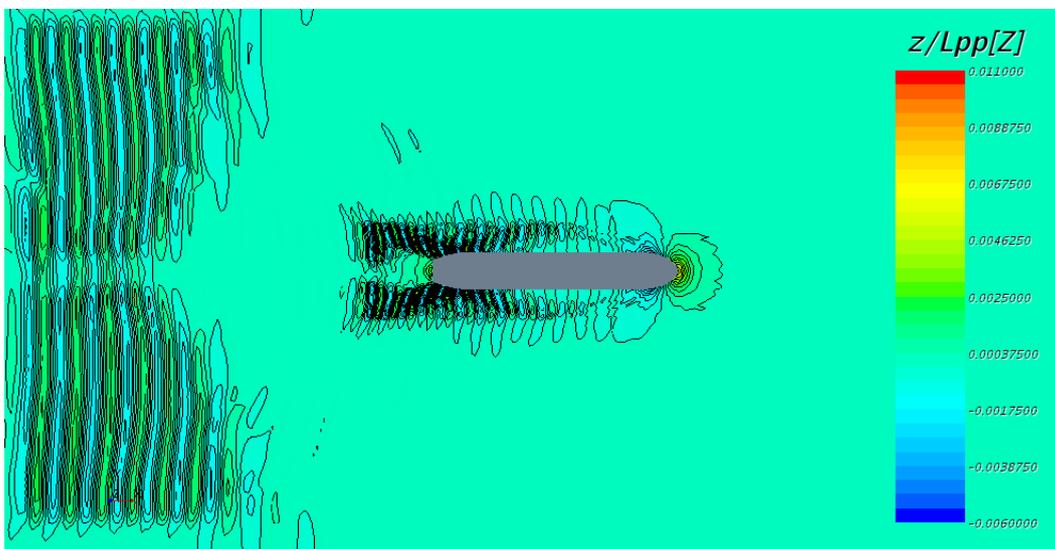
Full scale _ Half ship _ Fr = 0.1423



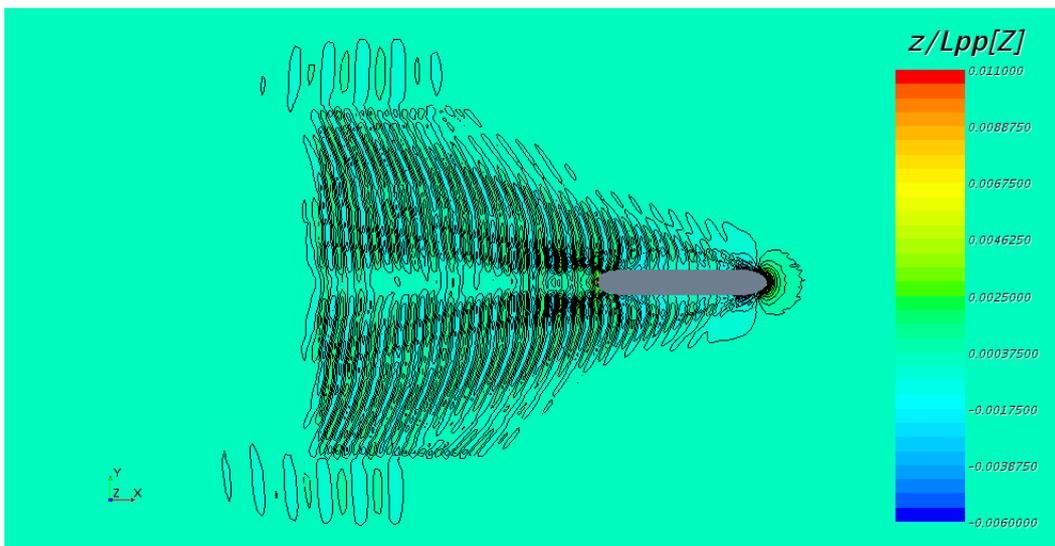
Full scale _ Fr = 0.0785 (8 knots)



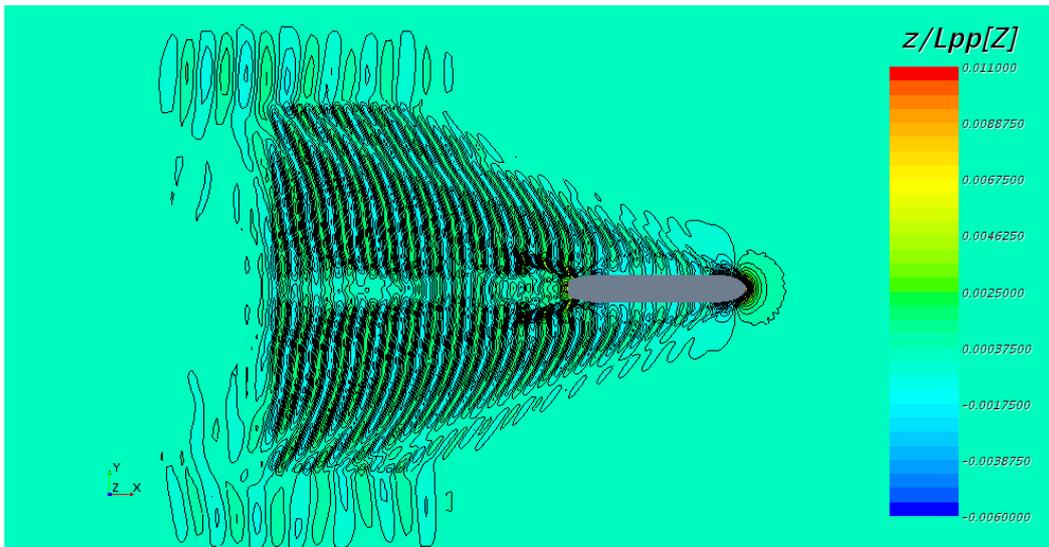
Full scale _ Fr = 0.0981 (10 knots)



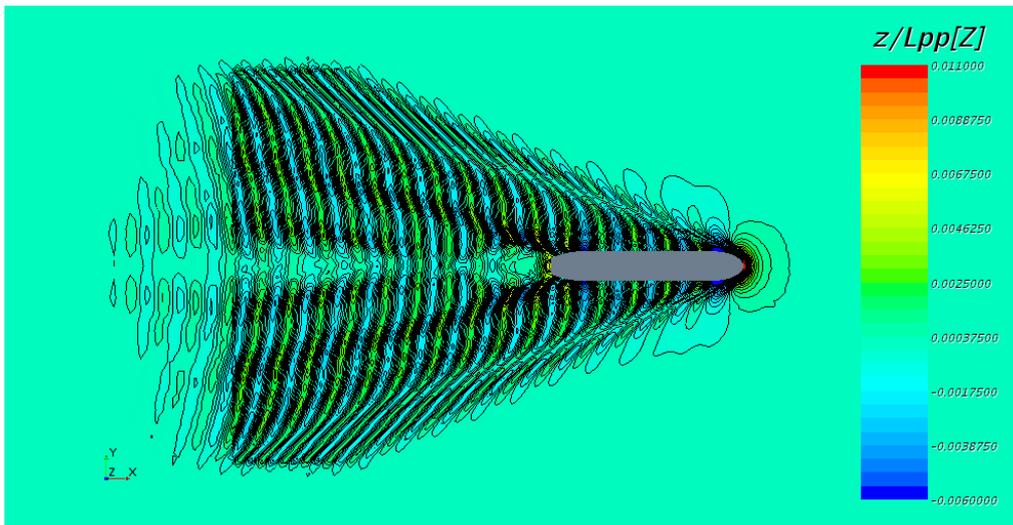
Full scale _ Fr = 0.1178 (12 knots)



Full scale _ Fr = 0.1374 (14 knots)

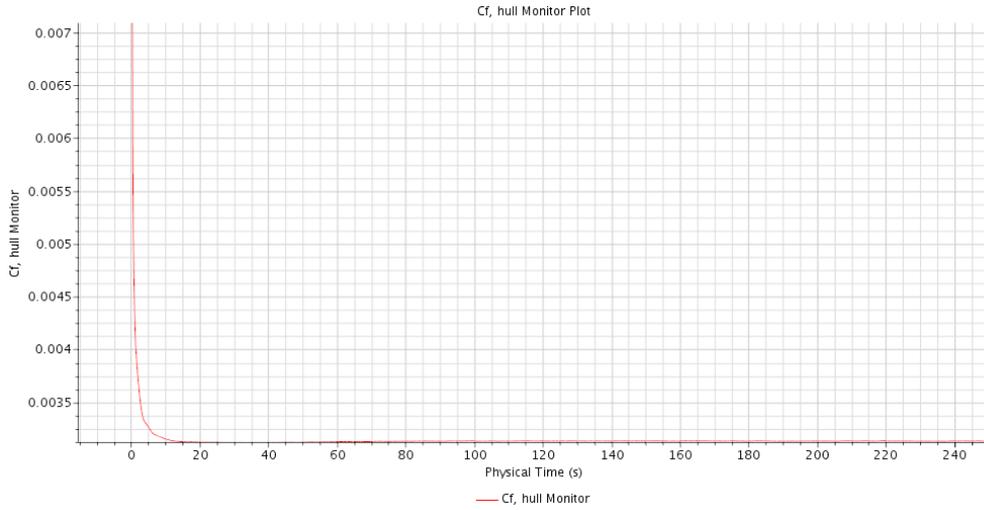


Full scale _ Fr = 0.1472 (15 knots)

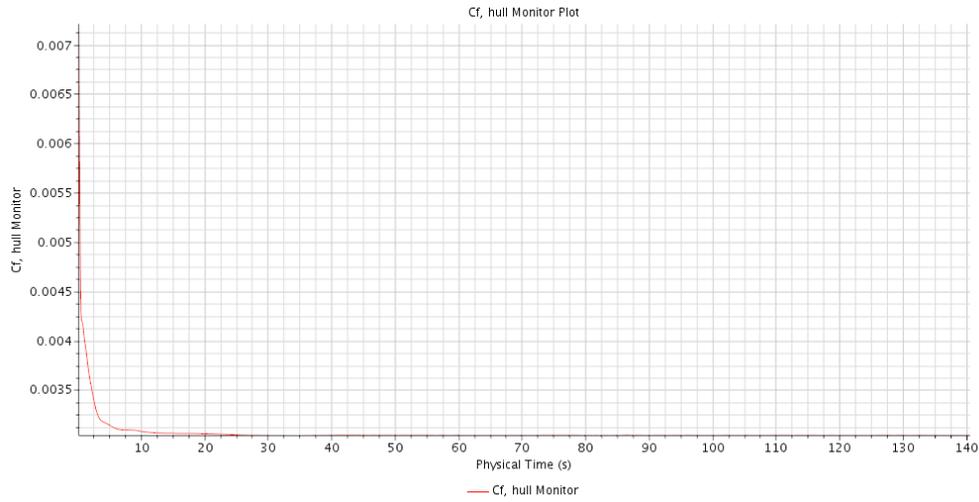


Full scale _ Fr = 0.1669 (17 knots)

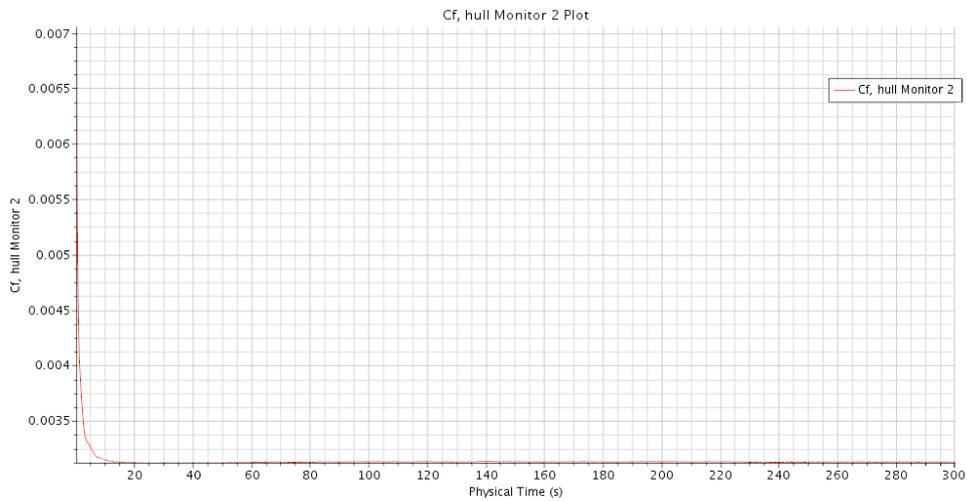
APPENDIX A3. CF



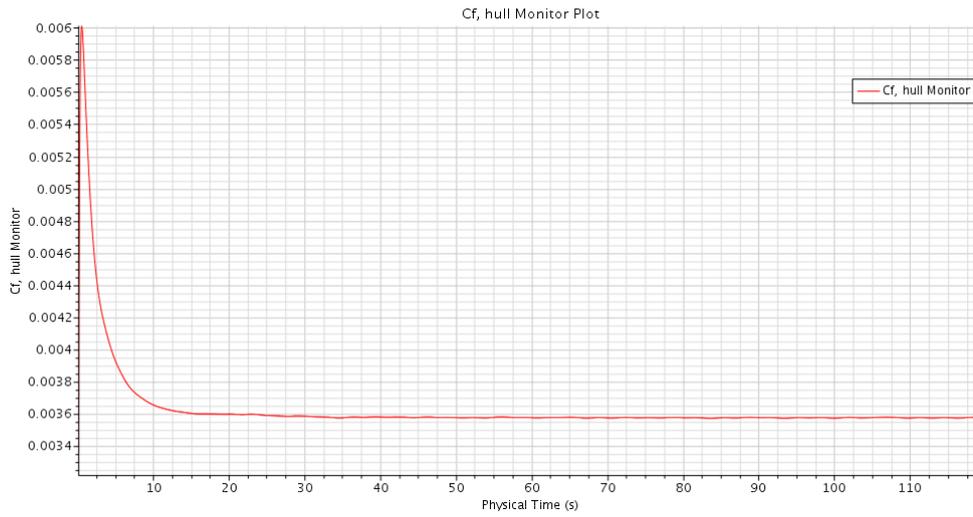
Model scale _ Fr = 0.1423 (14.5 knots)



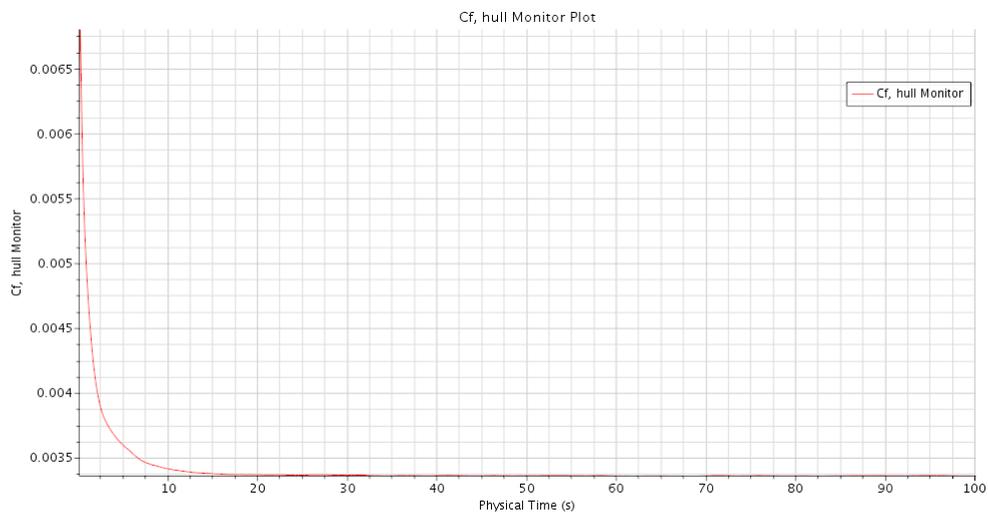
Model scale _ k-ε turbulence model _ Fr = 0.1423



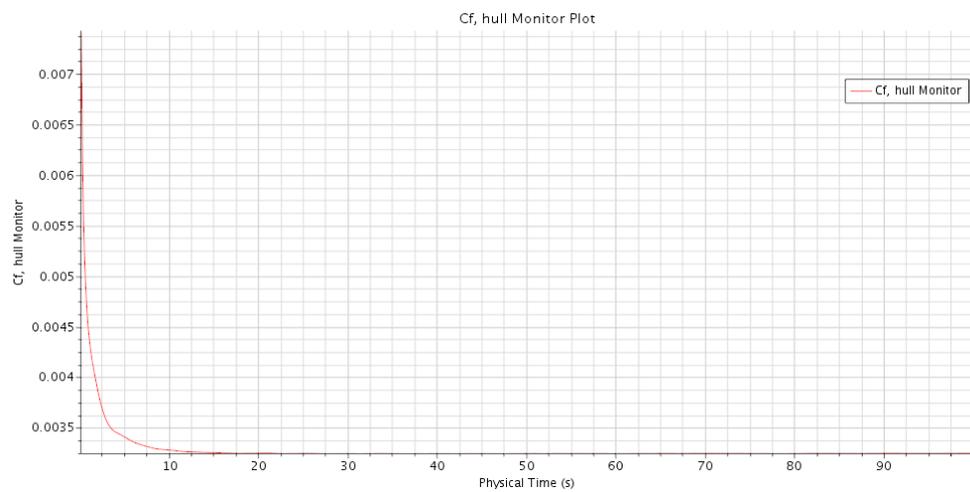
Model scale _ Half ship _ Fr = 0.1423



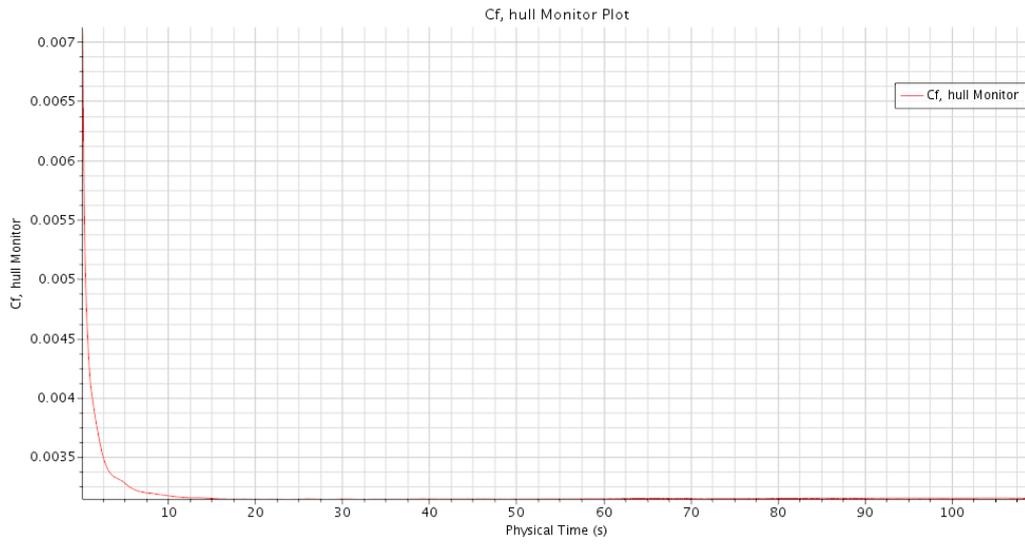
Model scale _ Fr = 0.0785 (8 knots)



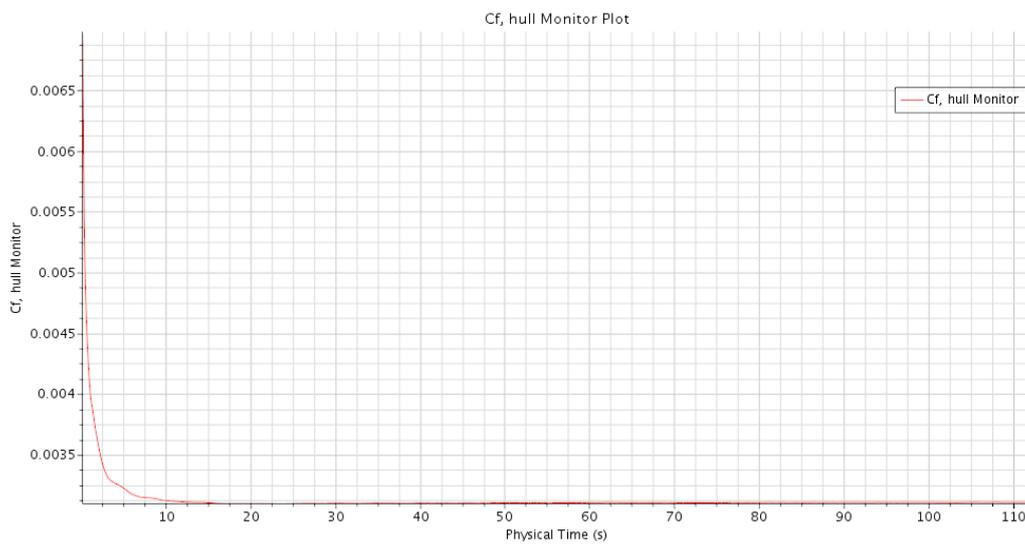
Model scale _ Fr = 0.0981 (10 knots)



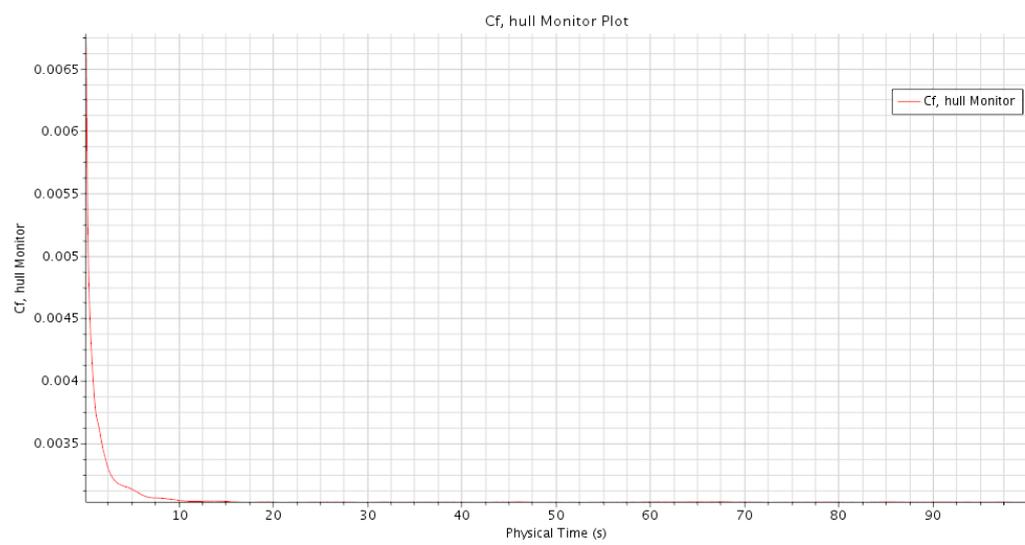
Model scale _ Fr = 0.1178 (12 knots)



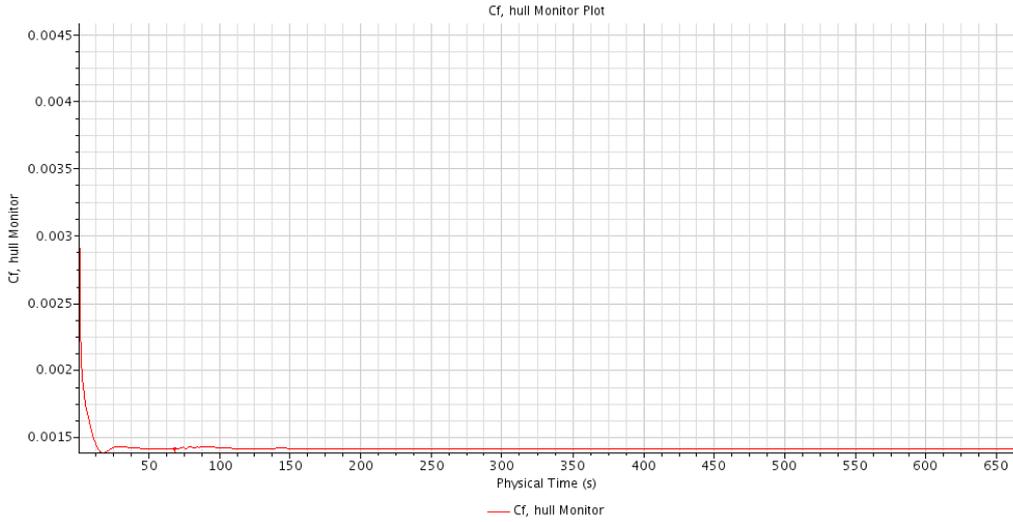
Model scale _ Fr = 0.1374 (14 knots)



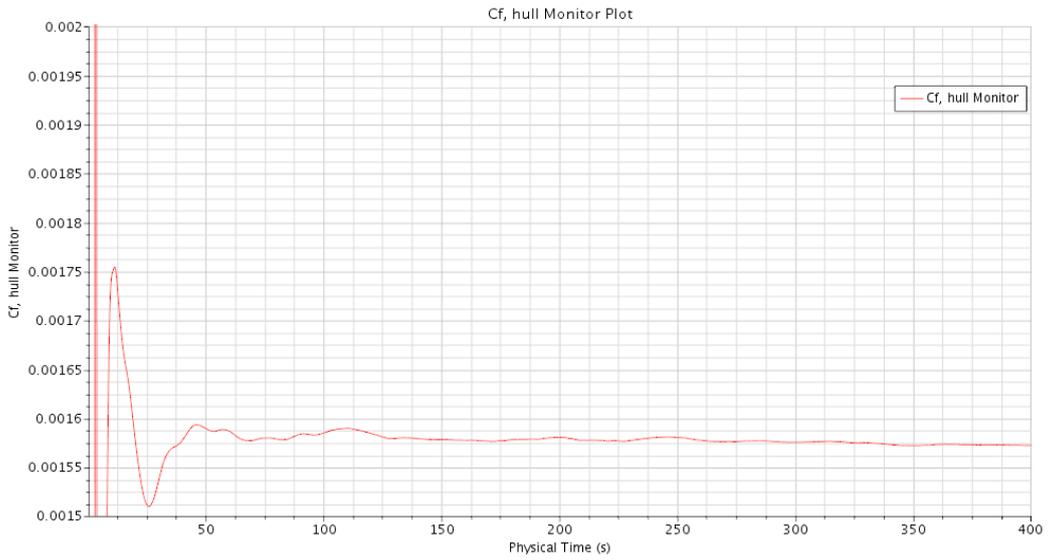
Model scale _ Fr = 0.1472 (15 knots)



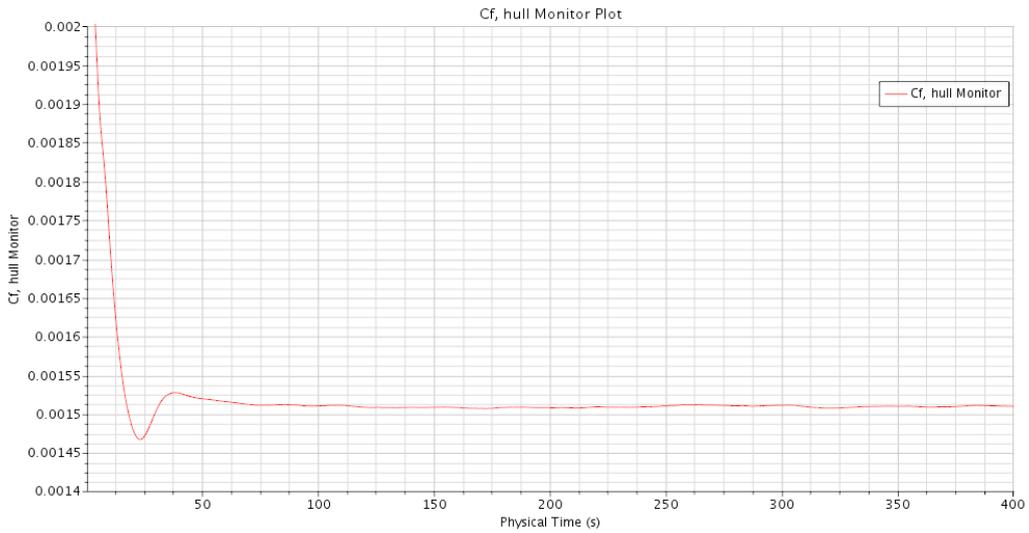
Model scale _ Fr = 0.1669 (17 knots)



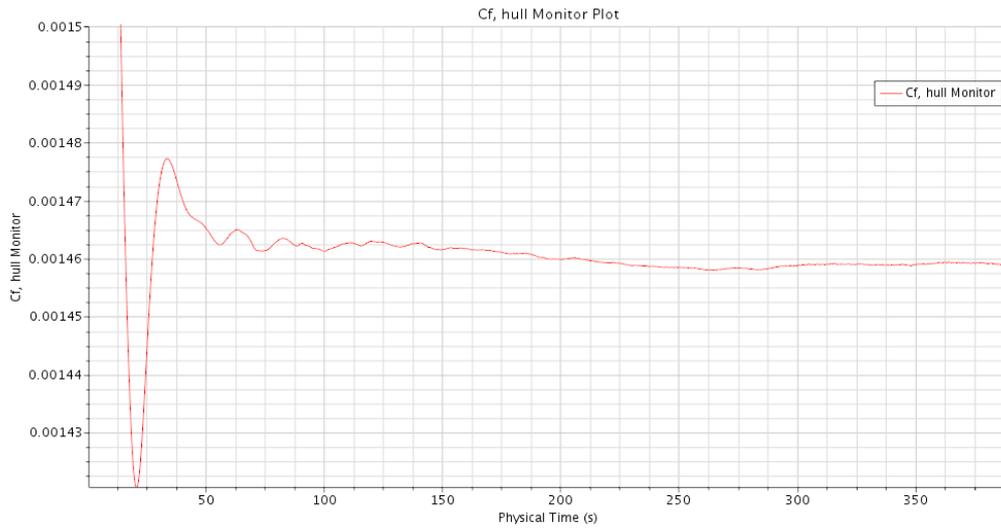
Full scale _ Fr = 0.1423 (14.5 knots)



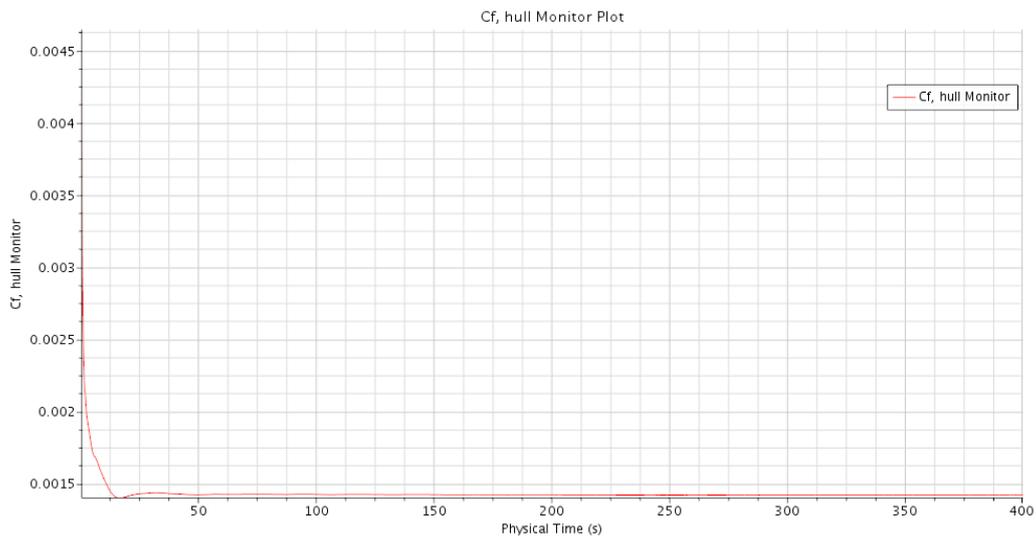
Full scale _ Fr = 0.0785 (8 knots)



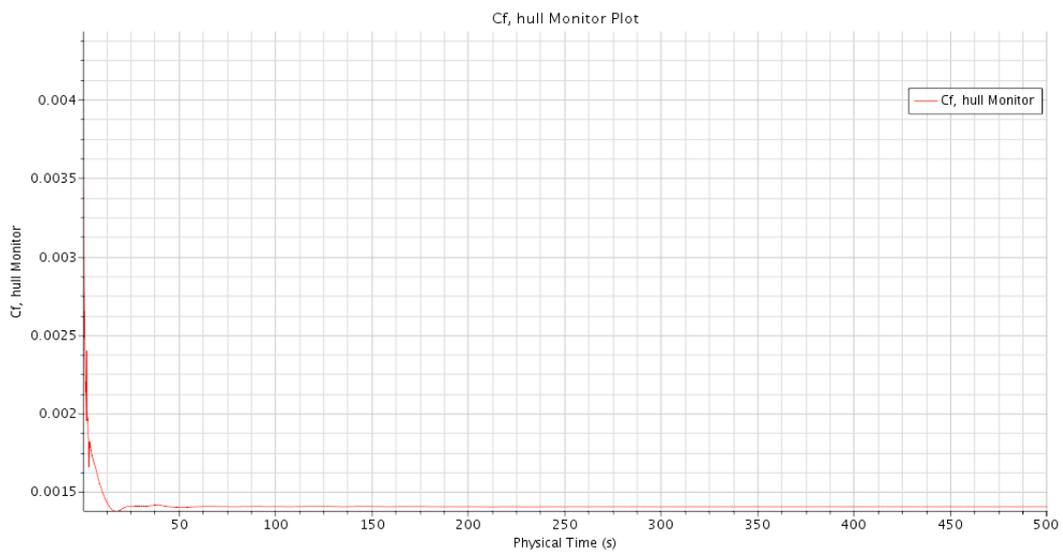
Full scale _ Fr = 0.0981 (10 knots)



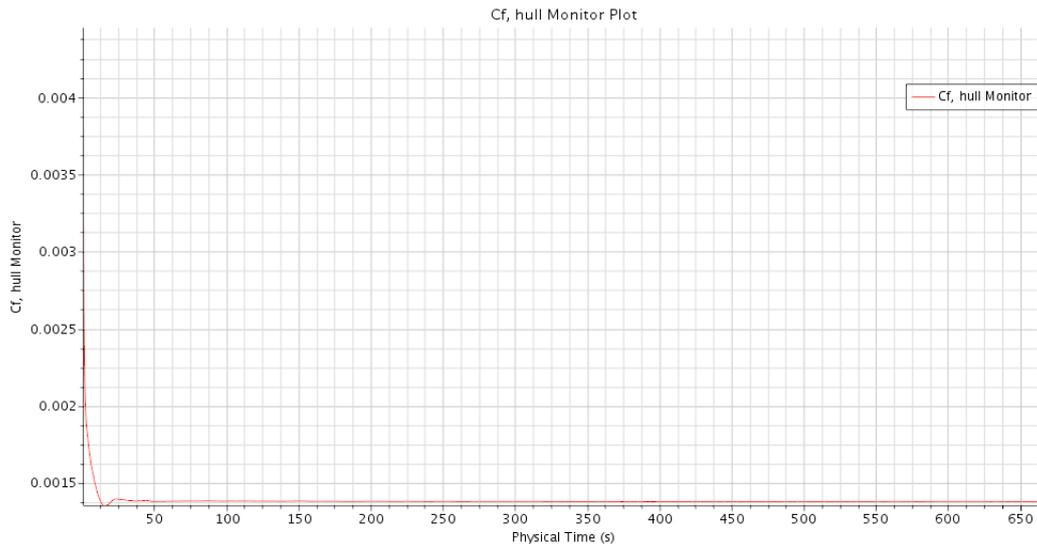
Full scale _ Fr = 0.1178 (12 knots)



Full scale _ Fr = 0.1374 (14 knots)

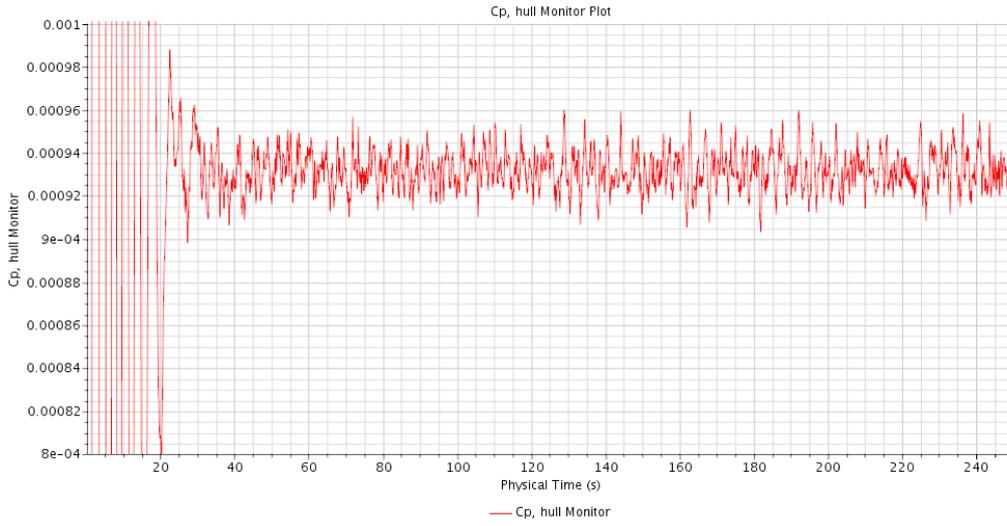


Full scale _ Fr = 0.1472 (15 knots)

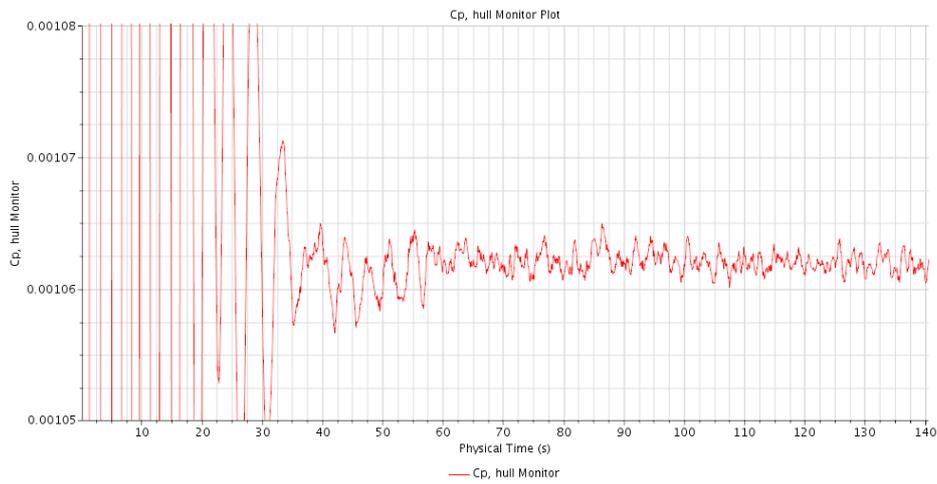


Full scale $_Fr = 0.1669$ (17 knots)

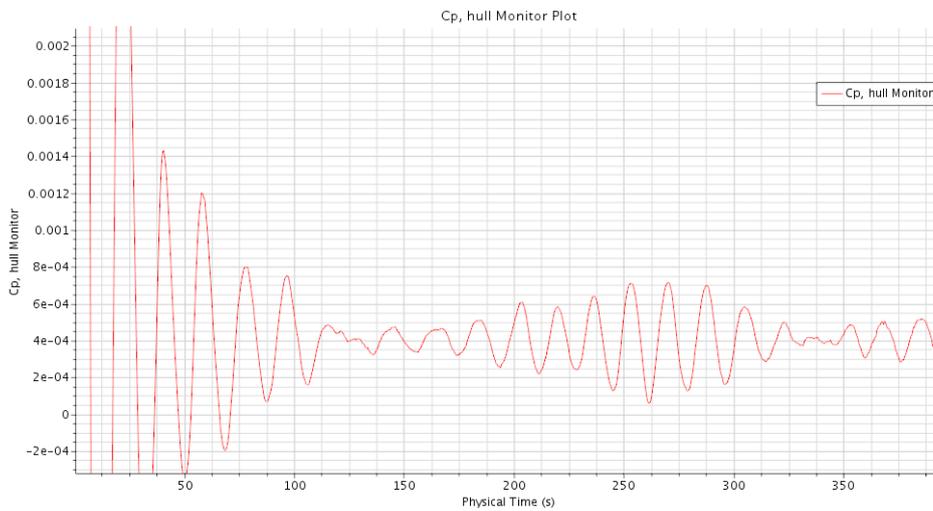
APPENDIX A4. CP



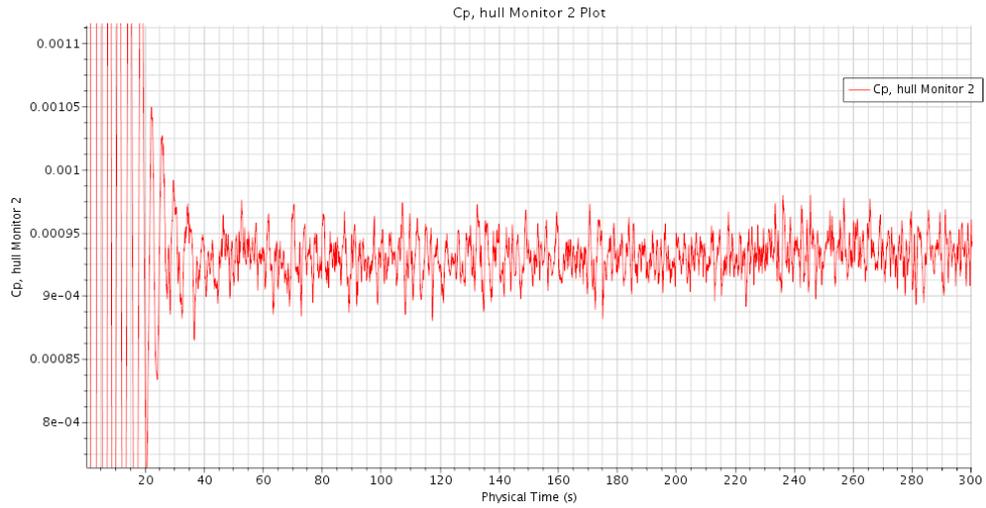
Model scale _ Fr = 0.1423 (14.5 knots)



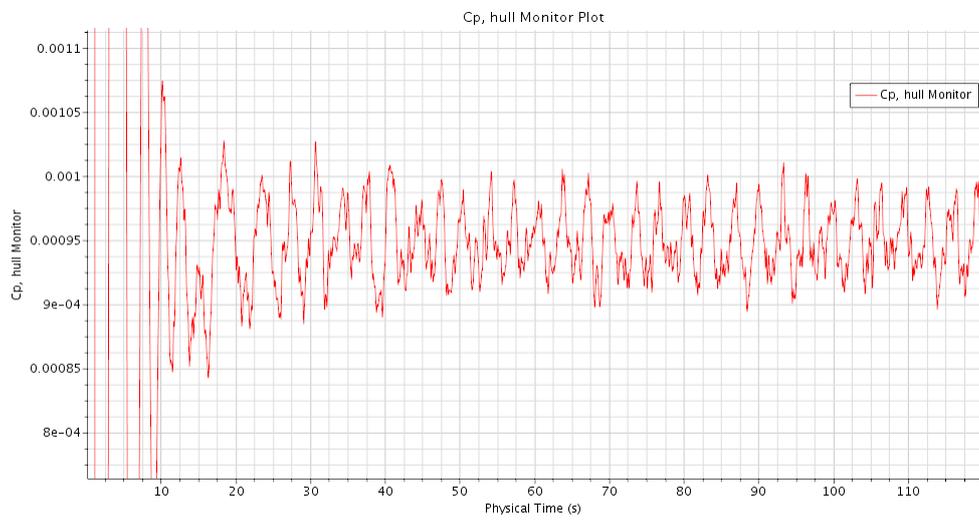
Model scale _ k-ε turbulence model _ Fr =0.1423



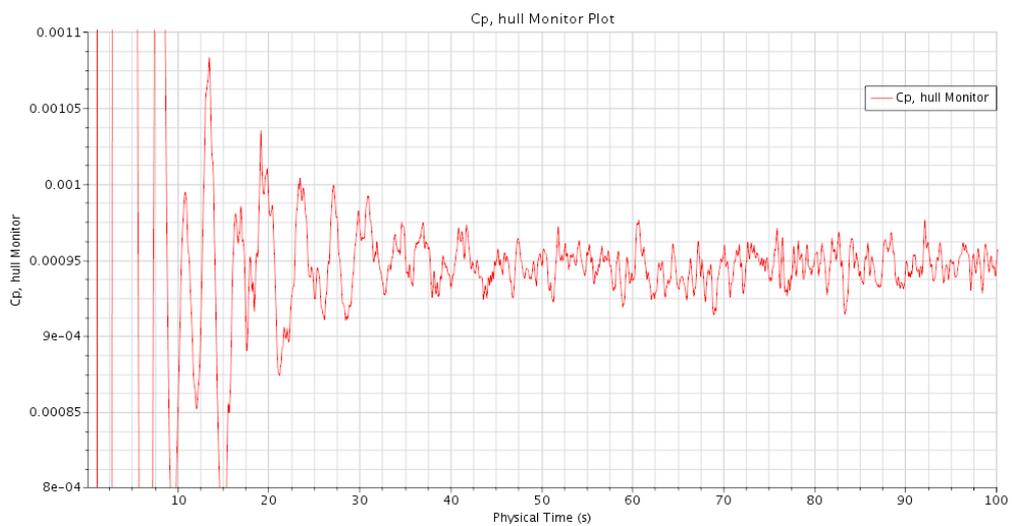
Model scale _ Half ship _ Fr = 0.1423



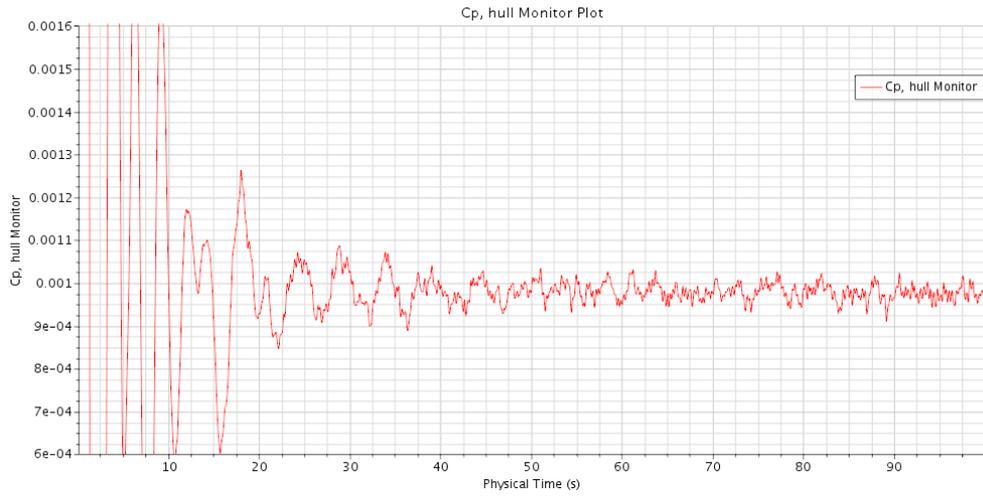
Model scale _ Fr = 0.0785 (8 knots)



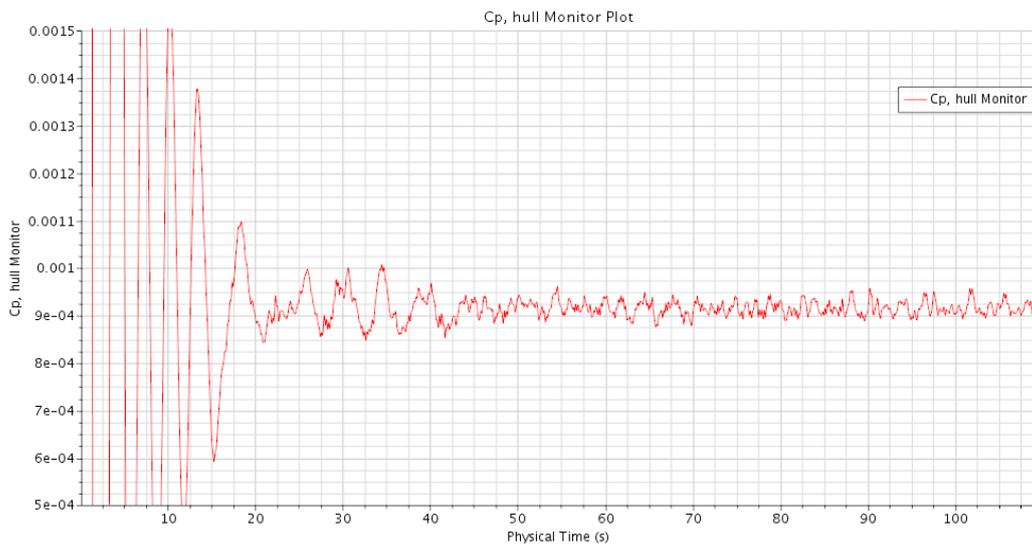
Model scale _ Fr = 0.0981 (10 knots)



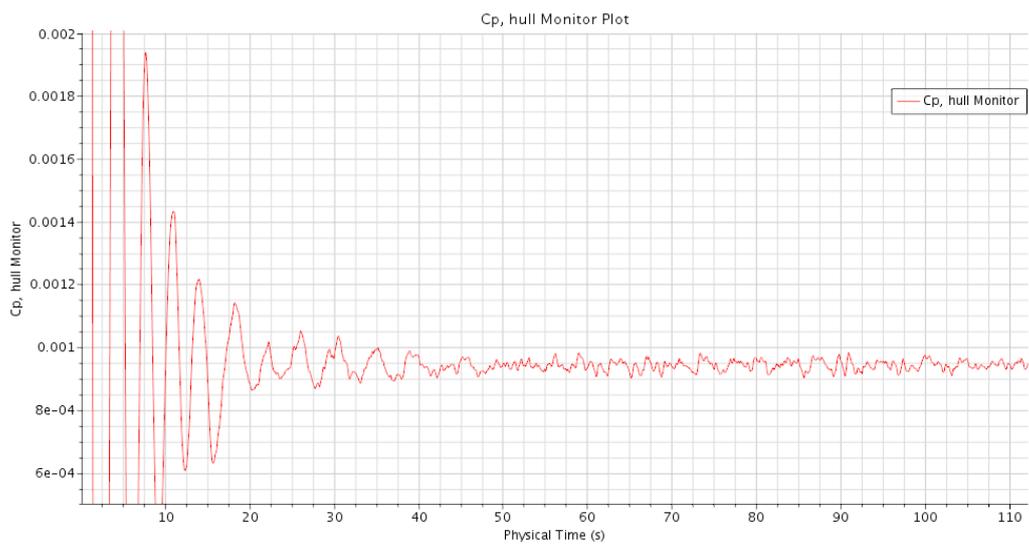
Model scale _ Fr = 0.1178 (12 knots)



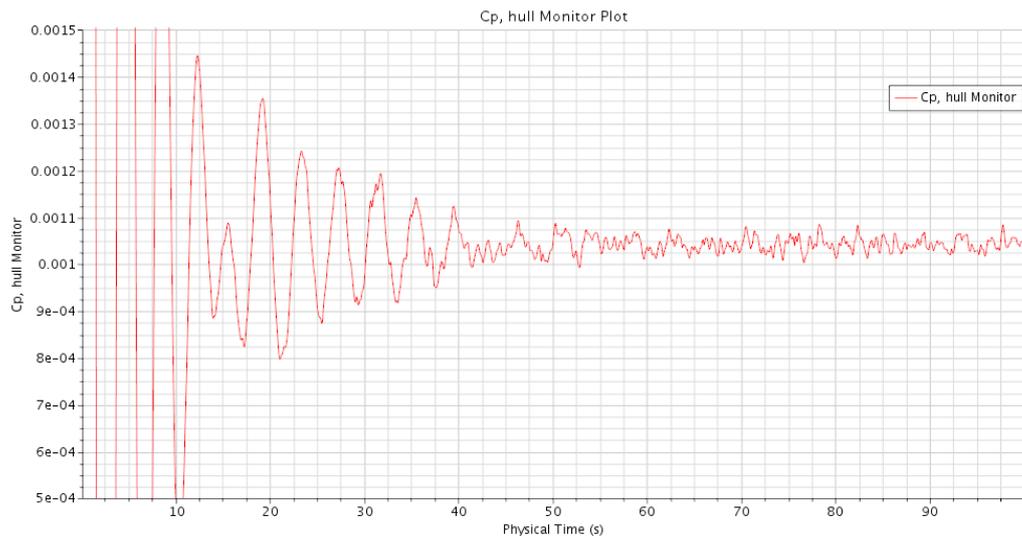
Model scale _ Fr = 0.1374 (14 knots)



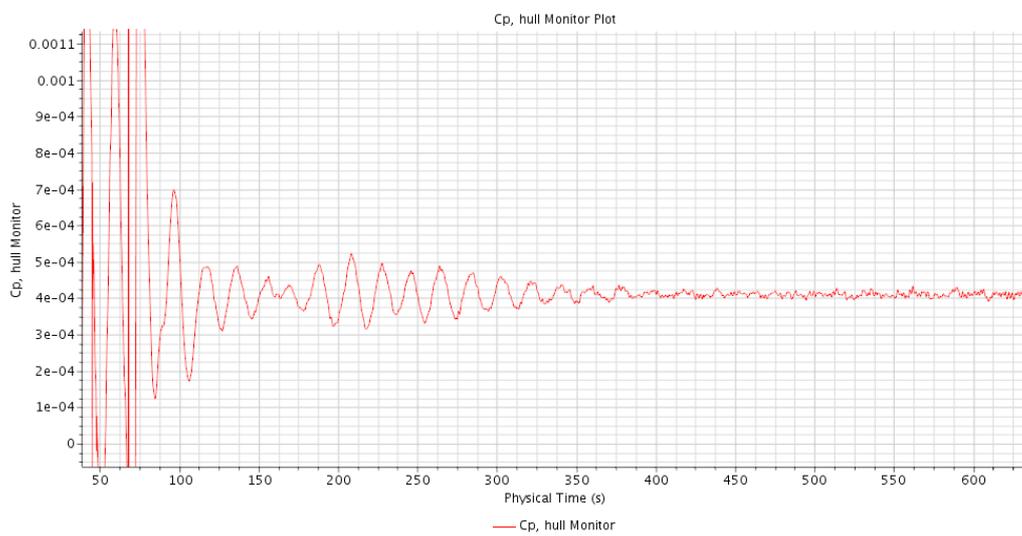
Model scale _ Fr = 0.1472 (15 knots)



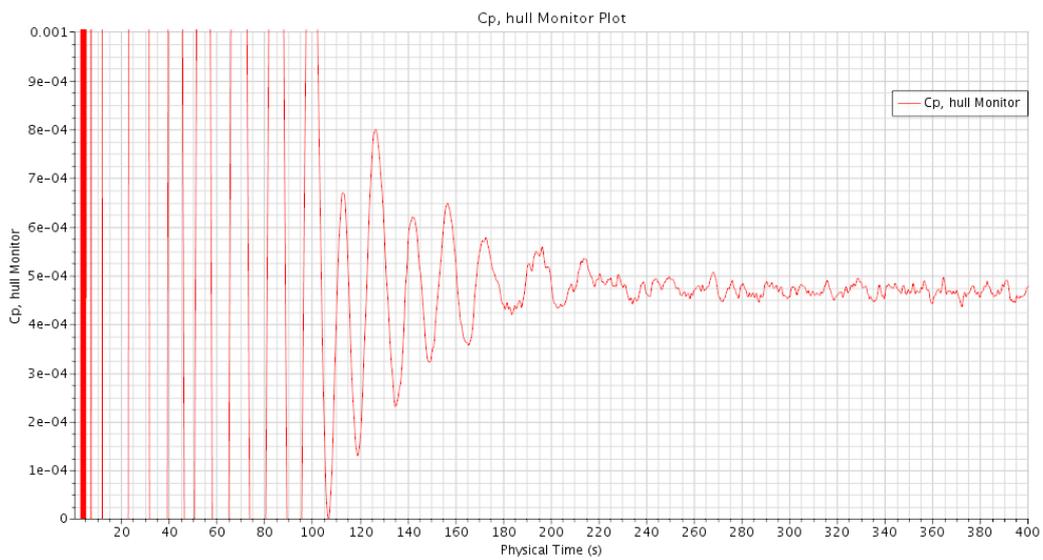
Model scale _ Fr = 0.1669 (17 knots)



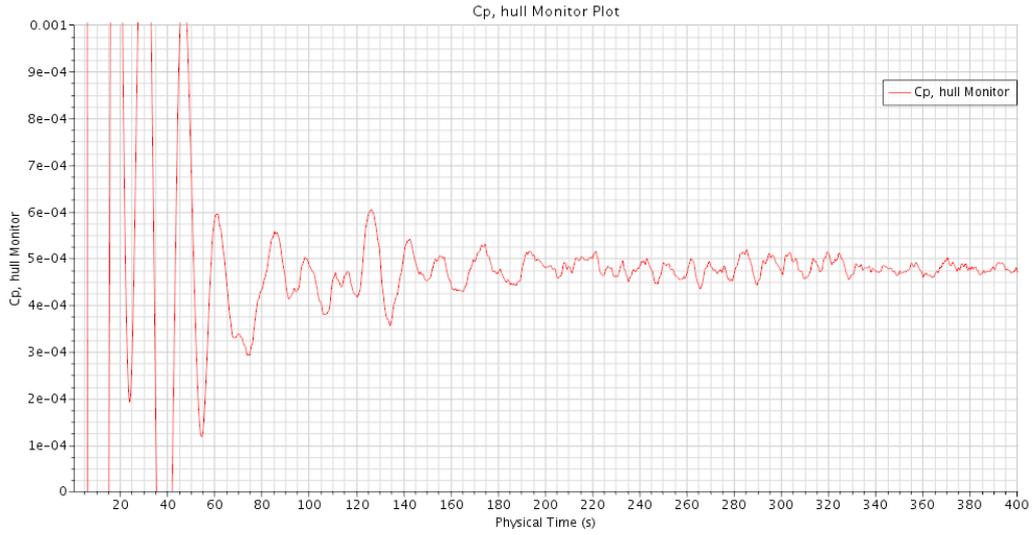
Full scale _ Fr = 0.1423 (14.5 knots)



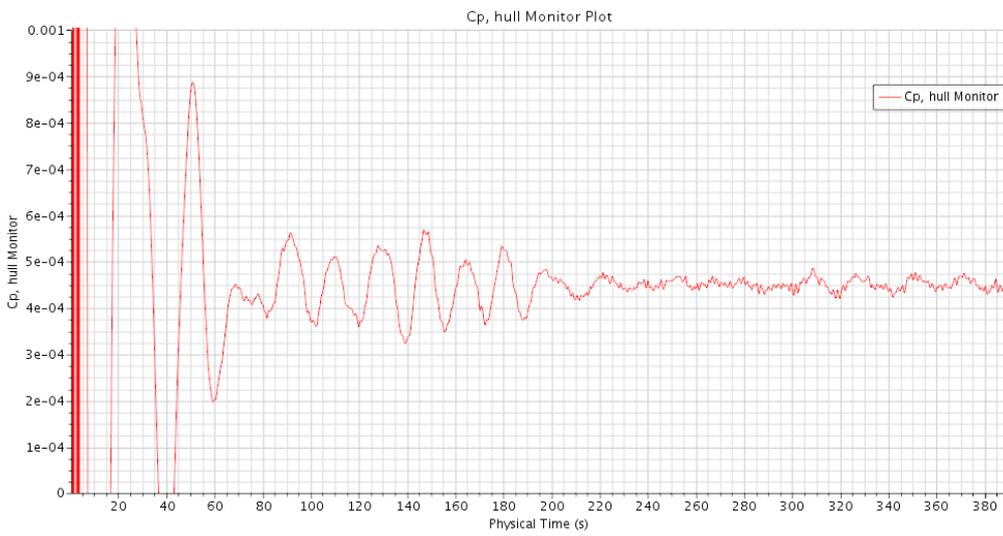
Full scale _ Half ship _ Fr = 0.1423



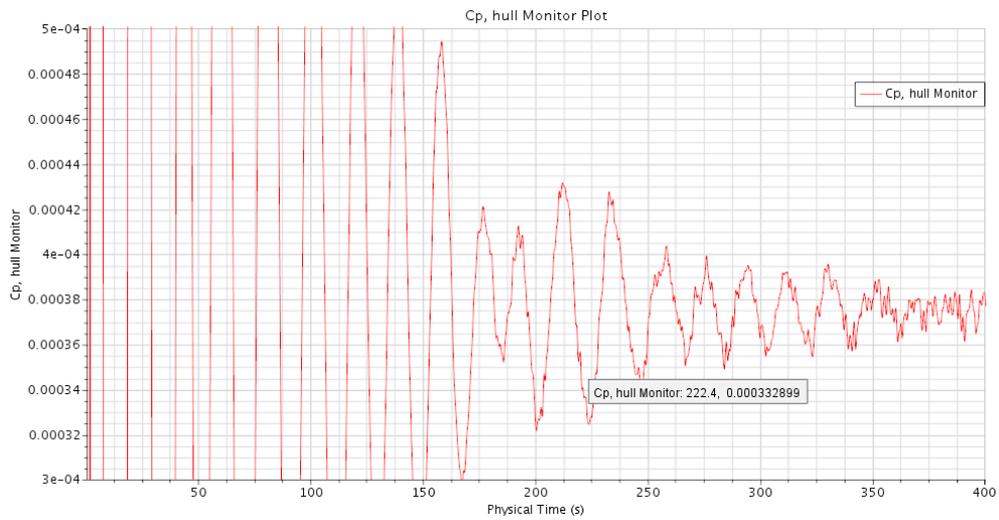
Full scale _ Fr = 0.0785 (8 knots)



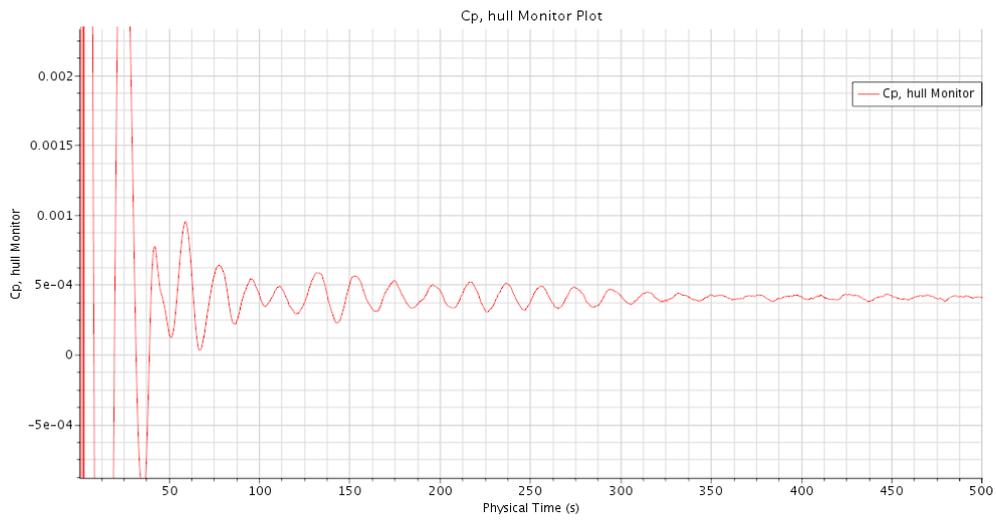
Full scale _ Fr = 0.0981 (10 knots)



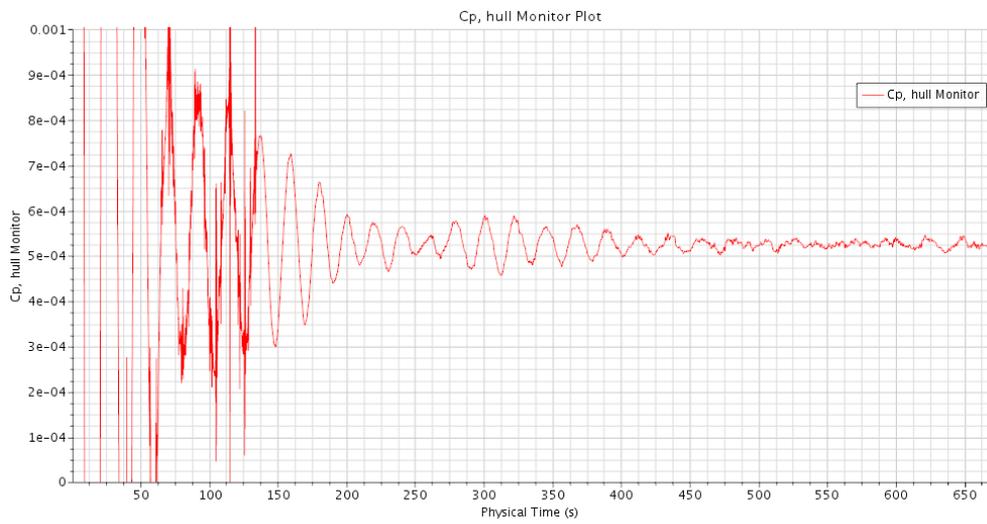
Full scale _ Fr = 0.1178 (12 knots)



Full scale _ Fr = 0.1374 (14 knots)

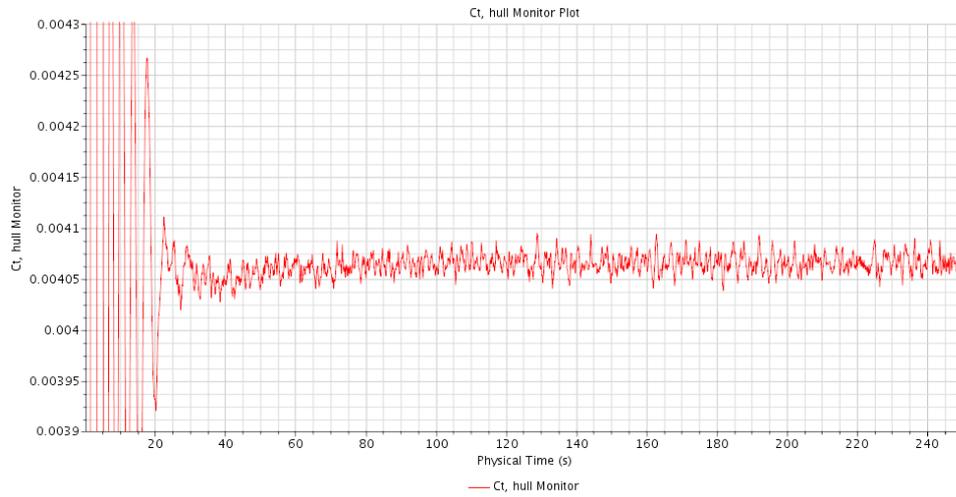


Full scale _ Fr = 0.1472 (15 knots)

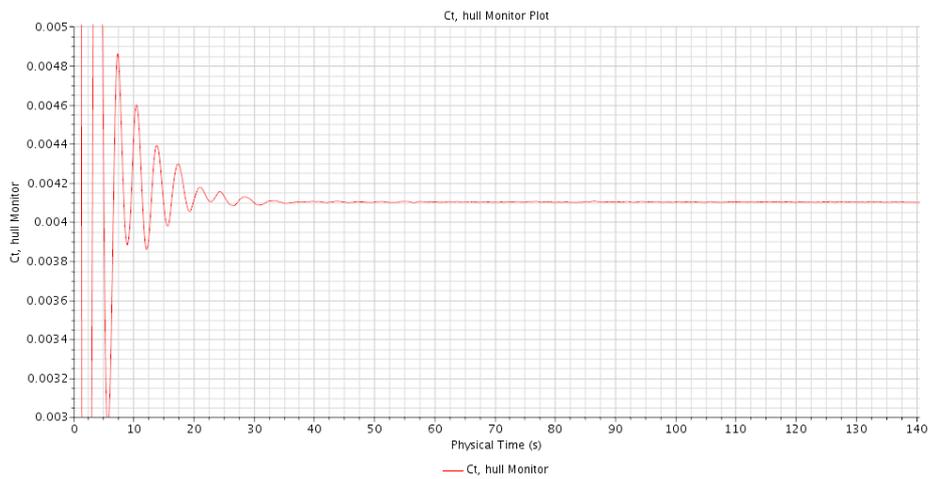


Full scale _ Fr = 0.1669 (17 knots)

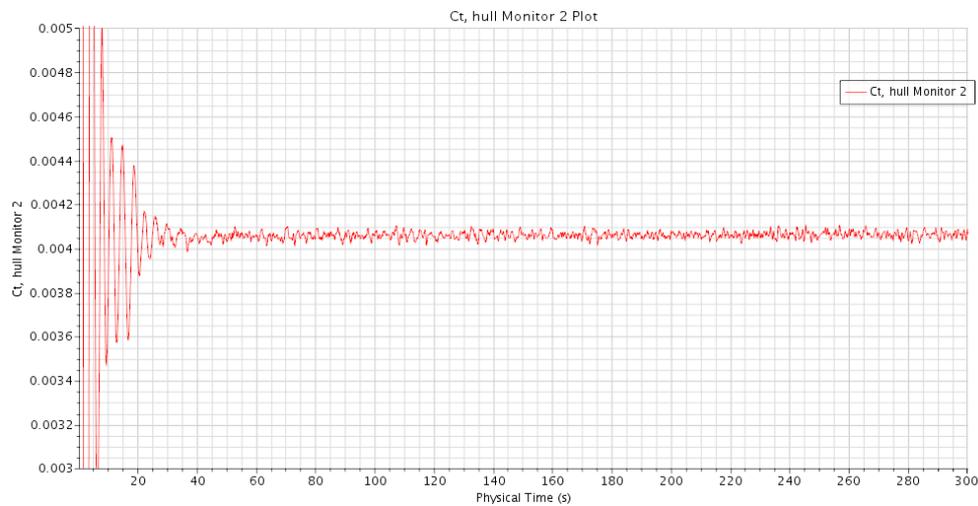
APPENDIX A5. CT



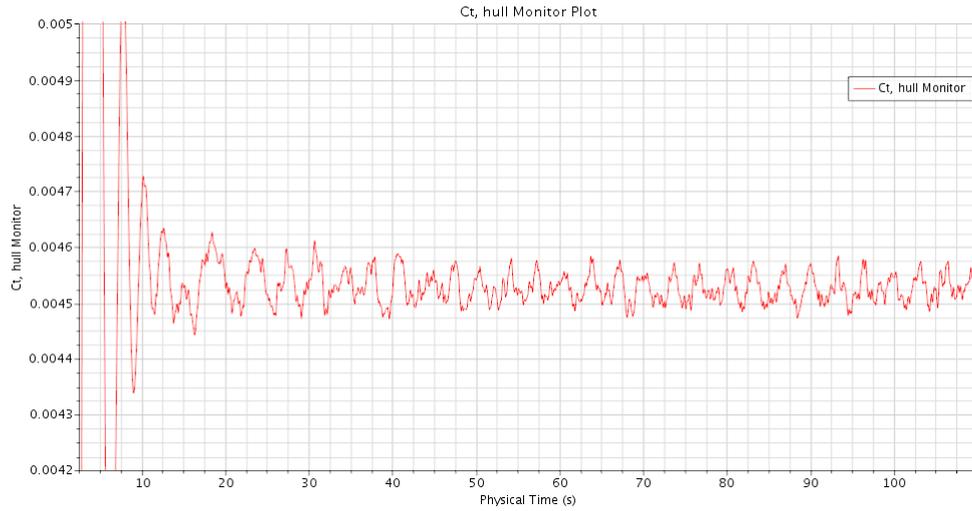
Model scale _ Fr = 0.1423 (14.5 knots)



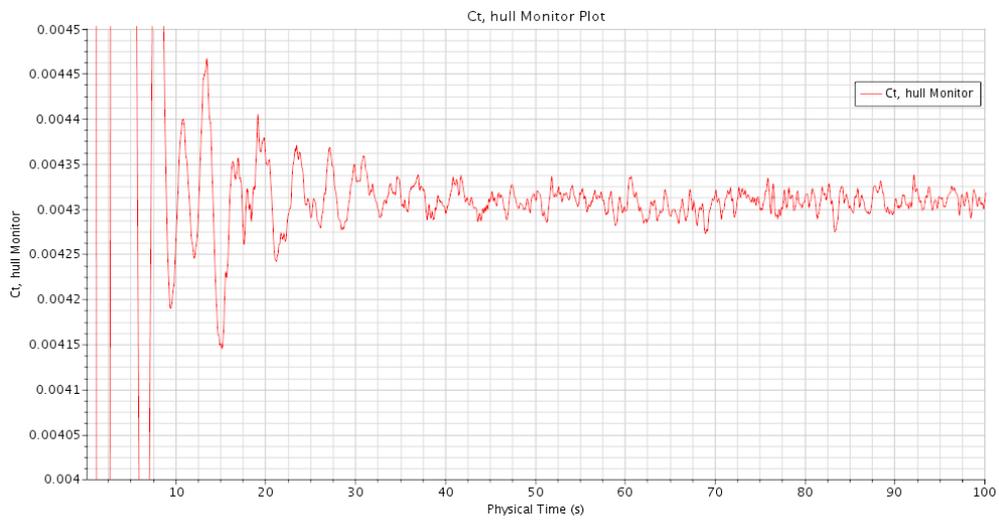
Model scale _ k-ε turbulence model _ Fr = 0.1423



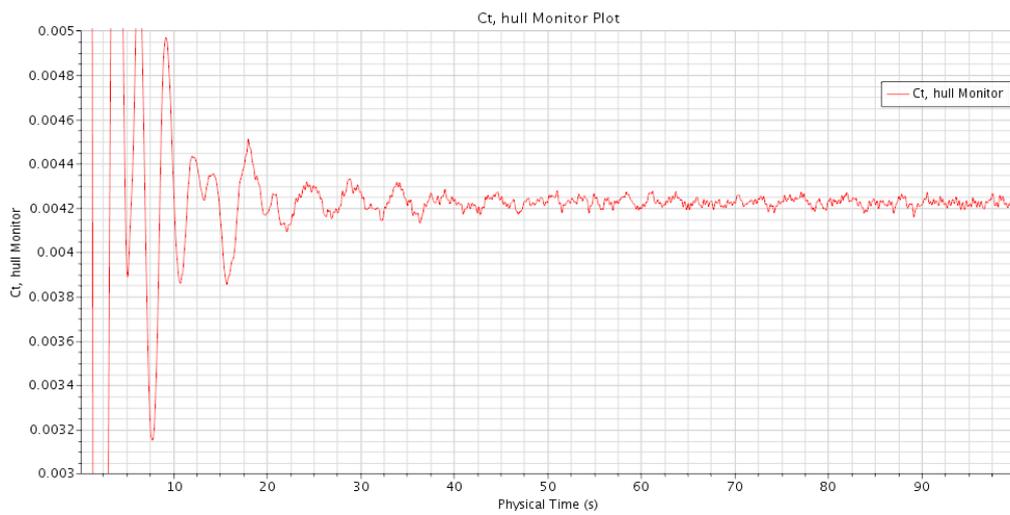
Model scale _ Half ship _ Fr = 0.1423



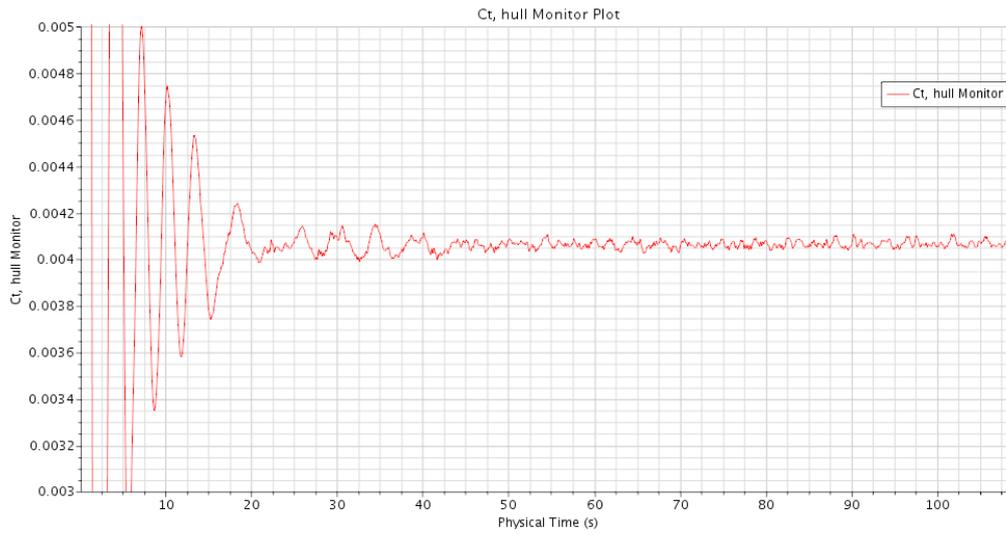
Model scale _ Fr = 0.0785 (8 knots)



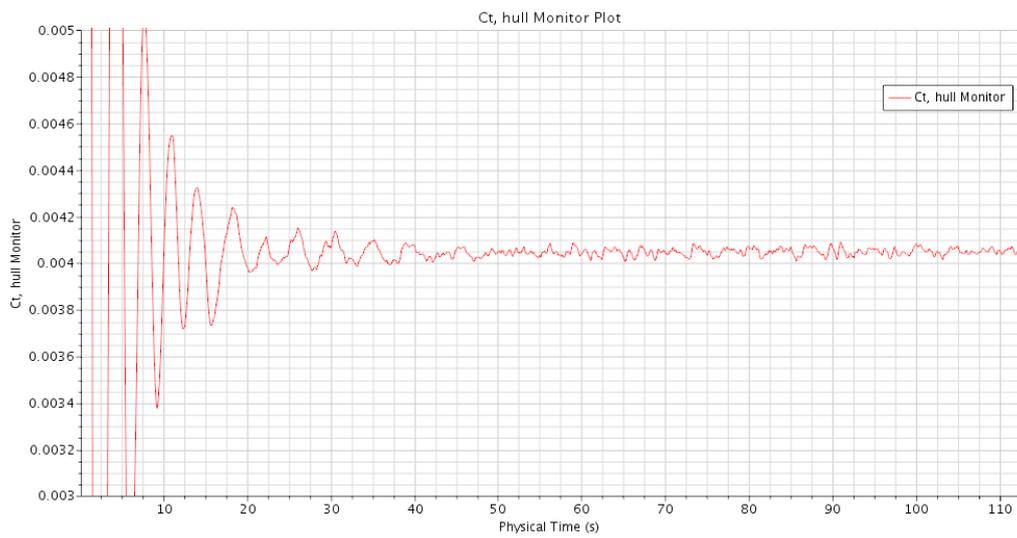
Model scale _ Fr = 0.0981 (10 knots)



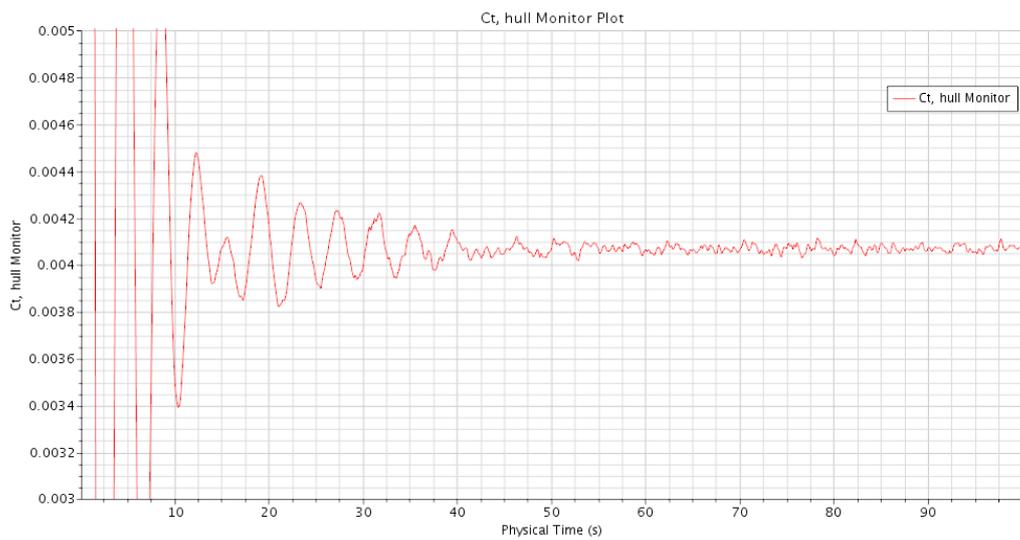
Model scale _ Fr = 0.1178 (12 knots)



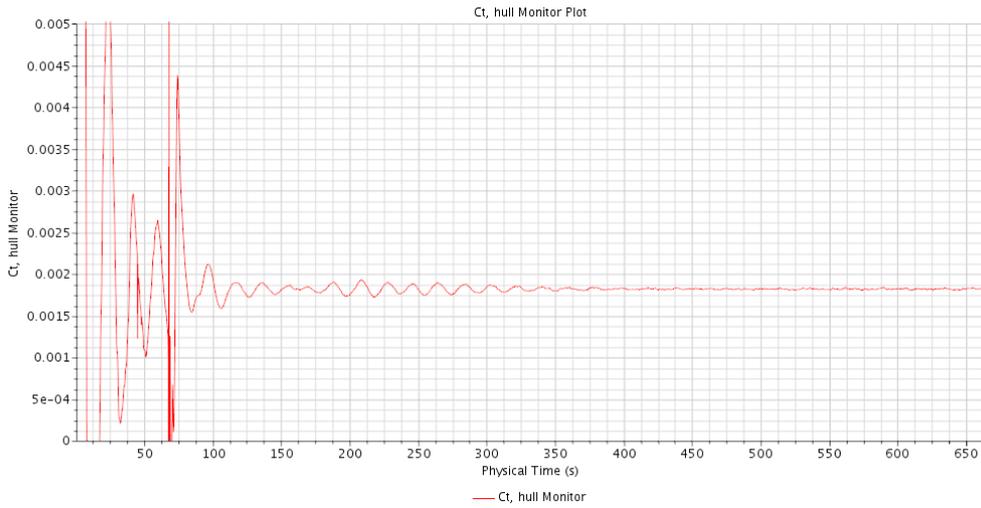
Model scale $_ Fr = 0.1374$ (14 knots)



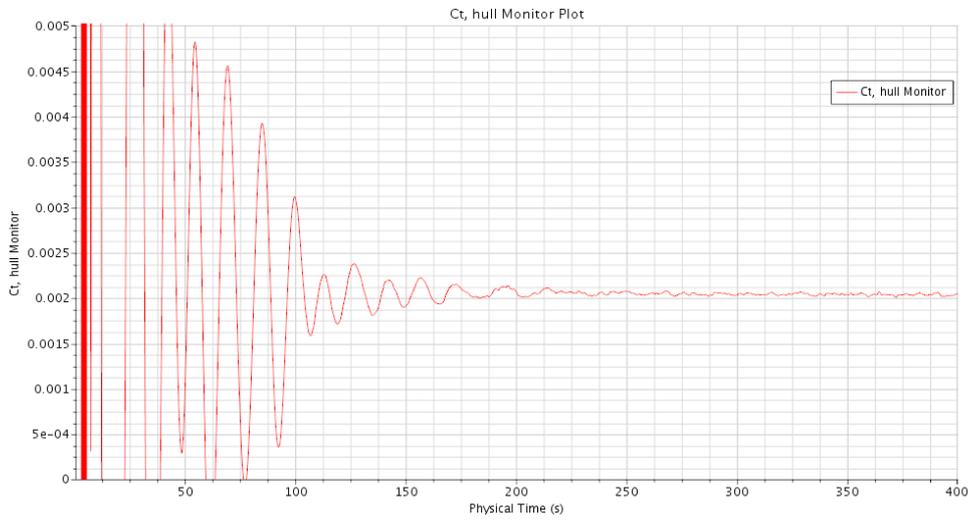
Model scale $_ Fr = 0.1472$ (15 knots)



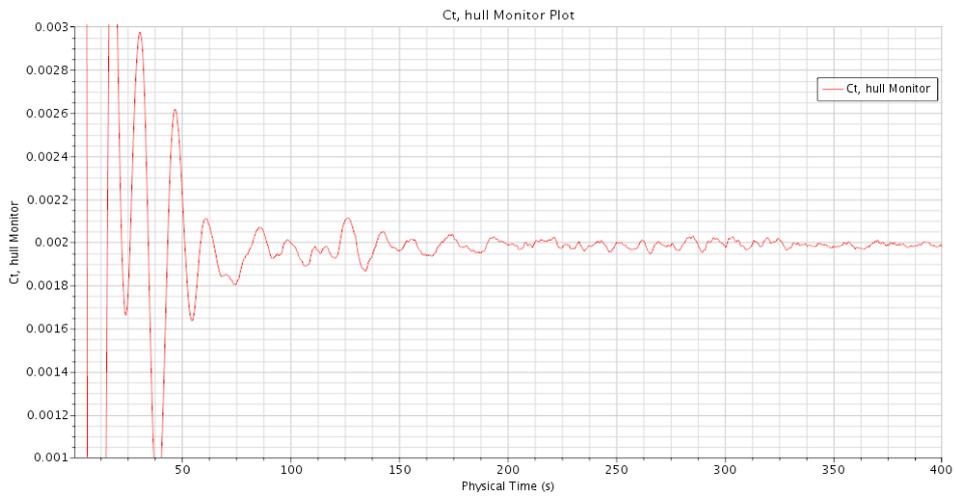
Model scale $_ Fr = 0.1669$ (17 knots)



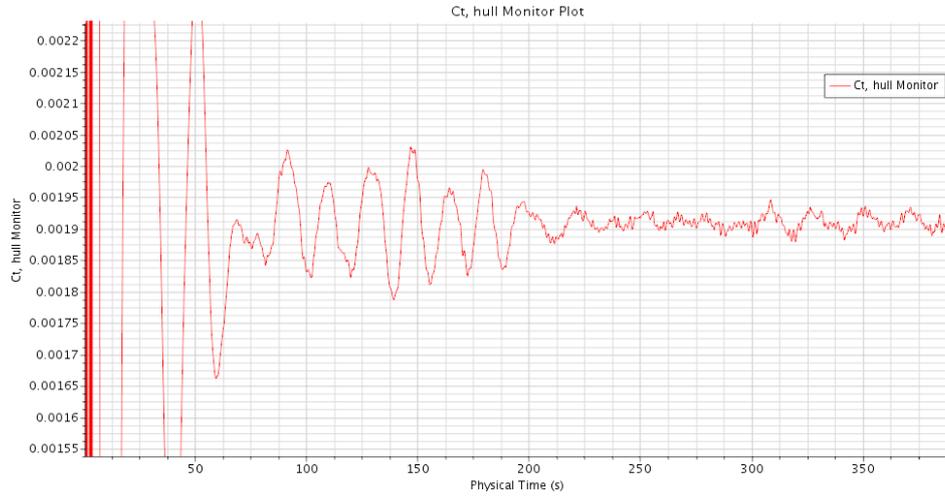
Full scale _ Fr = 0.1423 (14.5 knots)



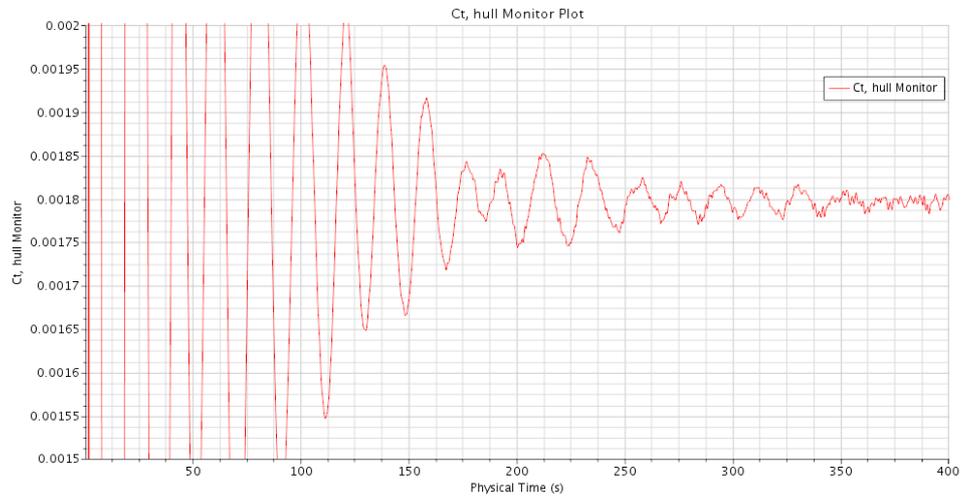
Full scale _ Fr = 0.0785 (8 knots)



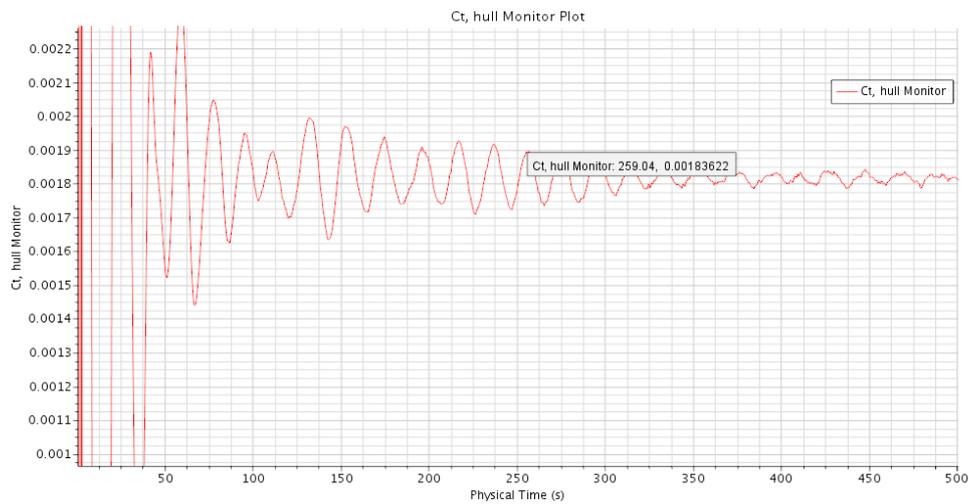
Full scale _ Fr = 0.0981 (10 knots)



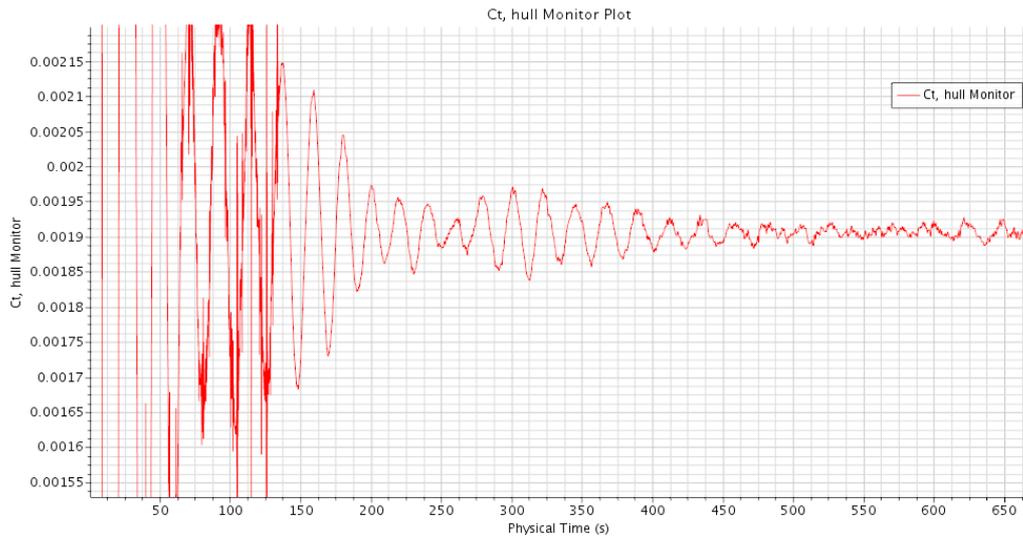
Full scale _ Fr = 0.1178 (12 knots)



Full scale _ Fr = 0.1374 (14 knots)

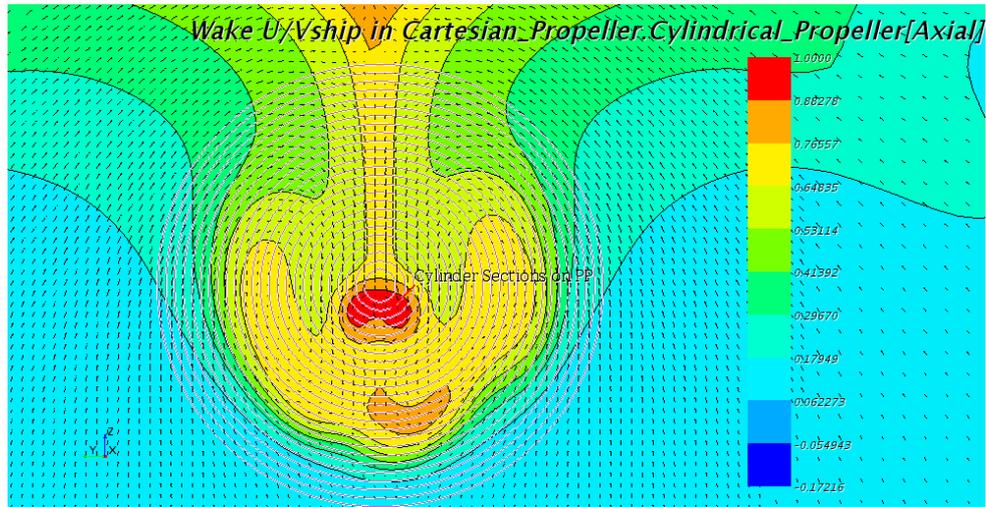


Full scale _ Fr = 0.1472 (15 knots)

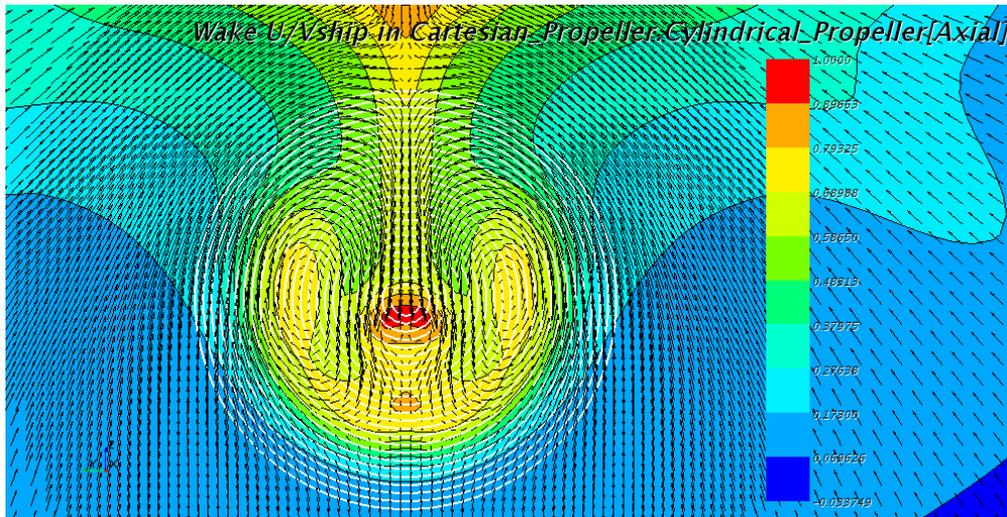


Full scale _ Fr = 0.1669 (17 knots)

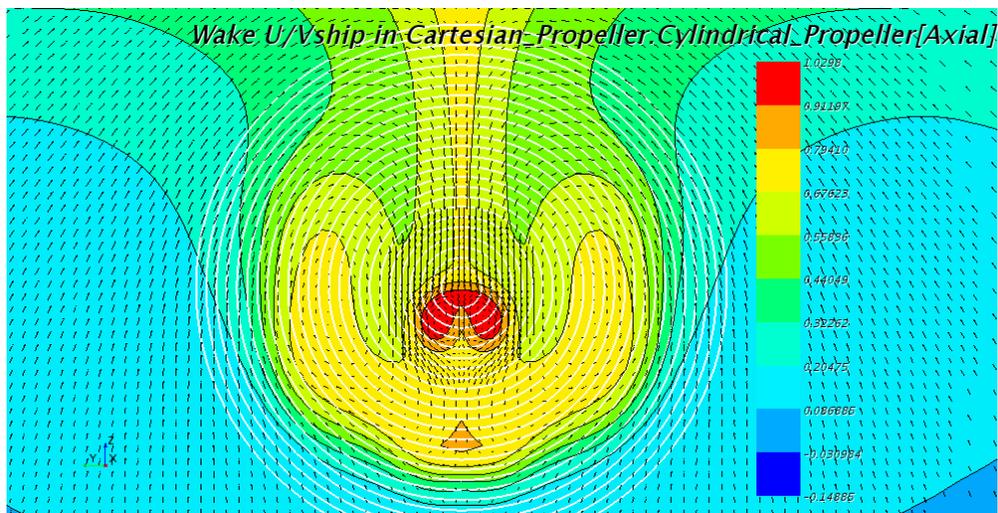
APPENDIX A6. WAKE FIELD



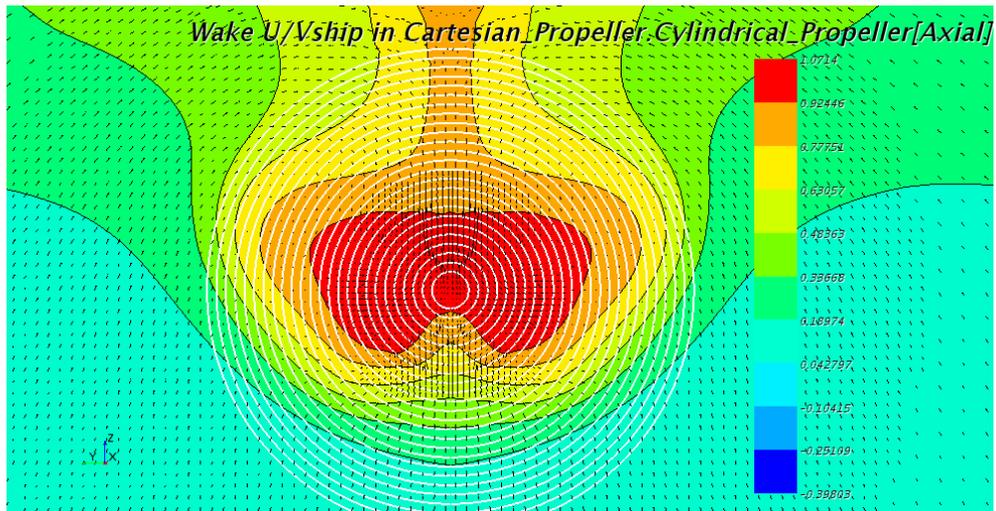
Model scale _ Fr = 0.1423 (14.5 knots)



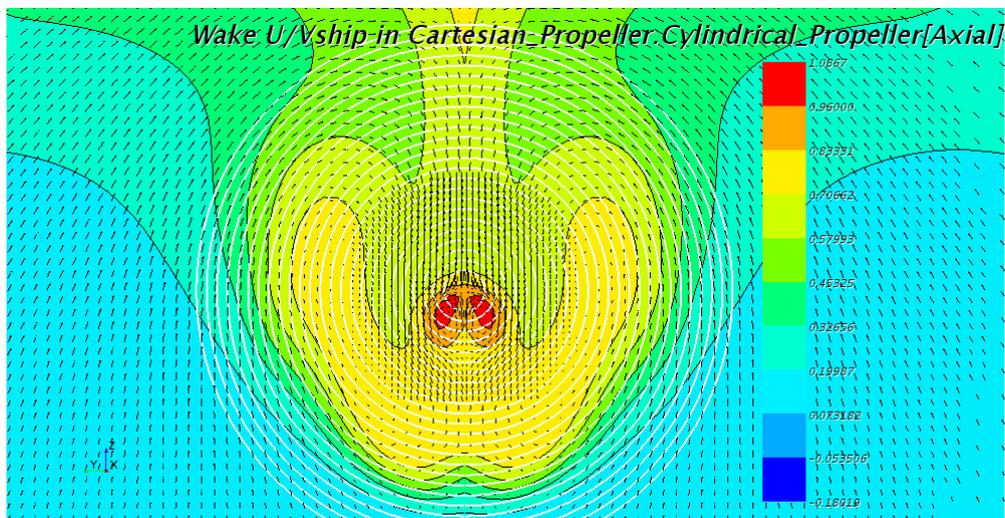
Model scale _ k-ε turbulence model _ Fr =0.1423



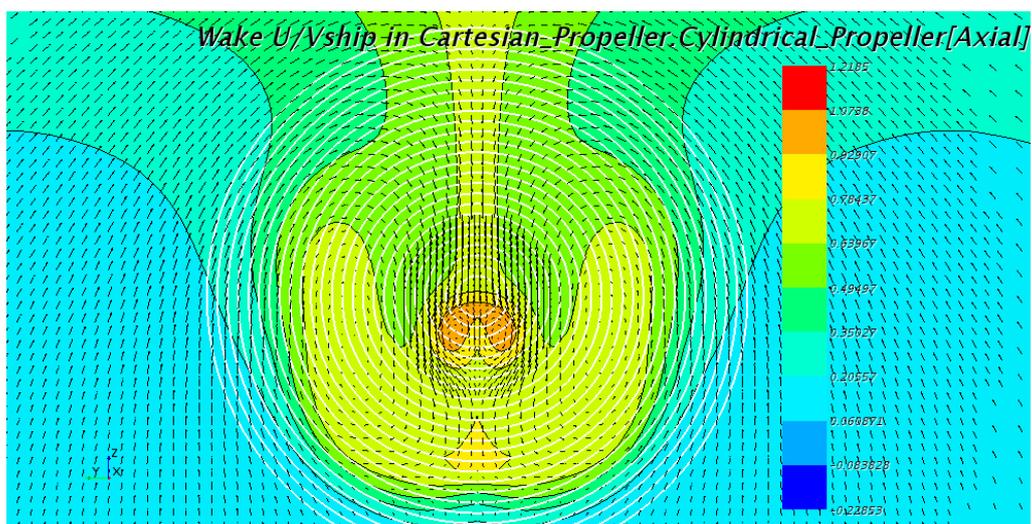
Model scale _ Half ship _ Fr = 0.1423



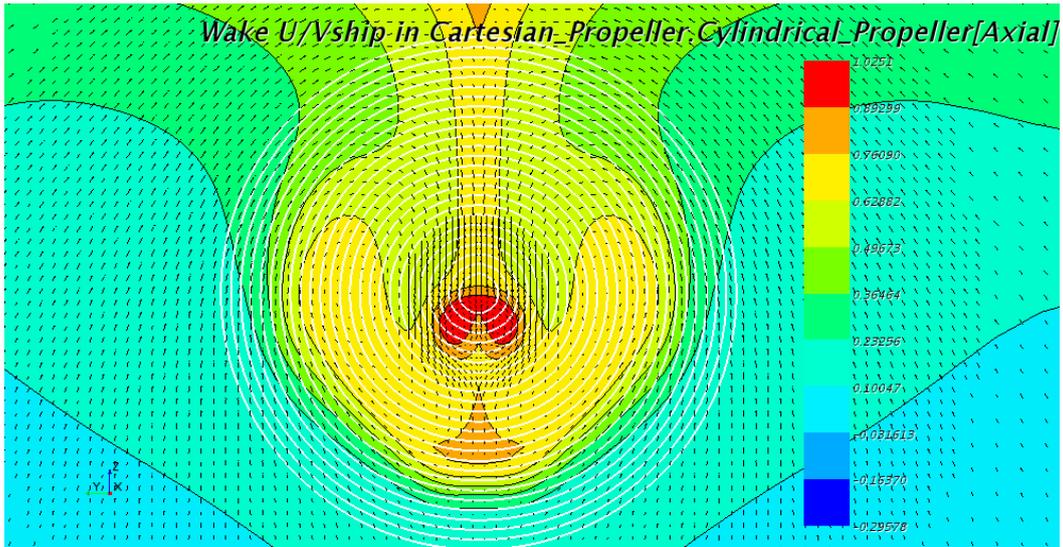
Model scale _ Fr = 0.0785 (8 knots)



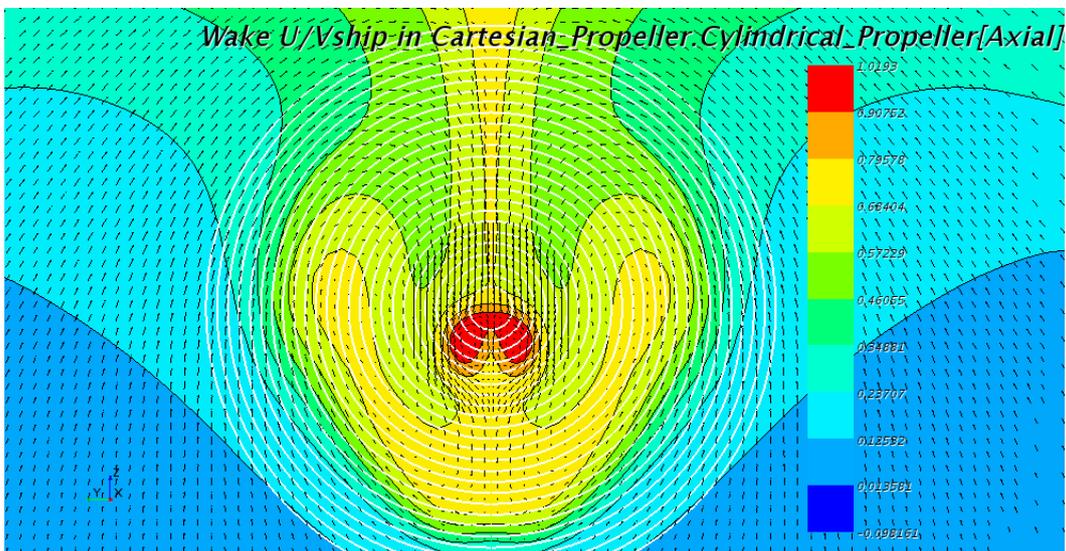
Model scale _ Fr = 0.0981 (10 knots)



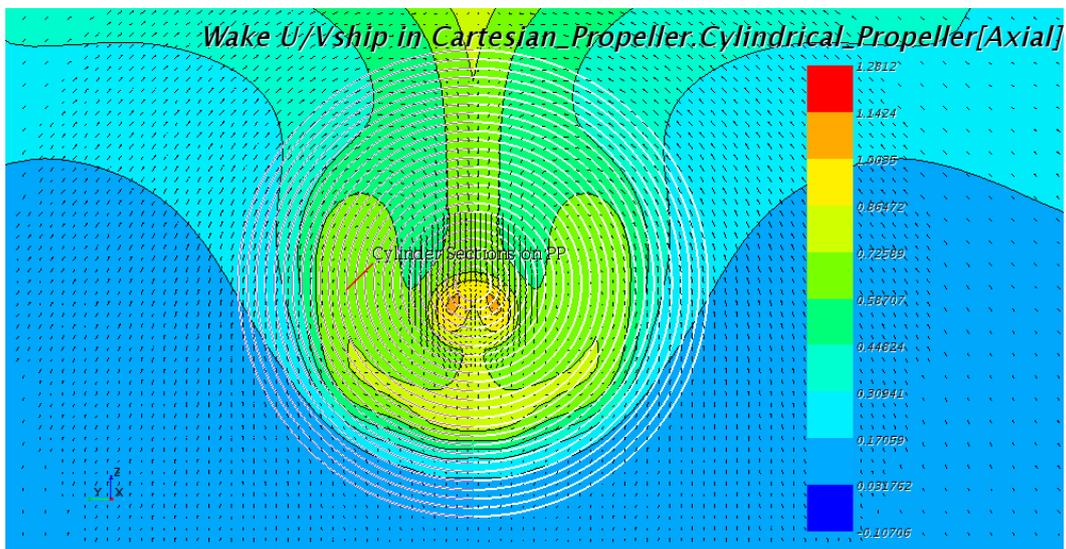
Model scale _ Fr = 0.1178 (12 knots)



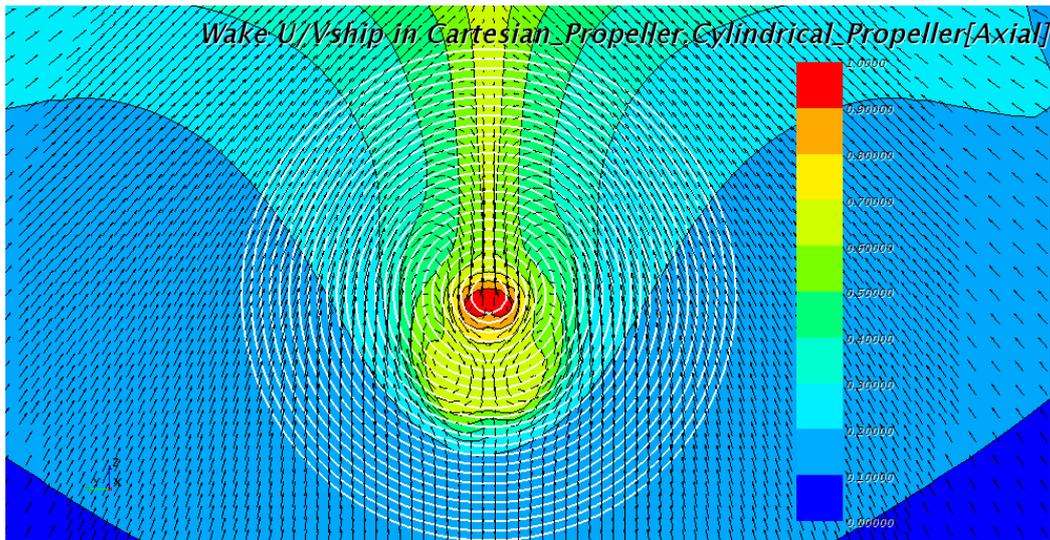
Model scale _ Fr = 0.1374 (14 knots)



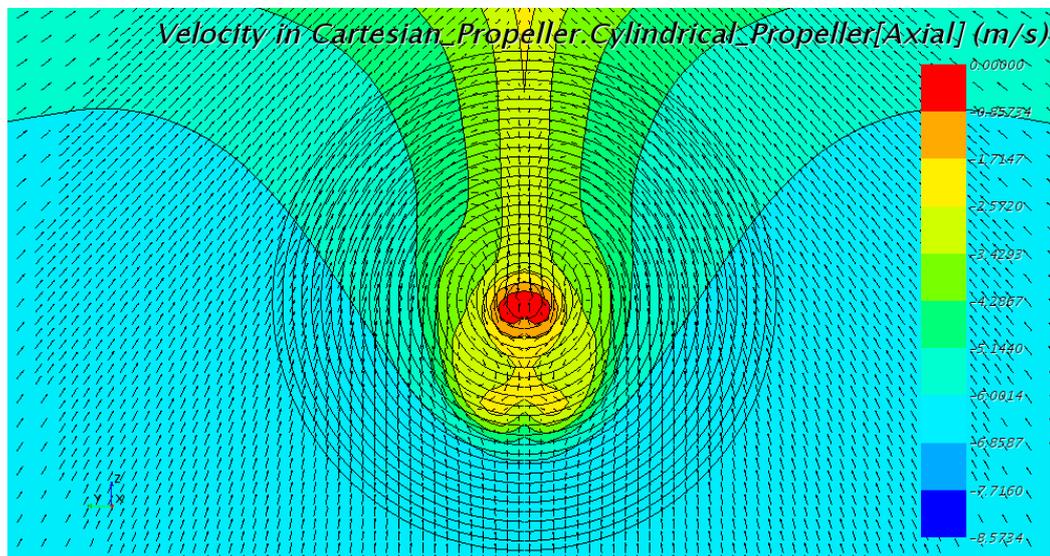
Model scale _ Fr = 0.1472 (15 knots)



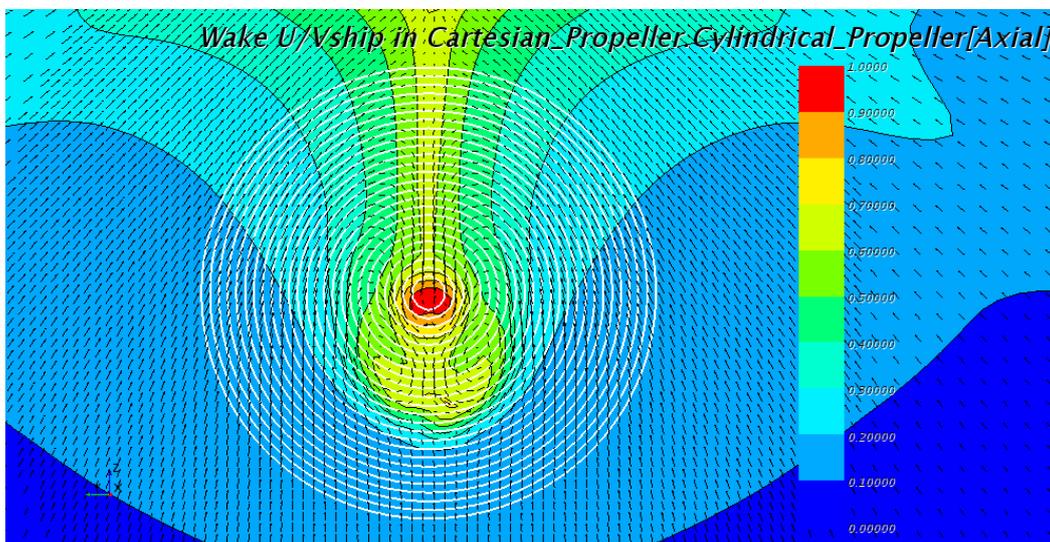
Model scale _ Fr = 0.1669 (17 knots)



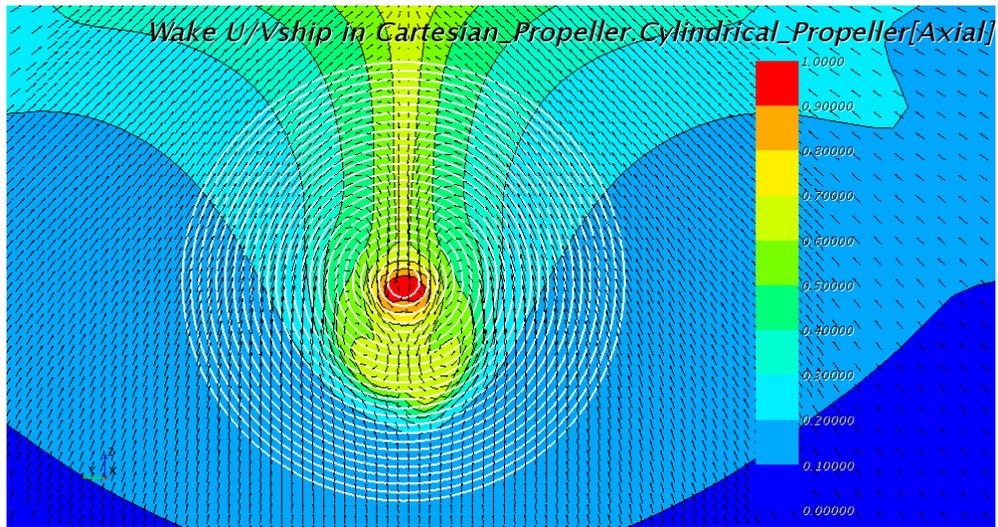
Full scale _ Fr = 0.1423 (14.5 knots)



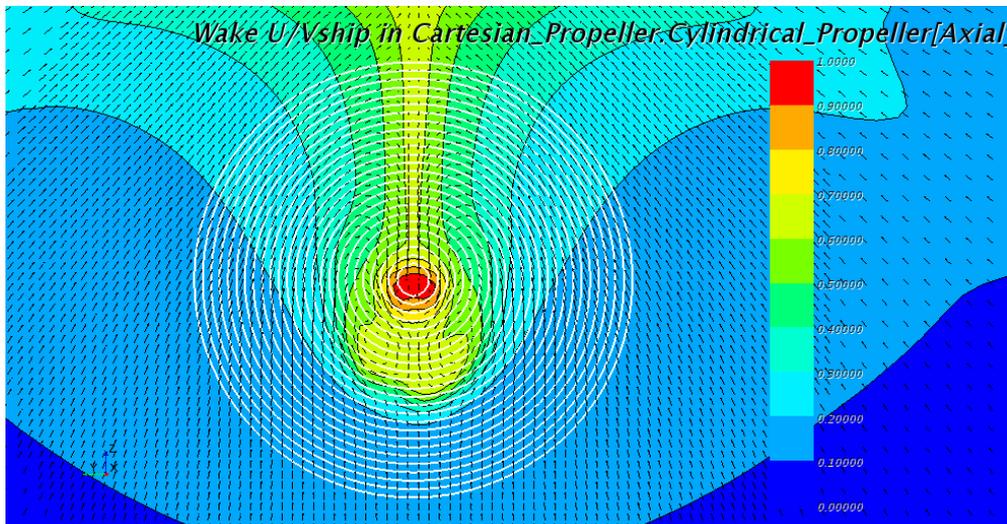
Full scale _ Half ship _ Fr = 0.1423



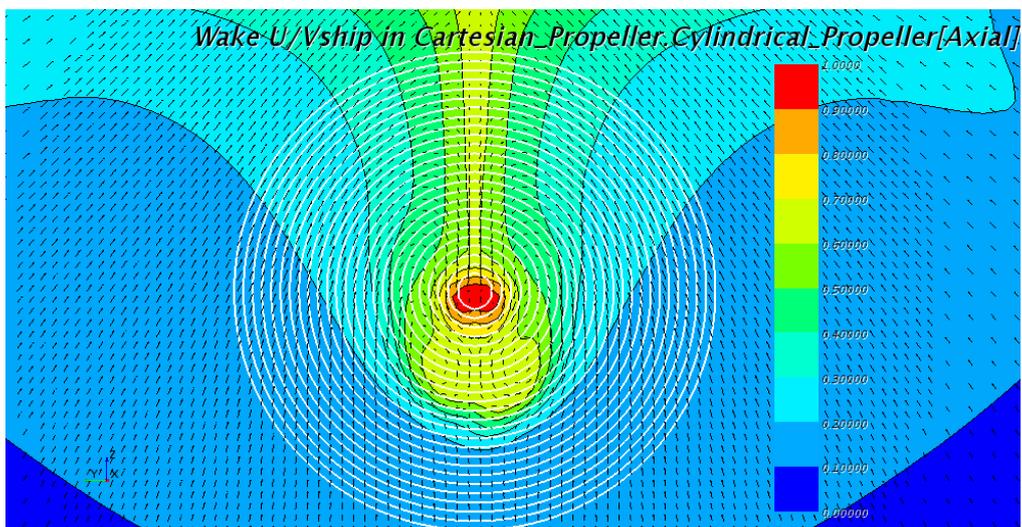
Full scale _ Fr = 0.0785 (8 knots)



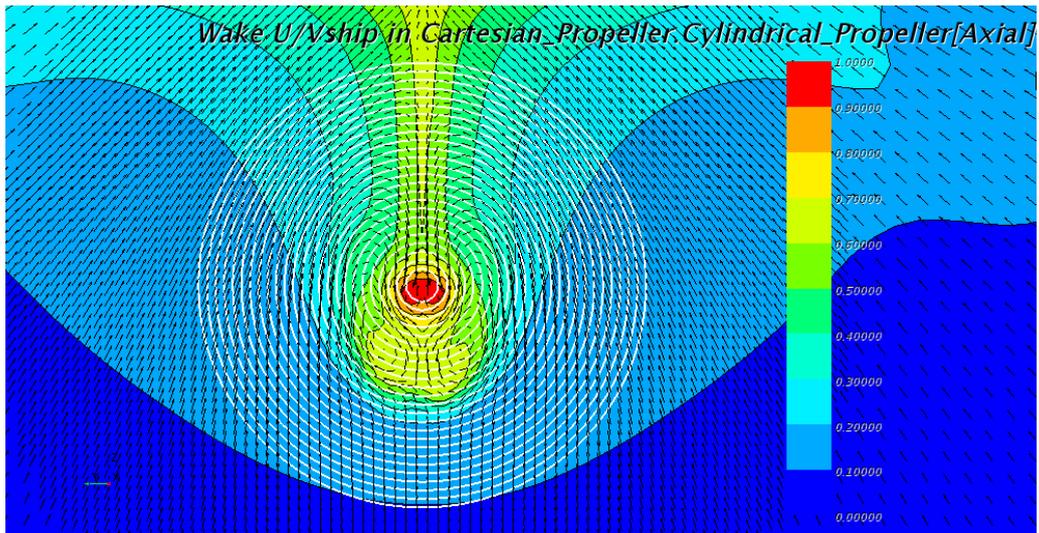
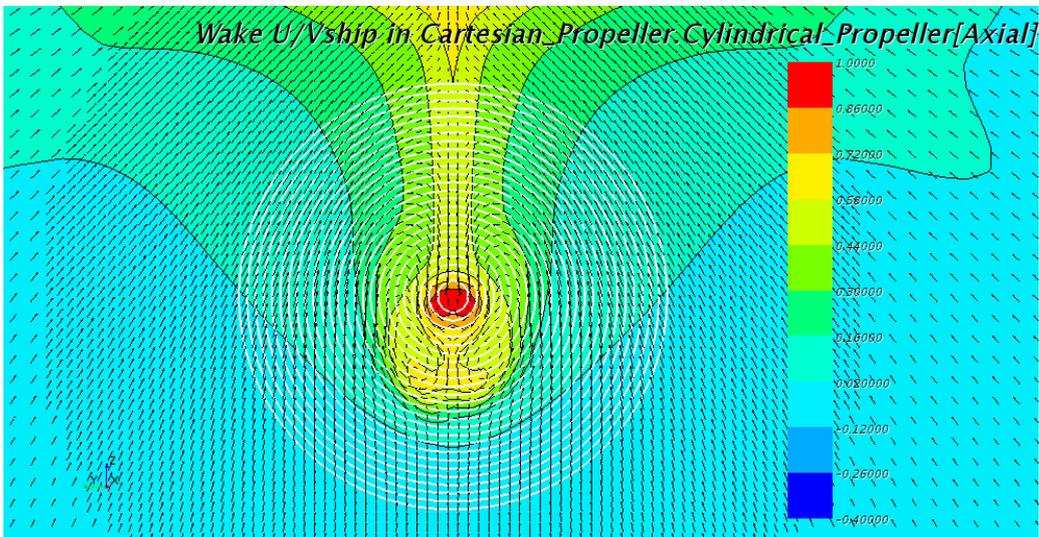
Full scale _ Fr = 0.0981 (10 knots)



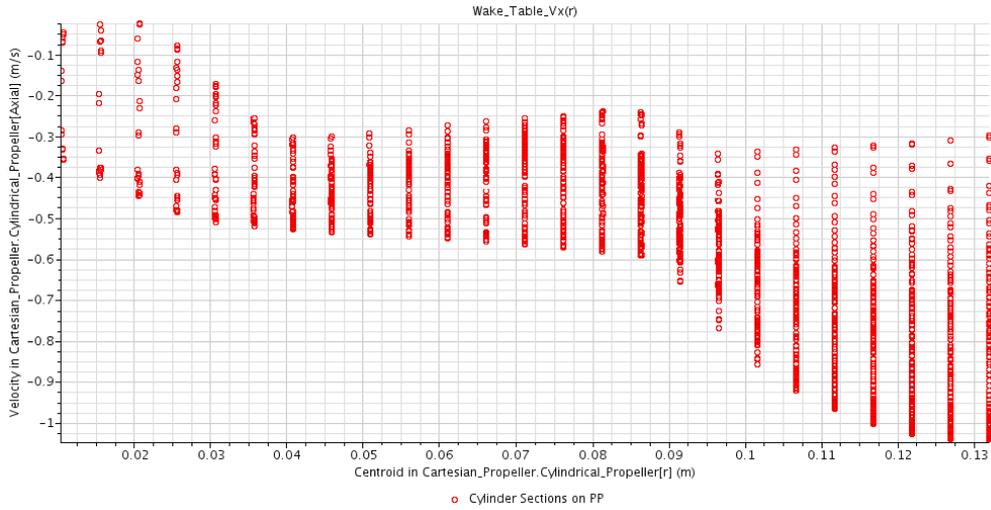
Full scale _ Fr = 0.1178 (12 knots)



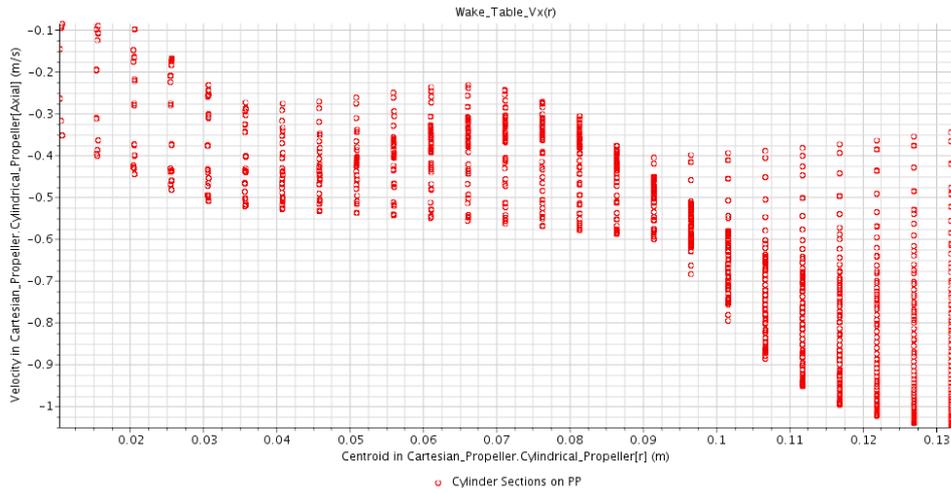
Full scale _ Fr = 0.1374 (14 knots)

Full scale $_Fr = 0.1472$ (15 knots)Full scale $_Fr = 0.1669$ (17 knots)

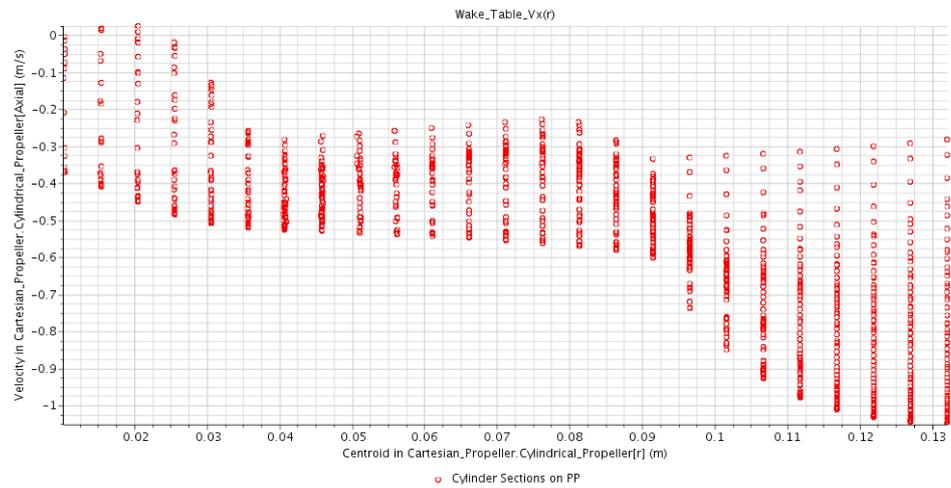
APPENDIX A7. WAKE TABLE



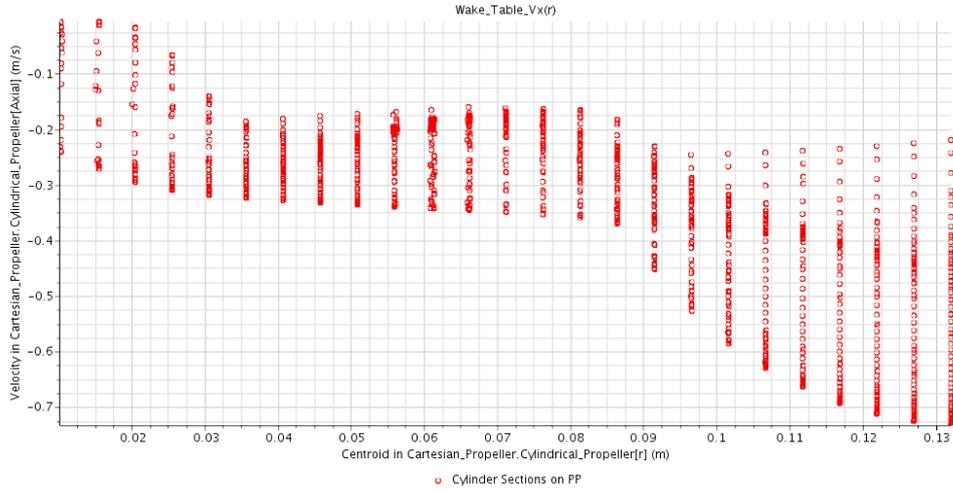
Model scale _ Fr = 0.1423 (14.5 knots)



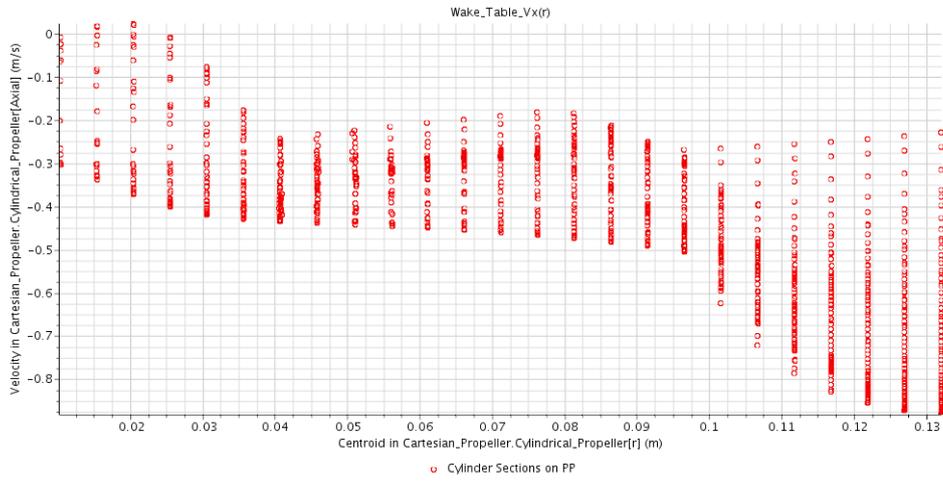
Model scale _ k-ε turbulence model _ Fr = 0.1423



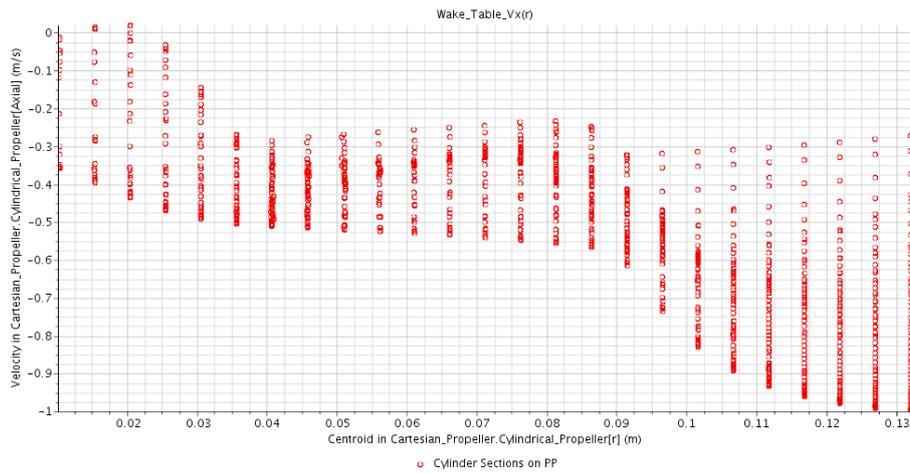
Model scale _ Half ship _ Fr = 0.1423



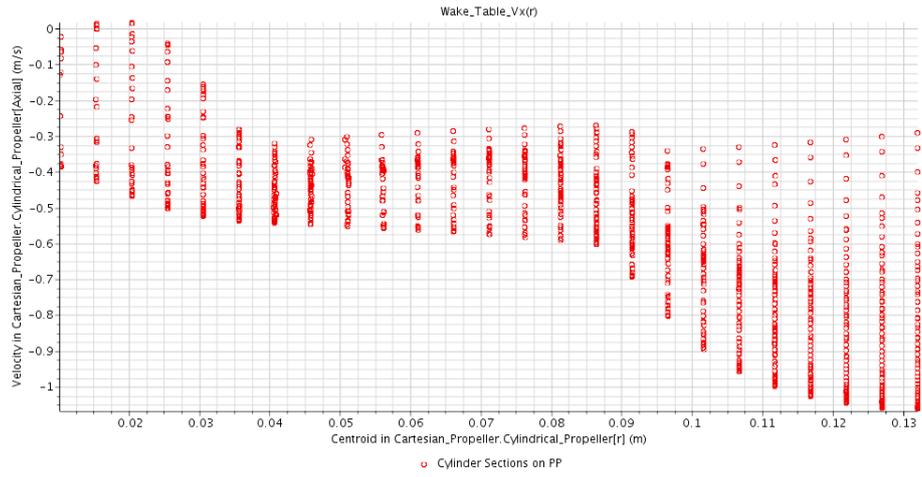
Model scale _ Fr = 0.0981 (10 knots)



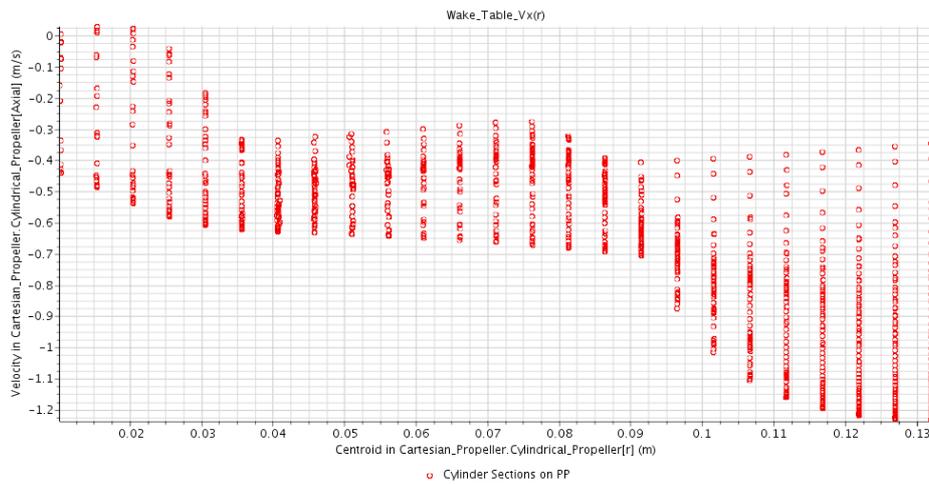
Model scale _ Fr = 0.1178 (12 knots)



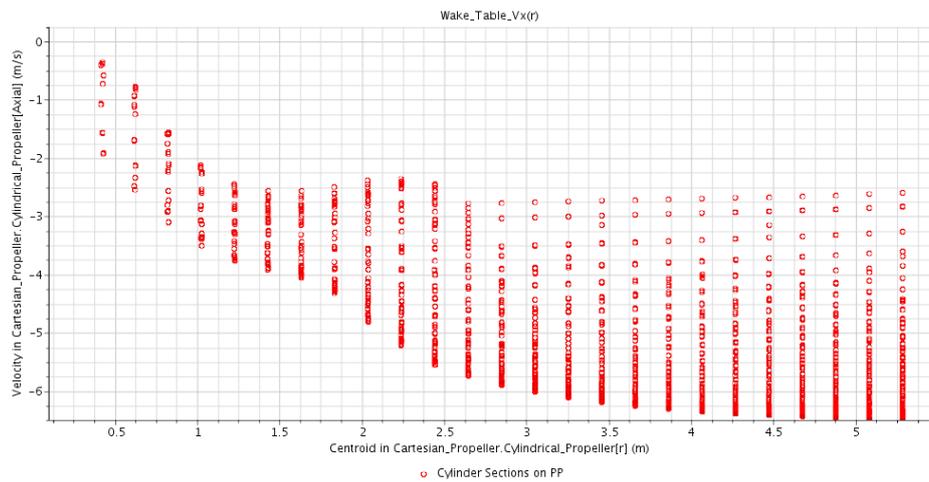
Model scale _ Fr = 0.1374 (14 knots)



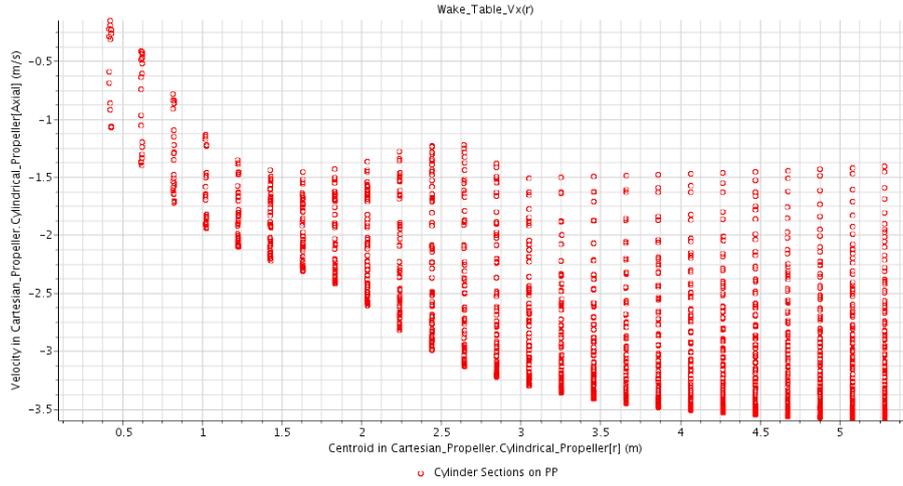
Model scale _ Fr = 0.1472 (15 knots)



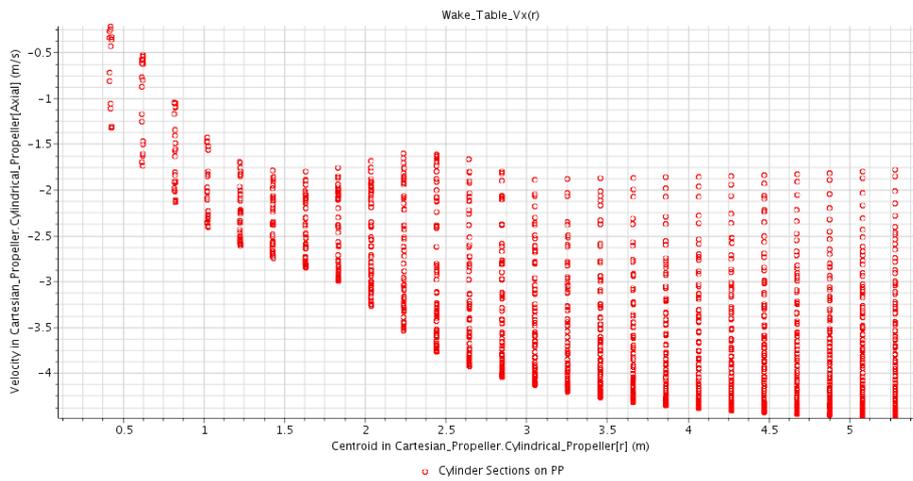
Model scale _ Fr = 0.1669 (17 knots)



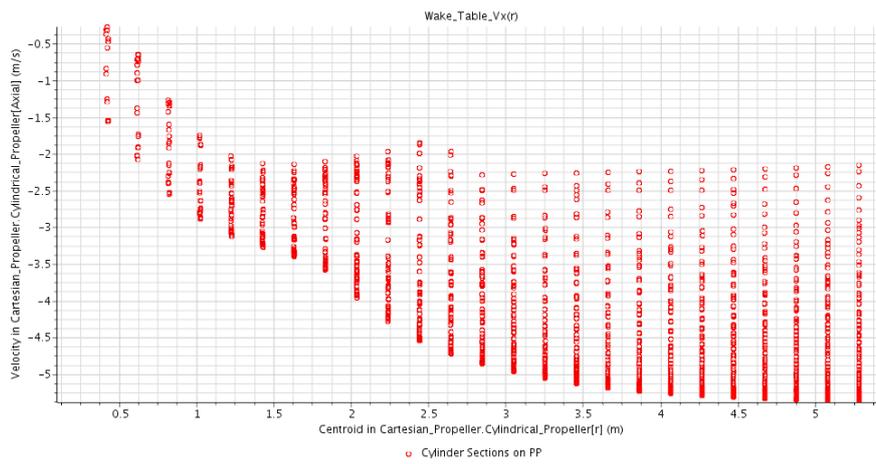
Full scale _ Fr = 0.1423 (14.5 knots)



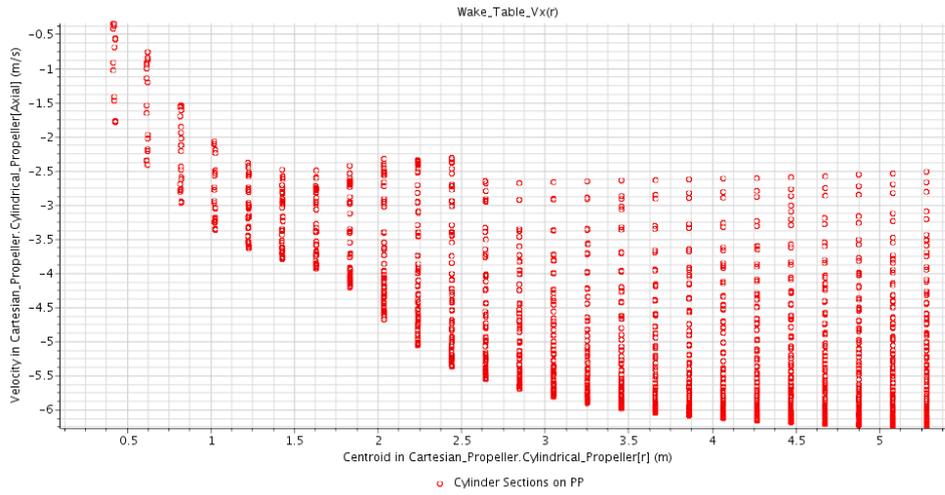
Full scale _ Fr = 0.0785 (8 knots)



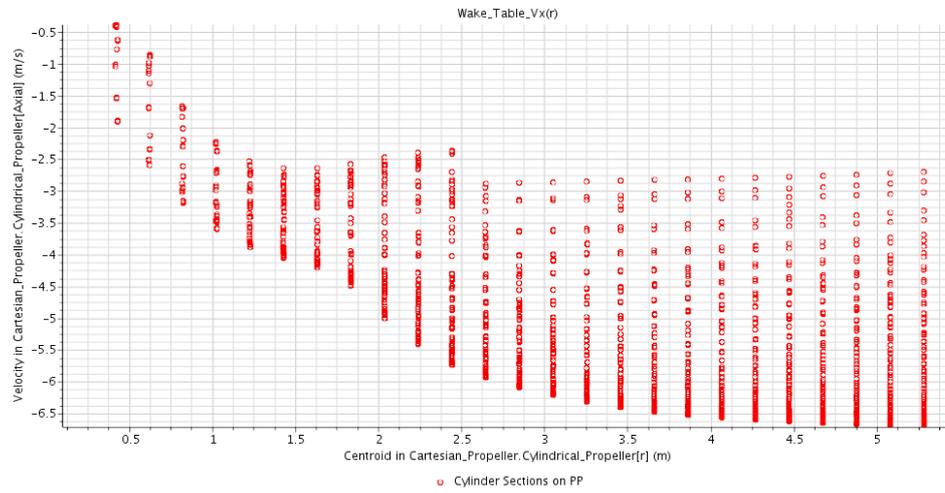
Full scale _ Fr = 0.0981 (10 knots)



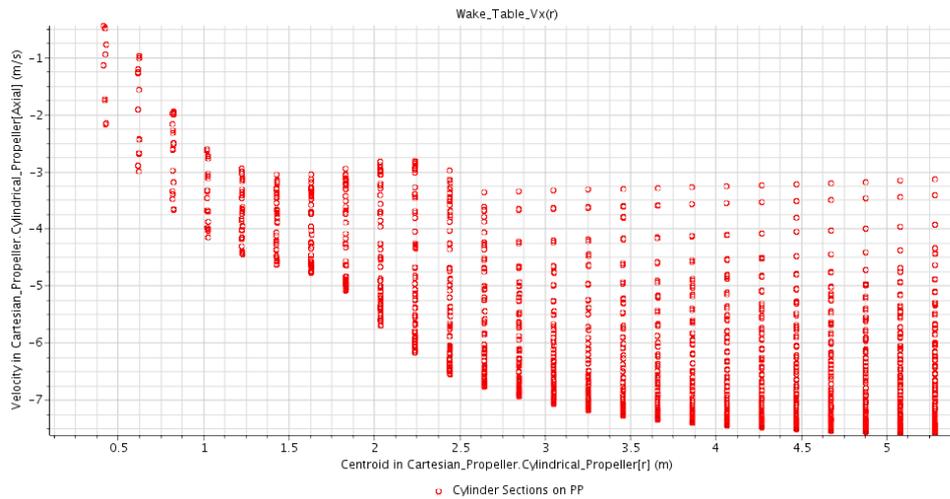
Full scale _ Fr = 0.1178 (12 knots)



Full scale _ Fr = 0.1374 (14 knots)



Full scale _ Fr = 0.1472 (15 knots)



Full scale _ Fr = 0.1669 (17 knots)

APPENDIX B. PROPSCALE OF MASTER THESIS

Scale effect on the wake field of a Single Screw Ship (PROPSCALE)

YUE DING

Ålesund University College

Supervisor: Karl Henning Halse

Summary

Now days, the CFD method has been become a famous approach that use to solve the problem involved the ship hydrodynamic problems. The objective of this master thesis is attempting to simulate a single screw stern ship sailing in the calm water with several different ship speed with the help of CFD program Star CCM+. Simulations contain both model scale situations and full scale situations. By analysis the results of the ship resistance coefficient and wake field of both cases, we figure out that the resistance and wake differences between model scale ship and full scale ship and what caused the scale effect between the real full scale ship and the model scale ship.

Master thesis mainly consist of four parts: background and methods, which is use to define the basic theoretical foundation of this thesis; Case study part introduces the model, mesh and physical parameters setting process in the CFD computer program: Star CCM+; Results part shows the results from simulations we get and Discussion part discuss the questions we encounter during the simulation. The entire simulation process in this thesis contains five phases: 1. Setup simulated models and meshes; 2. Setup physical parameters; 3. Verification of results; 4. Simulate all the test cases; 5. Analyze the results.

1. Introduction

For a long time, the ship hydrodynamics tests, such as the resistance tests are always based on the laboratory towing test. This method not only to spend lots of human and material resources, but also has lots of disadvantages, before the real ship sea trial, naval architects don't know if their results from towing test are correct enough. This master thesis paper aims to study how to use the new CFD method in a computer to arrive at test results that used to guide the hull form design.

In the past, ship designers most use parent ship transformation method to design a new type of ship hull. But after using CFD method, designers can design a totally new type of ship by determined the requirements. This method will bring ship design industry a good development.

Computational Fluid Dynamics (CFD) first appeared in 1960s with the development of computers and the rapid rise of disciplines. After nearly fifty years progress, this discipline has been quite mature. An important sign of maturity is that over the past decade, a variety of commercial CFD software is used in various industries. The performance and the range of applications are expanding. So far, the use of CFD technology has long been beyond the scope of traditional hydrodynamics and fluid engineering, such as aviation, aerospace, Ships, power, water conservancy, so as to extend to the chemical industry, nuclear energy, metallurgy, construction, environment and many other related fields.

In ship hydrodynamic area, CFD method mainly uses to solve the problems such as simulate the ship sailing status, compute the hydrodynamic coefficients of the ship and simulate the propeller. From the past literatures, we can know that for low single-hull vessels, using CFD technology to simulate can basically meet the needs of engineering applications and the satisfactory results. And in this thesis, we try to use this CFD method to simulate the sailing status of a single screw ship which named “Japan Bulk Carrier”.

2. Theoretical basis

2.1 Reynolds Average Navier-Stokes equations:

The equations discovered by Reynolds in 1883, and well known as “*Reynolds – average Navier Stokes Equations (RANS)*” is the physical foundation of the CFD calculations in Star CCM+. The equations describe four physical assumptions of fluid:

1. *The mass of fluid is conserved. (Continuity equation)*
2. *Momentum conservation law. (Newton’s second law)*
3. *Flow is incompressibility.*
4. *All the flow is turbulent flow.*

There four equations in RANS, which are

$$\nabla \vec{U} = 0 \quad (1)$$

For one direction such as x – axis the fluid velocity U:

$$\rho \cdot \left[\frac{\partial \bar{U}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{v}}{\partial y} + \bar{w} \frac{\partial \bar{w}}{\partial z} \right] = - \frac{\partial \bar{p}}{\partial x} + \frac{\partial}{\partial x} \left[\mu \frac{\partial \bar{u}}{\partial x} - \rho \overline{u'u'} \right] + \frac{\partial}{\partial y} \left[\mu \frac{\partial \bar{u}}{\partial y} - \rho \overline{u'v'} \right] + \frac{\partial}{\partial z} \left[\mu \frac{\partial \bar{u}}{\partial z} - \rho \overline{u'w'} \right] \quad (2)$$

2.2 Discretization Method

Discretization is a technique of conservation of general scalar transport equation into an algebraic equation that can be solved numerically. This control volume technique can be split into two parts:

- a) *Integrating the transport equation about each control volume.*
- b) *Writing a discrete analog of equation.*

This analog is an algebraic form of the transport equation which use to transfer information from one mesh cell to another. In general, for hexahedral cell we use in this master thesis, each cell has six neighbors. But on the sides of the computational domain, every cell will have only five neighbors, not six. So information of one face will be missed. In order to close up the system of equations. We have to provide this face with information, this information named boundary conditions.

2.3 Ship simulation in Star CCM+

STAR-CCM+ is a new generation of CFD solver software, which is developed by CD-adapco company by using the most advanced “CCM” (computational continuum mechanics) algorithms.

In ITTC towing test, naval architects towing the ship in the deep-water test pool. But in this master thesis, we use another to solve the problem: build a huge region to simulate the environment which contains both water and air, then we fix the ship in a reverse flow of the fluid flow without winds in the region, waves and currents. Shown in figure 1.

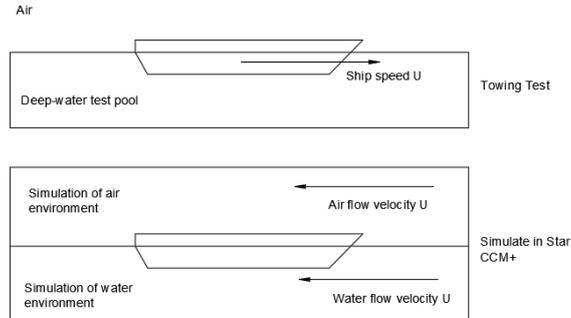


Figure 1. Simulation in Star CCM+

Should be noted that this method is only use to the simulation that ship is moving in a constant velocity. If ship has an acceleration, there will be added mass and that will be different between these two method.

3. Model & Mesh

3.1 Japan Bulk Carrier (JBC) Model

JBC (Japan Bulk Carrier) is a capesize bulk carrier equipped with a stern duct as an energy saving device. This type of bulk carrier designed by the National Maritime Research Institute (NMRI), Yokohama National University and Ship Building Research Centre of Japan (SRC), contain a ship hull, a duct and a rudder. The shape of JBC is a traditional large merchant ship, has a bulbous bow and a single screw stern.

In this CFD simulation, we simulate the condition without the ship appendix (Propeller, duct and rudder). Just calculate the influenced by ship hull in both model scale and full scale ship. A geometry of model scale ship is provided by “Tokyo 2015 - A Workshop on CFD in Ship Hydrodynamics”. The ship hull model contain five parts: Deck, hull, hull bottom, shaft_tube_end and transom. The model and full scale ship data shown in table 1:

		Full scale	Model scale
Proportion		1	40
Length between perpendiculars	LPP (m)	280	7
Length of waterline	LWL (m)	285	7.125
Maximum beam of waterline	BWL (m)	45	1.125
Depth	D (m)	25	0.625
Draft	T (m)	16.5	0.4125
Displacement volume	∇ (m ³)	178369.9	2.787029688
Wetted surface area w/o ESD	SW (m ²)	19556.1	12.2225625
Wetted surface area of ESD	SE (m ²)	745.2	0.46575
Block coefficient (CB)	$\nabla / (LPP * BWL * T)$	0.858	0.858
Midship section coefficient (CM)		0.9981	0.9981

LCB (%LPP), fwd+		-2.5475	-2.5475
Moment of Inertia	Kyy/LPP, Kzz/LPP	NA	
Design Speed	V _{design} (knots)	14.5	14.5
Design Speed	V _{design} (m/s)	7.4588	1.17934

Table 1. The dimensions of JBC

3.2 Mesh cells

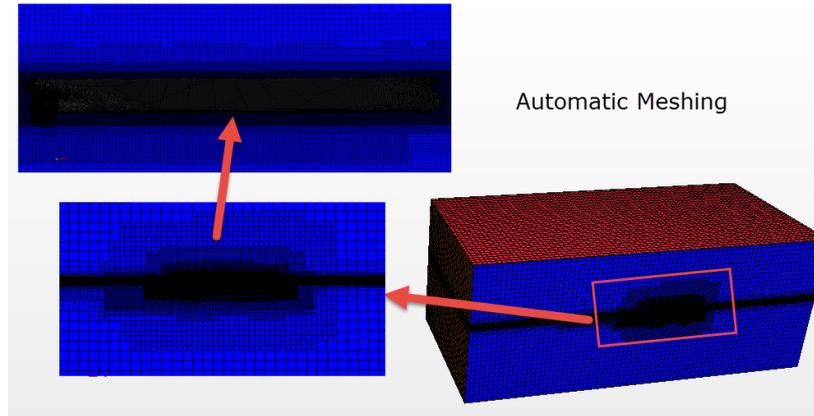


Figure 2. Automatic meshing

Figure 2 shows the auto-generated meshes in the simulation region. In the place that need higher accurate such ship hull and free surface, we use volumetric control method to build more details meshes. In table 2, the mesh settings for all simulations are exhibited.

	Ship status	Fr	Mesh basic size	Number of prism layers	Thickness of prism layers	Mesh cells number
Model scale	Half	0.0785	0.203 m	5	0.045675 m	2794083
	Half	0.0981	0.203 m	5	0.03045 m	2949867
	Half	0.1178	0.203 m	5	0.015225 m	3002787
	Half	0.1374	0.203 m	5	0.015225 m	3002787
	Half	0.1423	0.203 m	5	0.015225 m	3002787
	Full	0.1423	0.203 m	5	0.015225 m	5881132
	Half	0.1472	0.203 m	5	0.015225 m	3002787
	Half	0.1669	0.203 m	5	0.015225 m	3002787
Full scale	Full	0.0785	8.12 m	10	0.19082 m	6495589
	Full	0.0981	8.12 m	10	0.19082 m	6495589
	Full	0.1178	8.12 m	10	0.19082 m	6495589
	Full	0.1374	8.12 m	10	0.19082 m	6495589
	Half	0.1423	8.12 m	10	0.19082 m	6495589
	Full	0.1423	8.12 m	10	0.19082 m	6495589
	Full	0.1472	8.12 m	10	0.19082 m	6495589
	Full	0.1669	8.12 m	10	0.19082 m	6495589

Table 2. The mesh setting for model and full scale case

4. Physical parameters

4.1 All physical setting

After set the model and the mesh, we need to set the physical parameters that use to define the simulation region. Figure 3 shows the example of physical settings we set in some of the simulation,

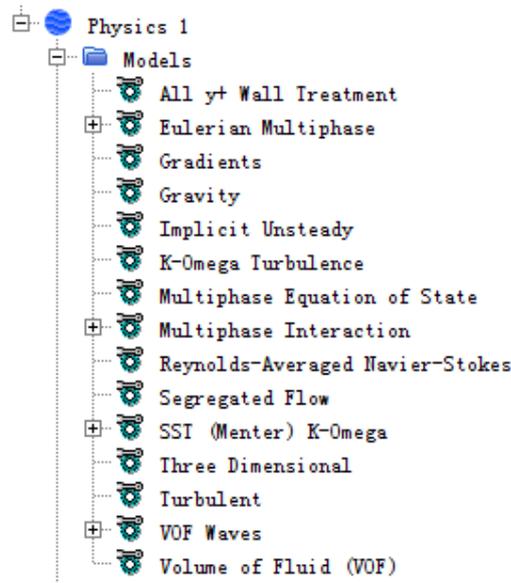


Figure3. Physical settings

4.2 Time step

In CFD, the Courant–Friedrichs–Lewy (CFL) condition is a necessary condition for convergence while solving certain partial differential equations (RANs) numerically by the method of finite differences. And time step of physical setting influence this CFL numbers. Time step means the real physical time for each calculation iteration step in CFD. As a consequence, the time step must be less than a certain time, otherwise the simulation will produce incorrect results. Table 3 shows the time step setting in our simulations.

	Model scale	Full scale
Time step	0.05s	0.15s

Table 3. Time step setting

5. Results

5.1 Ship resistance

The scale effect of the ship resistance coefficient is one of the most important parts that we focus on in this master thesis. As we mentioned in the background part, ship resistance always contains two parts: friction resistance and residual resistance. Friction resistance is due to the viscosity between water flow and ship hull, this type of force is related to the Re; the residual resistance is another type of resistance which is due to the generated wave of ship, this type of force is related to the Fr number. In order to intuitive to read the magnitude of resistance, we use the force coefficient to explain them:

$$C_F = 2R_F / (\rho S V^2) \tag{3}$$

Full scale fixed ship							
Ship Speed	8	10	12	14	14.5	15	17
V(m/s)	4.1152	5.1440	6.1728	7.2016	7.4588	7.7160	8.7448
Fr	0.0785	0.0981	0.1178	0.1374	0.1423	0.1472	0.1669
Re (×10 ⁹)	1.04	1.30	1.56	1.82	1.89	1.95	2.21
Expectation of Cf (×10 ³)	1.52	1.48	1.45	1.42	1.42	1.41	1.39
Cf (×10 ³)	1.57	1.51	1.46	1.42	1.41	1.41	1.38

Difference of C_f (%)	3.26	1.91	0.67	0.06	0.15	0.33	0.69
Expectation of C_p ($\times 10^4$)	9.53	9.48	9.78	9.17	9.36	9.45	10.45
C_p ($\times 10^4$)	4.65	4.80	4.50	3.80	4.12	4.10	5.24
Difference of C_p (%)	51.2	49.4	54	58.6	56	56.6	49.8
C_t ($\times 10^3$)	2.04	1.99	1.91	1.80	1.83	1.82	1.90
$Re=(V*LPP)/\nu$					LPP	280	m
					Water density, ρ	999.1	kg/m^3
Expectation of $C_f = 0,075/(\lg Re-2)^2$					Kinematic viscosity, ν	0.00000	m^2/s
					Dynamic viscosity, μ	0.00110	Pa-s

Table 4. Ship resistance coefficient

From the table 4, we can see that the friction resistance coefficient is significantly decreased with the increased of Reynolds number, the difference of C_f is very small, which is in the acceptable range. But for pressure resistance coefficient, the difference is quite big. The actual pressure resistance coefficients are nearly half smaller than what we expect, from 49.4% to 58.6%. This means the method that use ITTC towing test to estimate the full scale ship resistance does not fit this Japan bulk carrier. The reason for this result maybe complex, the most possible reason is due to the reduction of a large flow separation and vortex formation domain at the aftship. Which we will discuss later.

5.2 Wave pattern

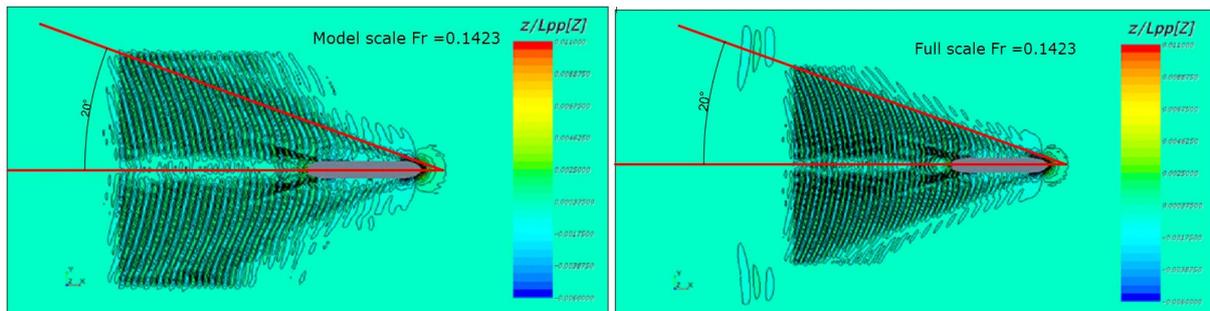


Figure 3. Comparison of Wake pattern

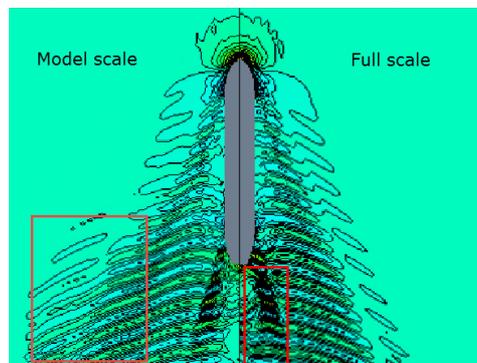


Figure 4. Comparison of wave patterns

Figure 4 shows the comparison of wave patterns between model scale and full scale ship. From figure 3 we have known that the degree of the wave pattern is nearly seen. In figure 4 we can see this phenomenon more intuitive. The shape and the wave length is also the same between model scale and full scale.

The difference come from free wave pattern region and turbulent wake region, which has been marked by using red blocks in figure 4. The difference in turbulent wake region is because the reduction of a large flow separation and vortex formation domain at the aftship. In wake field part we also can see the difference between model and full scale. The difference happens in free wave pattern region is we will discuss in discussion part.

5.3 Wake field

Figure 5 shows the wake field on the propeller plane at the aftship of model scale and full scale respectively. The cases choose are “Model scale _ Fr=0.1423” and “Full scale _ Fr = 0.1423”.

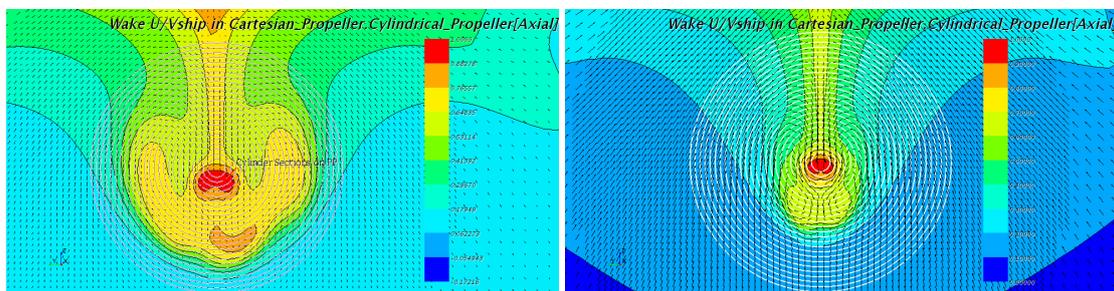


Figure5. The wake field on the propeller plane of model scale and full scale ship (Fr = 0.1423)

From the picture, we can find the huge difference between the model scale and full scale ship. For model scale ship, the wake field almost occupied the whole propeller plane, and more than half proportion of the wake fraction is larger than 0.5. In fact this value is not actually happening in the real ship, for single screw merchant ship, the wake fraction's range always is 0.2 – 0.3. In the figure 5.23, for full scale ship, only small region has large wake numbers in the middle of the propeller plane, and the wake field of full scale ship has a much smaller wake fraction.

Average nominal wake fraction is the mean value of the wake fraction of these data.

Speed (knots)	8	10	12	14	14.5	15	17
Fr	0.0785	0.0981	0.1178	0.1374	0.1423	0.1472	0.1669
Model scale	0.6736	0.5658	0.535	0.5157	0.5194	0.5145	0.5089
Full scale	0.316	0.314	0.312	0.3099	0.31	0.31	0.3048

Table5. Average wake fraction

Table 5 indicates the mean wake fraction value, and the plots of these data presented in figure 5.25. The conclusion are:

- 1) There is a huge scale effect of wake fraction number of model scale and full scale ship. The average wake fraction of model scale ship is much bigger than the full scale ship.
- 2) With increasing Fr number, the wake fraction is decreasing for both model and full scale ship.

6. Discussion

6.1 Cp influenced by bulbous bow

In this part, we discuss the trend of resistance coefficient. In previous chapter, we notice that with the Re increasing, the Cf number decreased smoothly. But for Cp curve, the situation become different. Shown in figure 6.

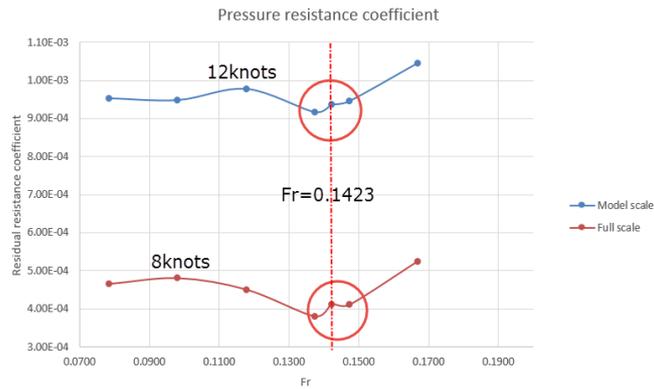
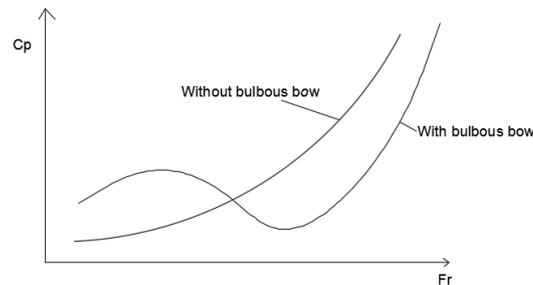


Figure6. Pressure resistance coefficient

From figure 6, we can find that, at the beginning of the C_p curve, the pressure resistance coefficient go up and then reach the peak of the highest place: 8 knots for full scale and 12 knots for model scale; And then, the C_p number start to sharp drop until get the lowest point around the design speed ($Fr = 0.1423$); After this, the C_p number start to raise once again until the largest speed of the ship. In fact, this trend of the C_p curve is fit our knowledge about the pressure resistance coefficient. When a ship has a bulbous bow, the real pressure resistance of the ship is going up and down again and again with the speed increased, which can be indicated in figure 7.



Figur7. The curve of the C_p with/without bulbous bow

A bulbous bow is a protruding bulb at the bow of a ship at the underwater part. The bulbous bow modifies the way the water flows around the hull, reducing drag and thus increasing speed, range, fuel efficiency, and stability. This is no surprise that the naval architects set 14.5 knots as the design velocity: total resistance equals the sum of the C_f and C_p . C_f value is decreasing with the speed increasing, and the C_p number get the lowest value around $Fr = 0.1423$.

6.2 The causes of scale effects

A boundary layer is the layer of fluid in the immediate vicinity of the ship surface where the effects of viscosity are significant. The thickness of the boundary layer is generally defined as approximately 99% of the vertical distance from the outer flow velocity at the surface of the object, which is shown in figure 8a. Only the water in the boundary layer consider the viscosity between water and ship surface. When the Reynolds number increased, the boundary layer become thinner, which we can see from equation 4, which describe the thickness of the boundary layer for turbulent boundary layers:

$$\delta \approx 0.382x/Re_x^{1/5} \quad (4)$$

Where x is the distance downstream from the start of the boundary layer; Re is the Reynolds number. When the boundary layer become thinner, the number of water that need to consider the viscosity is also reduced, and the friction resistance coefficient is decreasing at the same time.

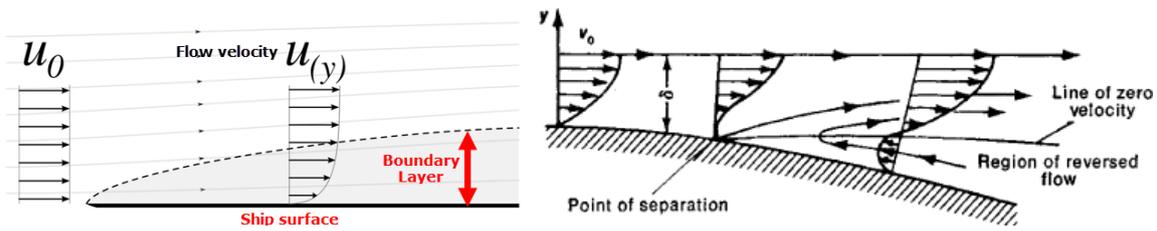


Figure8. a) Boundary layer b) Flow separation

When velocity of fluid flow increased or the flow encounter the sharp mutations at ship hull surface, because of the reversed flow, the fluid flow start to separate from the ship-hull surface (water will not flow along the shape of ship surface), and these separated flow generate vortex at aftship, which can be seen in figure 8b. These separate flow and vortex will cause the pressure difference between foreship and aftship, what means, the pressure resistance coefficient C_p is increased due to the flow separation. We compare the simulation between model scale and full scale, the Re number of full scale ship is much higher than model scale. As we mentioned before, the boundary layer is thinner in full scale ship, and has a smaller C_f number compare with the model scale.

Also, because the boundary layer is thinner than the model scale, the flow separation is generally delayed in full scale and vortices are much weaker. This phenomenon can be observed in figure 9.

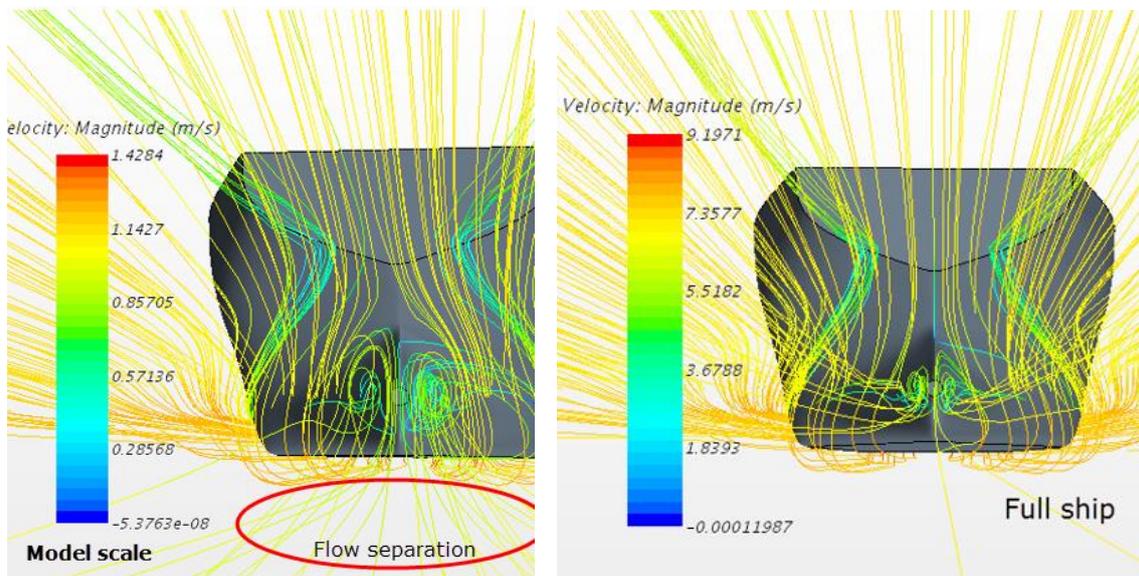


Figure9. Streamline at aftship of model and full scale ship ($Fr = 0.1423$)

Figure 9 indicate that, for model scale ship, the water flow has a larger flow separation after ship stern, the flow from ship bottom go straight line directly but not along the ship surface. Compare with it, full scale ship's flow is along the stern surface. So we can make a conclusion here that because the full scale ship has less flow separation and vortex, the pressure resistance coefficient is of course smaller than the model scale ship. What we observed is 50% nearly.

Scale effect of the wake fraction is caused by same reason that the boundary layer flow separation. We are focussing on the wake field of figure 5 again and see the arrows in the figure. These arrows represent

tangential velocity of the water flow. We find that, because the flow separation, the flow in model scale has very small tangential velocity that go straight line and flow to after ship directly. Not very much water flow to the propeller plane along the ship surface. In opposite, most the flows in full scale flow to propeller plane, a larger velocity compare to model scale of flow at full scale aftship. This is why the wake fraction of full scale ship is smaller than the model scale. The conclusion is:

- a) Compare to the model scale simulations, the Reynolds number of full scale ships are larger;
- b) Boundary layer of full scale ship is thinner than model scale ship; Which cause friction resistance coefficient C_f lower than model scale;
- c) Flow separation is generally delayed in full scale and vortices are much weaker than model scale.
- d) C_p of full scale ship is smaller than model scale ship
- e) Wake fraction is much smaller than model scale ship because less vortices and flow separation

6.3 Results Influenced by mesh & physical parameters

Friction resistance coefficient & Prism layers

Before finally determine all the parameters setting for full scale ship simulation, we performed several failure simulation that gets the really weird results. One of them is the friction resistance coefficient.

In fact, this is the most stable variable in these simulations, the change is very slightly when setting is changed. Where the greatest impact is prism layers setting on this resistance, or we can say the Y^+ number.

Y^+ number should be kept in the range of 30 -100, or at least 30 -500 around ship hull in these simulations. This is the suggestion we get from the Norwegian Marine Technology Research Institute (MARINTEK) and test conclusion we make after series of attempts on the prism layers setting. Otherwise the friction resistance coefficient will be more different from the expectation numbers.

Pressure resistance coefficient & VOF Waves Damping length

C_p is quite sensitive with VOF Waves damping length. In fact we still not figure out why this should happen. Slightly change of this number will let the pressure resistance coefficient changed in large numbers. At last we accept the suggestion from “MarineTek” that use the distance between the ship Central Plane (CP) and the Side boundary. For model scale ship, we use 13 meters and for full scale ship we use 520 meters.

Waves & Parameters setting

For some simulation cases, such as “Model scale _ $k-\epsilon$ turbulence model, $Fr = 0.1423$ ” case and the cases of lower speed of ship. The wave patterns generated looks wired when the others results get good number. Shown in figure 10.

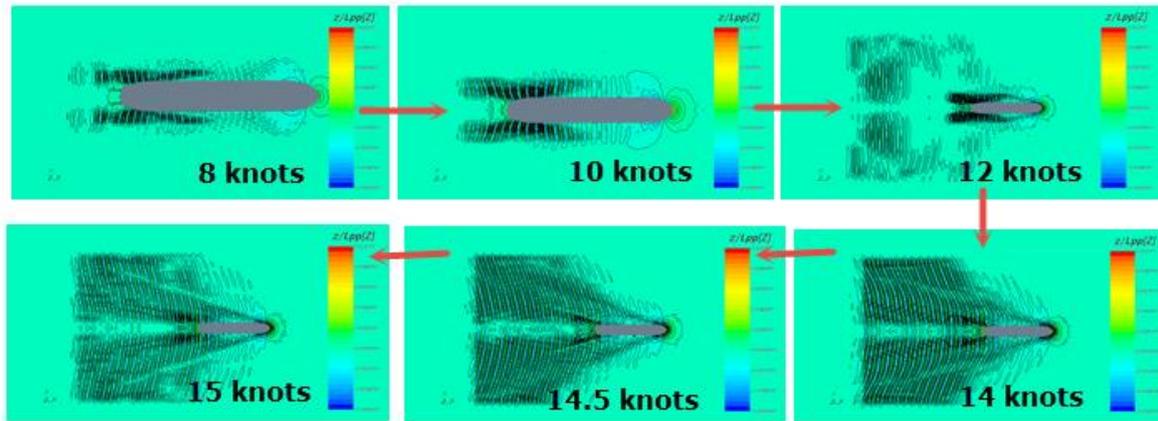


Figure10. Wave patterns for different speed ships (Model scale)

From the figure we find that, with speed increasing, the wave “grows” gradually until it get a complete Kelvin ship waves pattern when ship sailing at 14 knots. We didn’t do more works to focus on this phenomenon, but we guess it should be caused by the bad physical and mesh settings, consists of:

- The mesh size at the free surface region is not sufficient. The future work we should try more details mesh cells at the free surface region;
- Time step is not small enough: even we keep the CFL number smaller than 1 at most of simulation region, but at the place that around the ship, the CFL is still larger than 1. The future work should try smaller time step number to simulate;
- Turbulence models: by comparing the results difference between SST $k-\omega$ turbulence model and $k-\epsilon$ turbulence model, we find that these parameters influence the wave patterns very much. Try more turbulence model in the future works to be seen the difference.

7. CONCLUSION

By simulate 17 different ship sailing model in Star CCM+, we basically grasp the hydrodynamic characteristics of this type Japan Bulk Carrier ship for both model and full scale, compare the results of them and try to figure out the causes of scale effects on a deeper level. In addition, we summarize some precautions for the mesh and parameters setting and simulation after experiencing several failures of simulations

During the mesh setting process, we should notice that the prism layers’ setting is a quite important concerns that we need to try several times to find the best setting thickness to keep the Y^+ number around the underwater ship hull in the range of 30 -100. This number is actually can influence the accuracy of friction resistance coefficient.

“VOF waves damping length” is the parameter that impact the pressure resistance coefficient. We use the distance between the ship Central Plane (CP) and the Side boundary. On model scale ship, we use 13 meters and for the full scale ship we use 520 meters.

The scale effect for this type of ship is conspicuous, not only for the friction resistance coefficient C_f which is related to the Reynolds number, but also the pressure resistance coefficient C_p . For full scale ships, C_p number are 50% smaller than the model scale ship. The wake field we focus on in this master thesis, the differences are also huge.

The underlying cause of the scale effect is the difference of Reynolds number between model scale simulations and full scale simulations. With increasing of Reynolds number, the boundary layer around ship surface become thinner which cause the friction resistance decreased. The thinner boundary layer cause a delayed flow separation and weaker vortex, and then, this will cause smaller C_p number of full scale ship than model scale ship. The delayed flow separation let more water flow to the propeller plane along the ship hull surface, which cause a smaller wake fraction of full scale ship than the model scale ship.

At last, we also can make the conclusion that because the big scale effect between model scale ship and full scale ship, we can't use the model towing test to measure the ship hydrodynamics of this type of ship directly. More simulation we need to be done to estimate the factor of the pressure resistance coefficient between model and full scale in the future work.