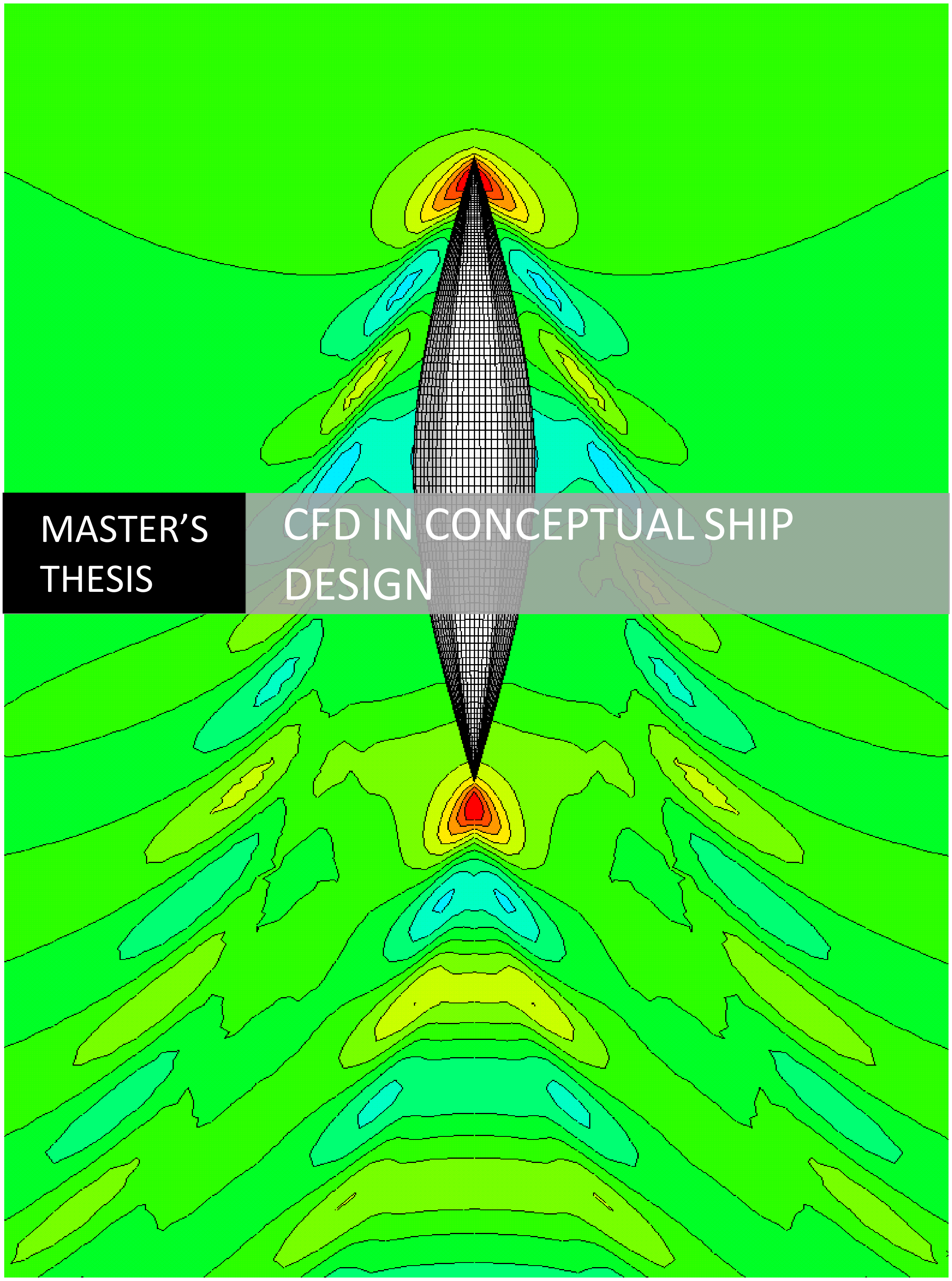
NTNU – Trondheim
Norwegian University of
Science and Technology

2011



MASTER'S THESIS

CFD IN CONCEPTUAL SHIP DESIGN

MSc in Marine Technology | Stud. Techn. Petter Olav Vangbo

Master’s Thesis in Marine Systems Design

Stud. Techn. Petter Vangbo

“CFD in conceptual ship design”

Spring 2011

Background

Traditionally, research on hull shapes has focused on optimizing for still-water conditions, design cargo loads and design speed conditions. New research should focus on including realistic operation profiles to design more "robust" hull shapes which are not necessarily optimal only for still-water, design cargo loads and design speed conditions, but optimal under a realistic set of operational profiles/scenarios with significant variations in external conditions (e.g. market fluctuations, fuel price).

Objective:

To make a robust hull shape the designer must have extensive knowledge in how the global parameters affect the performance. CFD could be used to gain more knowledge in how to make a robust hull shape. CFD have been more and more implemented in ships design, in what way is it used and how can it be applied in conceptual stages of ship design.

Scope and main activities:

1- The first step would be to find the decided CFD program that could be easily used for a wide variety of shapes. What kind of CFD method to use is of great importance and there should be used some time in discussing what to choose.
2- How can CFD be used in conceptual design exploration. Discuss different methods of conceptual design and value of CFD in design.
3- Create a robust hull design approach by using the selected CFD program. Use different draughts and different speed as input variables.

Modus operandi

At NTNU, Professor Stein Ove Erikstad will be the responsible advisor. Professor Sverre Steen will serve as an additional advisor.

The student may contact STX Europe, Project in Ålesund for input on how the early stage hull design process is performed in industry today. At STX, Henning Borgen will be the contact person.

The project is connected to a industry project where DNV, Grieg, Marintek and other industry partners are participating. Travel expenses may be covered by this project. In particular, DNV

Research and Innovation may provide additional support during the project. Evangelos Boutsianis will be the contact person at DNVRI.

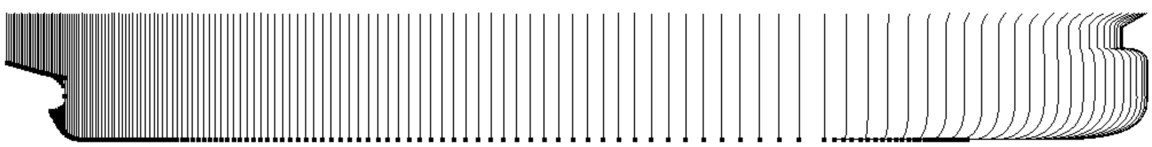
The work shall follow the guidelines given by NTNU for the MSc Project work

PREFACE

This is a Master's Thesis in project ship design at the department of Marine Technology NTNU (Norwegian University of Science and Technology). The origin of this thesis was a suggested topic that Professor Stein Ove Erikstad presented to me as a project in the fall of 2010. After doing the project I wanted to continue to work on the topic in a Master's Thesis. Though the scopes and activities may have changed during the work throughout the year, the main objective has always been to investigate the use of Computational Fluid Dynamics (CFD) in conceptual design.

The Master is connected to an industry project where DNV, Grieg, Marintek and other industry partners are participating. This project has its goal to investigate robust ship design and robust hull shapes. I have therefore included some basic knowledge of robust design and also tried to implement it in my research examples.

CFD is a computer tool which requires experience and knowledge. I had not much experience in using CFD, but had knowledge of fluid dynamics which is the physical basis of CFD. In my thesis I had to require a CFD program tool which had a low user interface so I would not use too much time in learning the program. Also because CFD should be used in a conceptual stage I would also need the program tool to have some sort of integration to optimization and Computer Aided Design (CAD). The program that I have been using is a Framework tool with CAD and optimization (FRIENDSHIP SYSTEMS, 2009). It is tightly integrated with the CFD program SHIPFLOW developed by Flowtech int. The program was not available and had to be required. This took time and I got the program late in the working process. I had therefore not the time to investigate the program thoroughly, and the research examples are therefore easy and simple. The activities were then changed towards a more literature study of conceptual design, optimization, CFD, robust design and modeling design. Also more focus was turned more to the process of CFD and simulation driven design.



It is expected that the reader has some knowledge in naval architecture, fluid dynamics and optimization to comprehend the content of this texts. But I have tried to make it as general as possible.

I would like to say my gratitude to my responsible advisor Professor Stein Ove Erikstad and also Professor Sverre Steen. Also I have had some discussions with Evangelos Boutsianis from DNV Research and Innovation of robust hull optimization which I am thankful.

Attached to the cover is a CD that contains the results from the three research examples done in the program and the two CAD models of the hulls that is used in the simulations.

Trondheim 14.06.2011

____

Petter Olav Vangbo

SUMMARY

Computational Fluid Dynamics (CFD) has been around for many years. It is a computer tool that can be used to find the hydrodynamic fluid performances. In ship design it is used in a wide area from smoke propagation to resistance estimations. It is however in resistance estimations that CFD have had most focus and research.

There are many tools a designer can make use of nowadays. Most of the tools are computer based. This is optimization algorithms, computer aided design (CAD) and computational fluid dynamics (CFD). Using the tools should shorten the time of ship design and make better solutions. I have used a computer tool that mixes optimization with model variation (CAD) and verification (CFD). My conclusion is that it is a powerful tool to use, but should be handled with care. Few variables in the optimization process are important.

Conceptual design methodology could be broken down to two outer ranges; point based design and set based design. The methods are quite different when approaching a complex design problem. There seems to be some favor in set based design when coming to a global 'optimized' solution to the design problem. More knowledge is gathered in set based design before deciding the final requirements and parameters. This is especially in new developing design where little knowledge is produced in the past.

CFD is a broad term. There is many different methods and area of use. In this thesis I will break it down to two terms; potential codes and RANSE codes. Potential codes are easy, robust and well developed. RANSE codes are difficult, takes a lot of time and not so well developed. Potential codes are used in areas where turbulent flows are not present, while RANSE codes are used when it is present and important to the result.

If designing new innovative hulls CFD should be used earlier in the design process and with a simulation driven design approach. Simulation driven design could be used with potential codes or RANSE codes. To have a high value rate of the modeling potential codes should be used when many sets of variation I needed and turbulence is not important to the answers. RANSE code should be used when turbulent flow is important to the answer, but must be done with few sets of variations because of high computational effort.

If designing a more standard ship, CFD should be used in a modeling design approach to verify the performance estimations that have been done earlier in the process.

LIST OF ABBREVIATIONS

AP – Forward Perpendicular

FP – After Perpendicular

BEM – Boundary Element Method

CFD – Computational Fluid Dynamics

DWL – Designed Water Line

FVM – Finite Volume Method

IMO – International Marine Organization

ITTC-57 – International Towing Tank Conference 1957

MARPOL – International Convention for the Prevention of Pollution from Ships

NURBS – Non-Uniform Rational B-Spline

NS – Navier-Stokes

RANSE - Reynolds-Averaged Navier-Stokes Equations

RFR – Required Freight Rate

SOLAS - The International Convention for the Safety of Life at Sea

CONTENTS

FIGURES

INTRODUCTION

How to achieve the best design and at the same time to be an innovative ship design company or ship yard? Or said in another way how to produce the best solutions for the costumer? This is the problem that every ship design company is facing in order to be competitive in a constantly more globally challenging market.

In the past ship design relied much on knowledge from past ship designs. The designer copied the solutions from the past design and progress was slow. Innovation was hard because of the uncertainty if the design would give the expected results and performances. In the last twenty years computers have made it possible to visualize and calculate performances. Innovation was made more possible because of increased knowledge provided earlier in the design stage by computer design and simulations. One of the tools that have been more and more included in ship design is Computational Fluid Dynamics (CFD). This tool can provide insight into the ship hydrodynamic performances. Traditionally ship performances have been found from empirical hull series and propeller series to estimate power and performance in conceptual design. Now CFD can provide knowledge and results that before was provided by model hull series and propeller series.

But what is a good solution or design? How can a design company say with certainty that this is the best solution to the customers' demands? One could argue that optimization could give the needed certainty for the designers that the best solution is found, but it all depends on how smart the optimization process is organized and the practical level of sophistication for each simulation. The challenge lies in the investigation of choosing objectives, finding the right constrains, weighing the objectives up against each other, finding the right parameters to use and last knowing what to want out from the optimization process. Optimization is more and more implemented into ship design. It is a useful tool in the decision making process, but even more powerful together with CFD.

According to (Harries, 2008) we are going from modeling design towards simulation driven design. It means that we drive the final solution not from the modeling, but out of the calculations or simulations. Computer Aided Design (CAD) is how you model the design and visualize the change. CFD is the calculations that show the result of the simulation. Put the two together and you get simulation driven design. Simulations driven design would intend that the designer will get feedback from the changes that are made and get knowledge and information. Simulation driven design is considered to be the key in innovative product development. The reason for this change from modeling design towards simulation driven design is the increase in computational efficiency. We can now use heavy computer tools, like CFD, to find the performances of the design. While integration of modeling and simulation is done quite well in structural mechanics this is not yet the case in fluid dynamics.

The increase in fuel price has gotten much attention the last years. It seems that the price of bunker oil will have a turbulent future and that shipping companies must have a strategy of how to meet the future. Focus is therefore turned to how to make a ship as fuel-efficient as possible for the future. The ship should meet a more demanding market with more fluctuations and uncertainties. The focus on getting a ship more fuel efficient has always been an issue for ship-owners and ship operators. There have been developed many methods to meet the uncertain future; like steaming the ship, schedule optimization algorithms, bunker hedging and speculation in using the cheapest fuel which exist. Most of this focus has been to optimize or to deal with a ship that already exists, not to optimize it already on the drawing board for uncertainties. Focus has therefore in the last years been pointing towards producing more robust ships.

Robust ship design has not been addressed much in design communities. It is just in the last years that research and development have taken place. How can CFD be a helpful tool to explore robust designs?

1.1 Aim of research

The aim of this research is to get to some conclusion of how CFD can be used in ship design with favor of conceptual design stage. There are many tools that can be provided in an early stage of design like; optimization algorithms, computational fluid dynamics (CFD), computer aided design (CAD) and there exists also a great diversity in design methodology. How will CFD fit into different design methods? How can it be used in the best way, how will the trade-off between time and accuracy be treated?

1.2 Approach

My approach will be to investigate the new tools that a designer has at hand. I will go through basic knowledge of CFD and resistance to get background knowledge of the physical aspects of CFD and how it is used in resistance estimations in Chapter 2 and Chapter 3. What is state-of-the-art in ship resistance prediction will be discussed in Chapter 4 followed by a discussion of time and CFD in

Chapter 5. When using CFD one should always have in mind that CFD is a computer tool that contains uncertainty and error. Chapter 6 will go through some basic principles of error which will also be discussed in relation to the research examples in Chapter 11.

To put CFD and simulation driven design into a context I will also discuss some aspects of conceptual design. I will divide the conceptual design methodology into two methods that represent two outer ranges of design; point base and set based design. This will be investigated in Chapter 7 followed by Chapter 8 that discusses optimization and design.

Simulation driven design includes CFD and CAD. The collaboration between the two and how to rate the value of modeling will be discussed in Chapter 9.

I will make use of simple examples of how a robust ship design investigation can be done in an early stage of a conceptual design stage. Some aspects of robust design will then be discussed in Chapter 10 as an introduction to the research examples. Chapter 11 contains the research examples. I have made three different simulating models to show a simple first approach of robust hull investigation.

The conclusion in Chapter 12 is based most from my experience of using the computer program and from investigating the basic concepts of CFD, conceptual design and optimization.

2 CFD – BASIC CONCEPTS

Computational fluid dynamics starts with one basic equation; the Navier-Stokes (NS) equation. This equation is a coupled, non-linear partial differential equation that describes the flow in and out of a control volume. In this equation the first assumption is that the fluid is incompressible, which leads to another equation; conservation of mass.

The Navier Stokes equation:

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t}+\mathbf{v}\cdot\nabla\mathbf{v}\right)=-\nabla p+\mu\nabla^2\mathbf{v}+\mathbf{f} \tag{2.1}$$

Conservation of mass:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{2.2}$$

The NS-equation is a fluid conservation of momentum of Newton's second law. In this case it is in cartesian coordinates, but it may be in polar or spherical coordinates. ρ is the density, μ is the frictional coefficient which represents the viscosity and $\mathbf{f}$ is external mass forces i.e. gravity.

The full NS-equation for all the fluid when looking at a ship is incredibly intensive to calculate. Faster and faster CPU speed and multicore processors will maybe make it possible one day to calculate it in a reasonable time. But still then you have not taken into account that the sea is not still, it is also moving, which will result in a double up problem. First you have to calculate the given sea state, and secondly to put a ship into the sea and see what differences it will give to the fluid. You would also have coupled effects between the two, the fluid will give the ship different motions, and that motions will again affect the fluid etc. But for engineering purpose it may not be of interest to make such a fully developed model.

To bring the NS-equation closer to a numerical solution, time averaging is introduced for the frictional term. This is called the Reynolds-averaged-Navier-Stokes-equation (RANSE). Often when talking of Navier-Stokes you would really mean RANSE. Further simplification is to narrow it down to a conservative form. That means to leave out the coupled relation. The equation (2.3) will be given an additional term on the right side for the RANSE simplification. The derivatives of the velocities will also be simplified, they will be time averages. The time averaging eliminates the turbulent fluctuations in all terms except the Reynolds stresses.

RANSE:

$$\rho\bar{\mathbf{v}}\cdot\nabla\bar{\mathbf{v}} = -\nabla\bar{p} + \mu\nabla^2\bar{\mathbf{v}} + \bar{\mathbf{f}} - \rho\nabla\bar{\mathbf{v}}'\bar{\mathbf{v}}' \tag{2.3}$$

Further simplification is to neglect the frictional force. This is called the Euler equations, but is of little use in ship designing problems, because they use nearly as much computational time as RANSE but gives no more information than a potential flow problem. But they are popular among aerospace engineers when calculating foils, where viscous flow is not so important. We say that the flow is inviscid when neglecting the frictional force.

Potential flow is the next step of simplification. The flow is now non-rotational. The velocities are now coupled by the potential. This has been the most used application in the past decades, because of the less computational time and the robustness. But potential flow is not very accurate for calculating forces on the hull where it is a lot of turbulence, like the stern part, appendices, propulsion etc.

The velocity potential equation:

$$\frac{\partial^2\phi}{\partial x^2} + \frac{\partial^2\phi}{\partial y^2} + \frac{\partial^2\phi}{\partial z^2} = 0 \tag{2.4}$$

The potential flow equation or Laplace equation is given above, and if there only exist gravity forces as the external forces the equation can be written for a stream line as:

$$\frac{\partial\phi}{\partial t} + \frac{1}{2}(\nabla\phi)^2 - gz + \frac{1}{\rho}p = const. \tag{2.5}$$

This is the well-known Bernoulli equation.

In my master thesis I will try to generalize CFD into two methods; potential and RANSE codes. The increase in user-knowledge is increasing when approaching the RANSE calculations. There is a great difference between the two models; diversity, complexity, time etc. Potential codes are used in a wide variety of the design areas and don’t need much modification depending on design areas. RANSE codes needs modification regarding turbulence models and when coming to the choice of techniques and grids. The areas that have had most attention in CFD and ship design are performance calculations in resistance and propulsion.

2.1 Techniques

There are basically two CFD techniques that are used to solve the equations; Boundary Element Method (BEM) with panels and Finite Volume Method (FVM). Both of them divides the fluid in a large number of elements that leads to a large number of equations. Change is given by boundary conditions at the surface and of the ship hull. Boundary Element Method (BEM) is used for a potential flow and Finite Volume Method (FVM) is used for RANSE calculations. This is very roughly said, there are other methods, but I will not go further into it this, because these two methods are mostly widely used in academia and commercially.

Making the grids or the panels is a complex task, and there are some right ways of doing it and some wrong. In the last couple of years there has been much focus on auto grid generation for both panel meshes (BEM) and grid volumes (FVM). For RANSE calculations the grid or volumes are much more complicated in geometric shapes than panel meshes. Taking a resistance prediction example from (Harries, Tillig, Wilken, & Zaraphonitis, 2011), the potential flow analysis required a body mesh with 1150 panels and a free surface mesh with 7175 panels. For the RANSE viscous calculation there were created 1.7 million volume cells with a longitudinal stretch towards smaller cells in the skeg region. This is a typical number of grids or cells. The increase in number of cells will increase the calculation time, but the accuracy will increase with increasing numbers of cells in the grid. Figure 1 shows an example of a complex grid. It is a grid volume method (FVM) with asynchronous cells which get more complex near the body surface (here the plain surface).

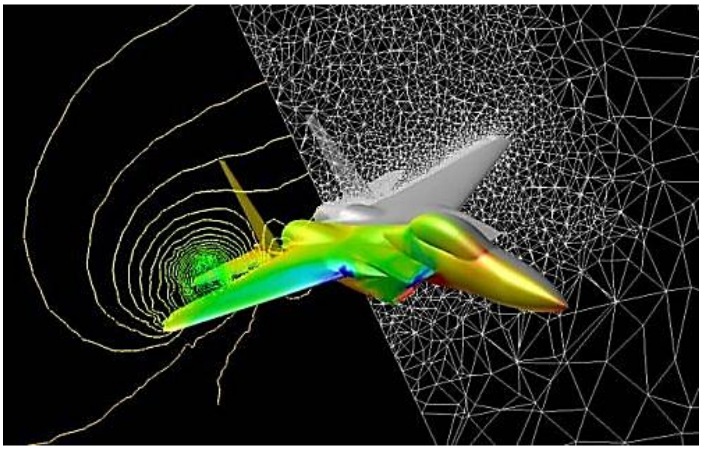


Figure 1: Complex grid generation (CFD Technologies)

The reality of CFD for RANSE calculations is also a bit more complex. The RANSE equations require external turbulence models. There are whole conferences that are dedicated to the turbulent flow; the reason is that turbulence is not fully understood. There are also more techniques which are described here and numerous ways of simplification and linearization that makes it possible and more computationally easy to calculate the fluid. The accuracy of RANSE is very dependent on the turbulence model that is chosen.

3 GENERAL SHIP RESISTANCE

The general way of calculating resistance is to decompose it in different components. There are different ways that this has been dealt with, but one way is to decompose the resistance as described in figure 2 (Larsson & Baba, 1996). This is resistance in calm water and that is the normal approach of calculating the needed power of a certain speed decided by the owner. This is a way the designer can focus on one part of resistance and what influences that part and not to think of interaction between them. As Volker Bertram says it (Bertram V. , 2000); 'its separation into components is merely a hypothesis to facilitate analysis, but the theoretically cleanly divided resistance components interact and require a comprehensive approach for a completely satisfactory treatment.'

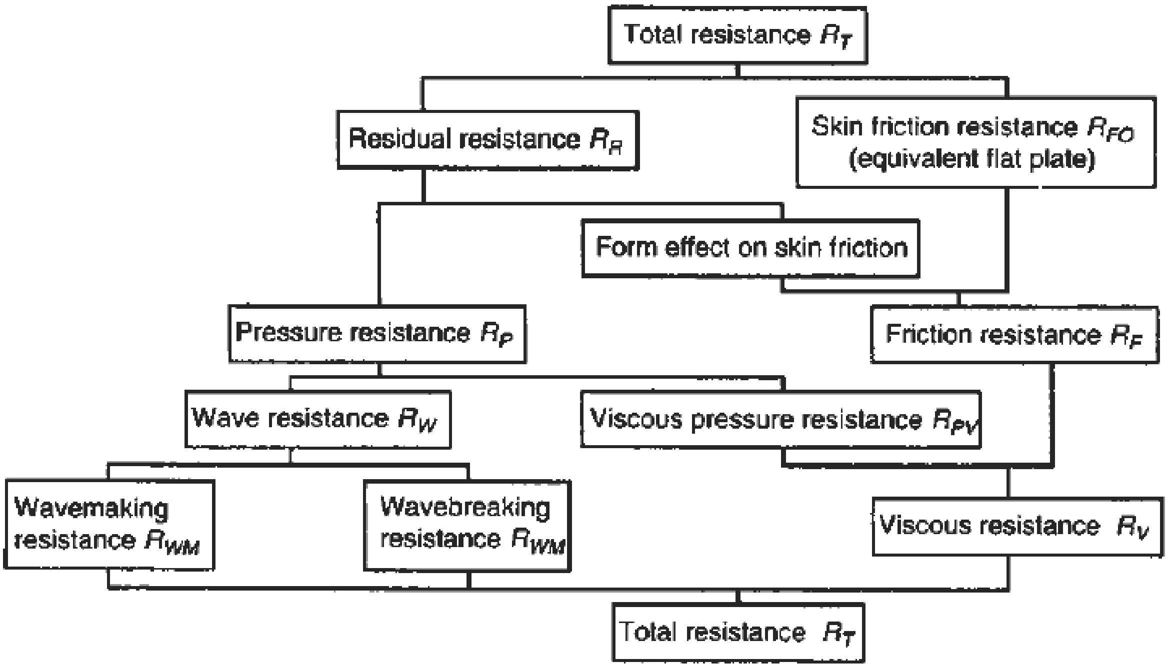


Figure 2: Decomposing resistance into components (Bertram V. , 2000)

From the figure 2 you can see that you can roughly divide resistance into viscous resistance and wave resistance.

3.1 Viscous resistance R_v

The viscous resistance is a function of Reynolds number. Reynolds number is a way of quantify the turbulence of the flow.

$$\mathrm{Re} = \frac{VL}{\upsilon} \qquad (3.1)$$

V is the speed of the fluid, L is the length and υ is the kinematic viscosity.

Water has a given viscosity, because of this there is a frictional force between the fluid particles. On a ship there will exist a boundary layer all around the ship where this frictional forces are present. A boundary layer is where the fluid goes from disturbed by the body to undisturbed. At the hull the fluid particles will stay attached to the wetted surface of the hull, and the difference in velocity will give higher shear stresses and give turbulent flow if the difference is high enough and acts ‘long’ enough. The turbulent flow takes energy, and this is a big part of the frictional resistance. Frictional force are normally calculated over the wetted surface and corrected by a form factor because of 3D effects.

For a ship the boundary layer is not far away from the skin in the front part of the ship, but when changing rapidly in form in the stern of the ship the boundary layer will not follow, and there will be a wake behind the ship. The wake is very turbulent. Because of a highly turbulent flow and different velocities there will exist a surge behind the ship that will give a negative force to the speed direction. It will also lower the propulsive efficiency because of the flow entrance to the propeller.

3.2 Wave resistance R_w

Wave resistance is a function of Froude number or a non-dimensional way of quantifying the speed with the hull. The number was developed to get around the scaling problem of gravity forces and inertia forces in model tests.

$$Fn = \frac{V}{\sqrt{gL}} \qquad (3.2)$$

V is the speed of the mass, g is the gravity force and L is the length of the given object.

In water there exists a boundary from water to air. A floating body that interacts with the water will give disturbance of the fluid. The shape of the hull and the speed will create velocity differences. This will give pressure differences and thus waves, which propagate away from the hull. The way wave resistance has been dealt with is to decompose it into primary and secondary wave system. The primary is formed by an ideal fluid (potential codes) and using Bernoulli's equation you can find the difference in velocity and thereby also the pressure. Where the speed difference is biggest there will also the biggest pressure be and also the highest wave. This primary wave system is speed independent. The meaning of this is that the location of extreme points will not change with speed. The highest wave will be in the same position relative to the ship. The height of the wave will be quadratically dependent on speed.

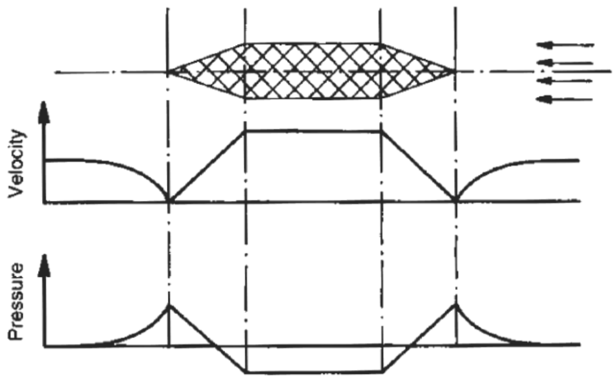


Figure 3: Primary wave system (Bertram V. , 2000)

The secondary wave system is diverging waves and transverse waves made by the shape of the hull, see figure 4. Secondary wave systems can also be divided into bow wave, waves made by the front and back curvature and stern wave. The creation of waves is strongly dependent on the geometric form near the free surface.

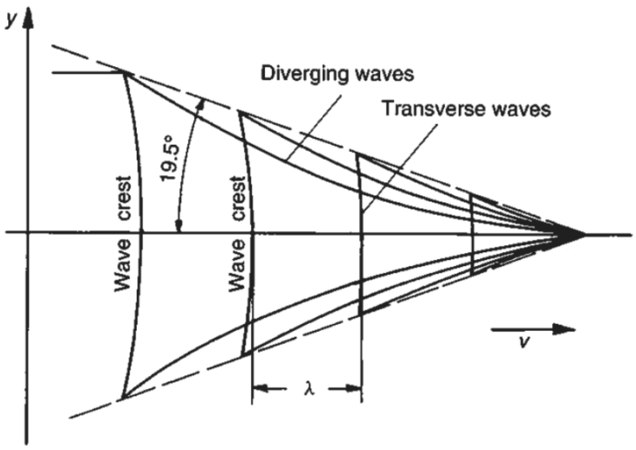


Figure 4: Secondary wave system (Bertram V. , 2000)

In reality the waves will break and change the pattern. There will also exist a dynamic trim and sinkage that will change the wave system. This can be treated by doing a non-linear approach which allows the hull to trim and sink. The solution will then be an iterative process. In wave resistance viscous effect is not very important and the calculation of the wave propagating system is close to satisfactory. But still there will be interaction effects that are hard to deal with when using potential theory.

3.3 Interactions between ship and propulsion

The propulsion system interacts with the ship hull. When talking of a propulsion system the rudder will be a part of this. The presence of a propeller will give an increase in the flow field in the aft body and thus increased frictional resistance. A propeller will also decrease the pressure and the inviscid resistance will also increase. The flow field is changed by the hull, but likewise the flow field is changed because of the presence of the propeller system. The way it has been dealt with is to separate the two problems and to introduce an efficiency factor. CFD may contribute much in understanding the interaction between ship and propulsion system, but to have a sufficient model you would need a good turbulent flow to fully utilize the CFD RANSE codes.

The choice of what kind of propulsion system will give a big difference in the performance, i.e. the resistance. Mainly a single screw or a pod/azimuth system is the most dominant choice depending on ship type. More than one propeller is typically used if the draught is not sufficient to contain the diameter of one propeller. A single –or two screw systems have usually a direct shaft into a gearbox and further to the diesel engine. A pod or azimuth has a diesel-electric propulsion system. How to choose a system depends on many variables like economy, practicality, space etc. The system is decided very early in the design stage. How the hydrodynamic performance will change is usually not considered when choosing the propulsion system, CFD can help to make better choice and trade-offs because of earlier knowledge of the performances.

3.4 Other resistance components

3.4.1 Appendages

The resistance of appendages is predominantly governed by viscous forces. Usually appendages are tested with CFD or models separately from the ship, but interaction between the two will not be taken care of and large errors are common. A ship with many complex appendages will be difficult to calculate. Appendages can be of importance of the overall resistance and there may be an increase according to V. Bertram (Bertram V. , 2000) of 1-6% for transverse thrusters. Bilge keels can contribute 1-2% and power shafts can increase by as much as 20%.

3.4.2 Shallow water

In more shallow water the frictional resistance will increase, but this effect is of more interest when looking at maneuvering capabilities.

3.4.3 Wind

Wind forces are normally not accounted for in merchant ship design, with exception of high speed craft.

3.4.4 Roughness

Roughness of the hull will increase the frictional resistance. Marine growing will normally be the problem.

3.4.5 Seaway

The added resistance of ship in seaway is difficult to address. Normally you will have a certain sea state prediction based on statistics, and find out from this the seaway resistance. But according to (Bertram V. , 2000) accuracy of sea state statistics introduces a larger error than the actual computational simulation. For added resistance the global parameters are important and bow shape especially. The size of the ship is generally more important than the ship shape. But this is an area that may be of interest for the practical use of CFD in the future, but models have to be created and made simple for the average designer.

4 STATE-OF-THE-ART CFD IN SHIP DESIGN RESISTANCE

There are many areas where CFD plays an important role in ship design. Hydrodynamic performance with seakeeping, resistance and maneuvering are the main areas of research and usage. Other areas are dynamic loads; slamming, sloshing, whipping effects in tanks or loads on the hull. Dynamic stability of ships, ship appendages and cavitation problems, ventilation, aerodynamic of superstructure, smoke propagation and fire simulation are other areas where CFD also plays an important role of design.

Figure 5 shows two examples of how CFD is used. In the figure to the left, CFD is used to provide pressure of the body surface such that cavitation can be found at the rudder and the propeller. In the right a temperature simulation is produced in a container vessel and how the temperature is divided.

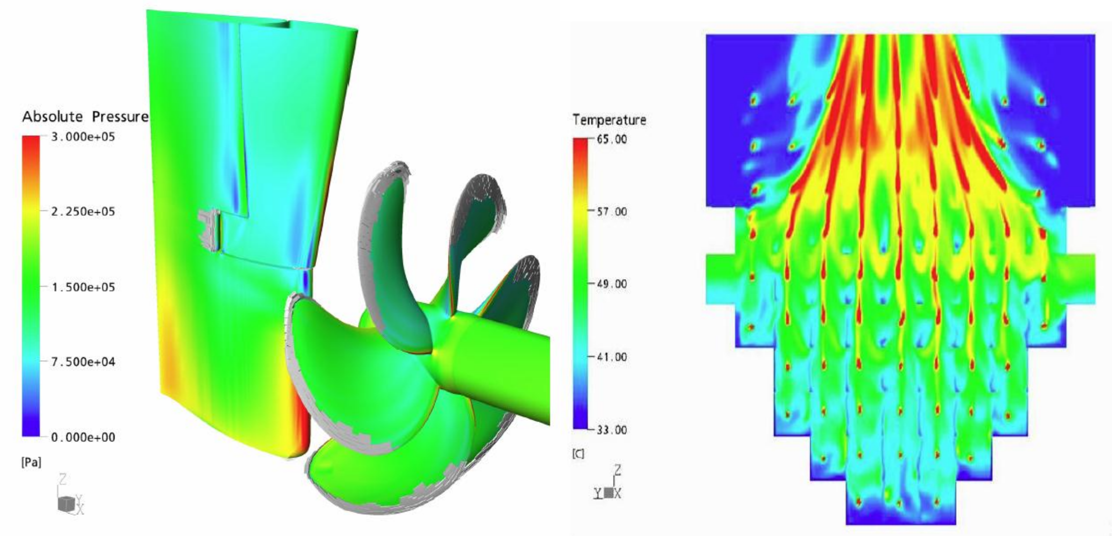


Figure 5: Cavitation of a propeller to the left and temperature distribution to the right (Moctar, 2008)

4.1 Resistance prediction

In this master thesis I will focus most on the hydrodynamic performance part and the estimation of resistance. CFD is not a simple tool to understand without the knowledge of mathematics and physics. I have tried to explain a simple way of how CFD works, but how it is used is not the same in all performance applications, CFD has to be 'tuned' according to what usage it will take, i.e. the elements or grids in the equations that will be the basis of calculations are quite different in shapes and numbers according to what area that is investigated of the ship.

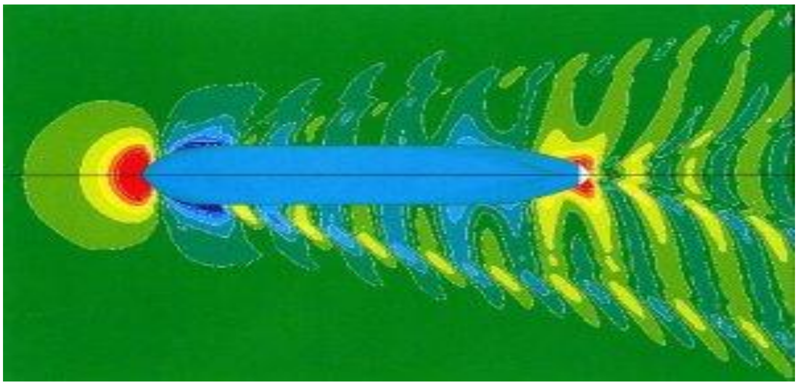


Figure 6: Forebody shape optimization using SHIPFLOW. Significant improvement in wave pattern at Fn=0.16 for an optimized forebody design (top) as compared to the original one (bottom) (Flowtech)

In ship resistance, potential codes are used together with a panel method to solve a numerous of areas. Potential codes accounts for around 50 % of CFD analysis and are expected to be the workhorse until at least 2020 (Couser, 2002). The main areas are investigation of bulbs, forebody, streamlines and free surface wave resistance, as seen in figure 6. RANSE codes with turbulence models are typically used in areas where turbulence and viscous effects are of importance. In ship design RANSE Codes are typically used to find flow distribution into the propeller, figure 7, or to find the drag.

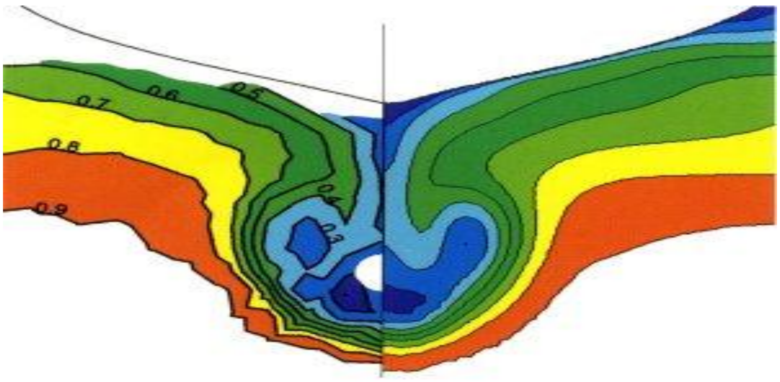


Figure 7: SHIPFLOW wake computation validation. Experiment (left) versus calculation (right) (Flowtech)

A State-of-the-art way of finding still water resistance is to both use viscous (RANSE) and potential flow computations. It is called a zonal approach developed in the program SHIPFLOW (Flowtech int., 2010). The ship is divided into two parts; front and aft. The potential flow computations are used to solve the non-linear wave resistance problem with free sinkage and trim for the entire domain. The frictional resistance is found by a thin boundary layer computation for the forebody and a RANSE computation in the aftbody of the ship with frictional and viscous pressure resistance. The propeller is usually modeled as a force actuator disk. This way of modeling can get quite close to the actual towing resistance. Some of the difficulties are the choice of turbulence models and errors and uncertainty. To get the power prediction it is required an individual estimate of propulsive efficiency.

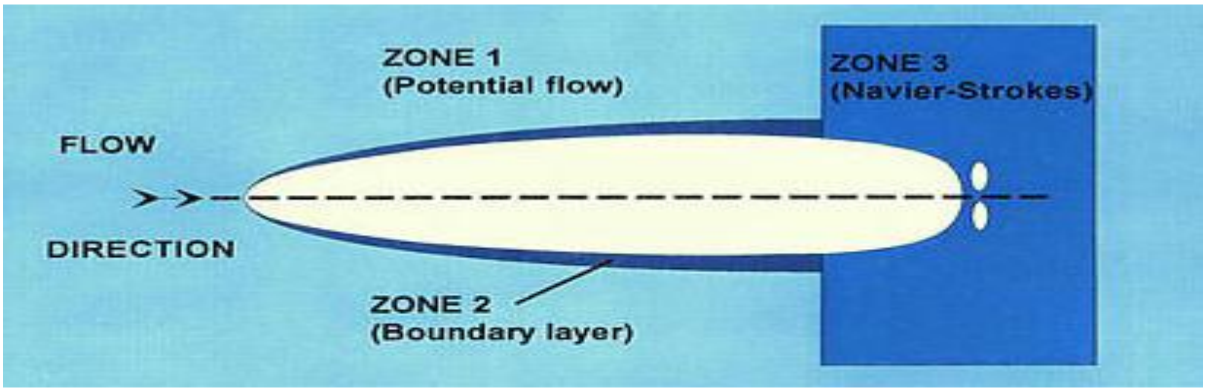


Figure 8: Zonal approach for towing resistance prediction (Couser, 2002)

5 TIME AND CFD

CFD has different approaches according to how accurate you would like the results to be; potential codes contra RANSE codes. The time difference between the two is quite different, RANSE codes needs much more computational effort than potential codes, but in reward you get more accurate results. Potential codes have a longer history of usage and it has therefore more experience than RANSE codes. It is in the last years that RANSE codes have been implemented in programs and used in a reasonable way in ship design. Though the RANSE codes give more accurate results, it does not mean at all that the use of potential codes is going towards an end. It is a long way before the RANSE codes is as robust, easy to use and use a reasonable time before it can take over the tasks of the potential codes.

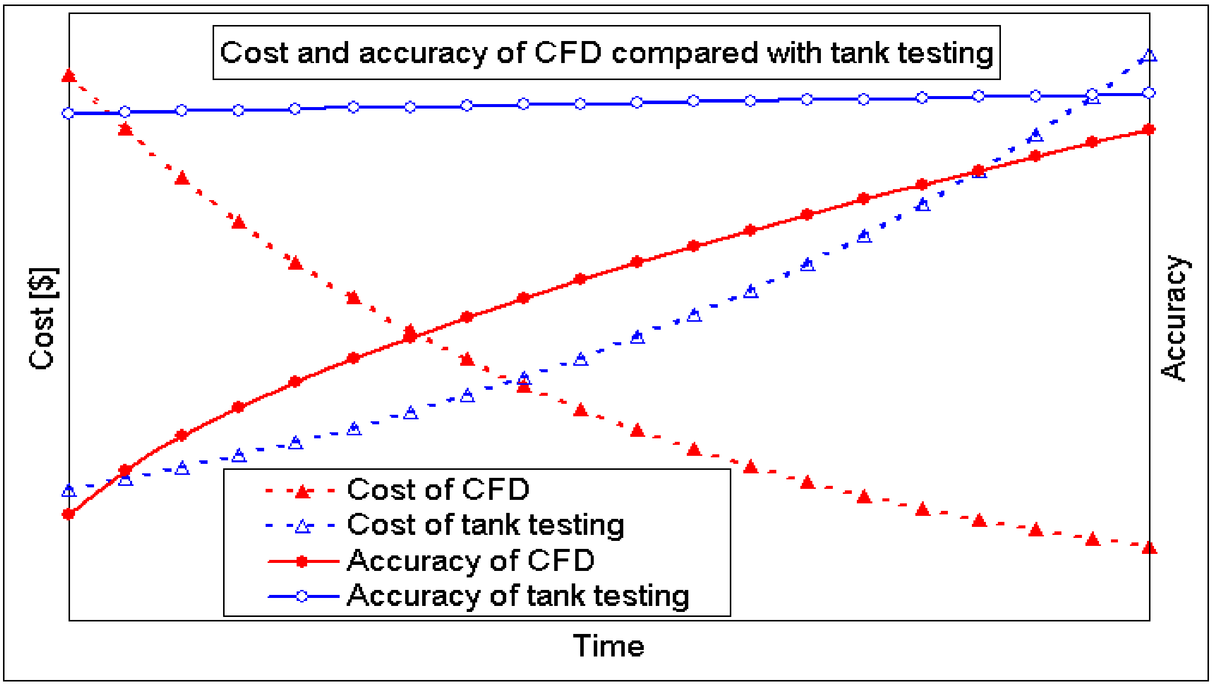


Figure 9: Cost and accuracy of CFD versus time (Couser, 2002)

Figure 9 shows a prediction of cost and accuracy development (Couser, 2002). The figure 9 is just a generalized prediction of the development, and should not be taken as an exact representation, i.e. the cost of model tank testing is a bit pessimistic. The time axis will also be different from the cost and the accuracy. It will also look different from the CFD method and area of use. It is just an example of how to illustrate the development in time. But if talking of resistance prediction we are now in the right of the figure 9 where the accuracy of CFD is a bit under the model tank testing. The cost is also much less in CFD than model tank testing. This is only in the area of resistance predictions; in seakeeping the CFD method is still more to the left in the figure and model testing is the favored choice (Steen, 2011)

Because RANSE codes use more time potential codes is the favored choice in optimization simulations. In figure 10 you see that the rate of time is much higher for the potential codes then RANSE Codes. In the future the rate of RANSE Codes will get higher because of the higher computational effort of computers, but it will still be more complex then potential codes. It will always take a certain amount of time in modeling and setup. Error and uncertainty will also increase and thus the time as well.

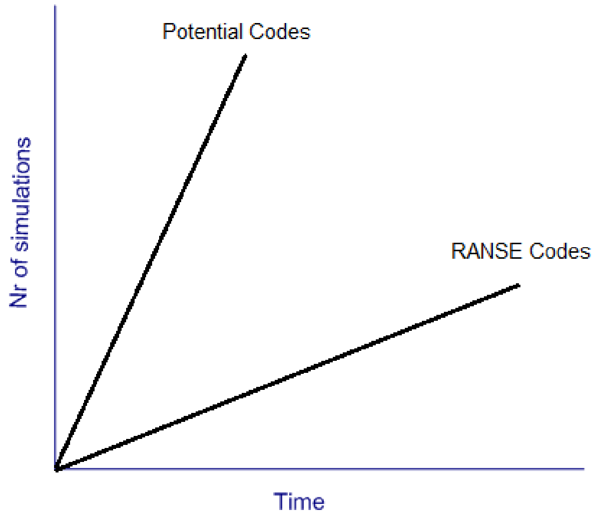


Figure 10: Simulations of potential codes versus RANSE codes

How shall one deal with time and accuracy of CFD? How can you make use of time in the best way? A way of dealing with the trade-off is to start out with a simple model and identify performances with global variables and then to make the model more and more complex and thus more accurate. A stepwise approach could be:

1. *Identify rough and robust variation of the design*: Make a simple model with potential codes and a simple CAD model of the hull. This type of model can generate thousands of versions in an optimization process. It is to investigate rough variations and to get insight and knowledge of how the global parameters affect the performance. In my research example I will create models in this area of trade-off between time and accuracy.
2. *Identify local variation with the basis hull from 1*: Make a full parametric hull shape that identifies local and global changes, use potential codes for optimization of parts of the hull where viscous effects are neglectable and use RANSE codes where turbulence is present and

viscous effects are important. Optimization with RANSE codes should be made with a few iterations and big steps to identify accurate variations performance without using too much time.

3. *Identify accurate performance result*: Use the most accurate method to find the performance with preferably only one model of the hull.

There may be that there is no need to start from point 1. The global geometric form may have been decided for or that the knowledge of the performance with respect to geometric change is already known. Then there is no need to start from point 1, but start from point 2 with a more local geometric search.

Another way of dealing with the time issue is to develop a response surface for a standard case to be used later as a *numerical* hull series (Couser, Harries, & Tillig, 2011). Start out with creating different points by systematically changing the parameters and then to interpolate between them. To use a hull series is an old way of thinking of design exploration. The old hull series were made from model tests and not from numerical computer simulations, but the theory and usage is the same; it allows quick exploration of the design space and is useful in early design. In creating such a response surfaces the designer use a big amount of time. The theory is that the time spent should weigh up against the time saved later in future design processes. A problem with the numerical hull series or hull response surfaces is that there is an error and uncertainty problem in the calculations. Response surfaces should be validated in some kind before they are used, but because they are preferably used in early design exploration accuracy is not the main goal.

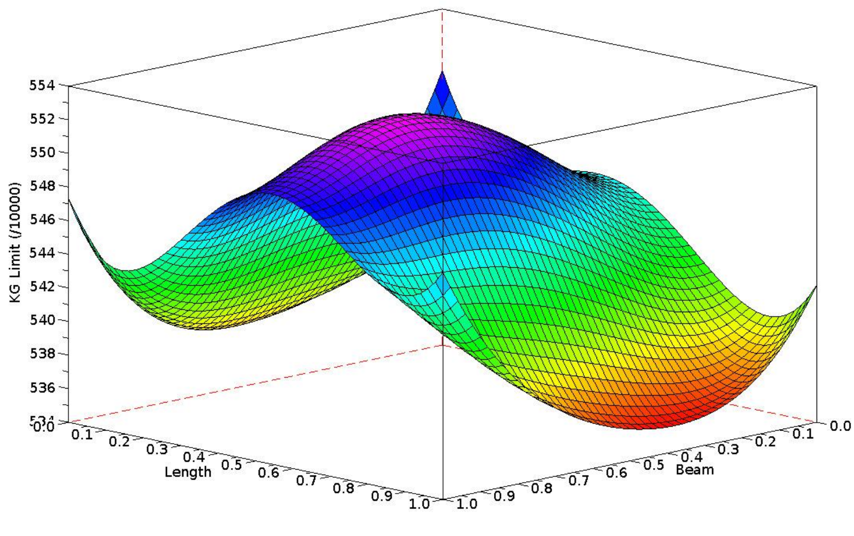


Figure 11: Response surface of KG Limit versus length and beam. Objective: KG limit. Free variables: Length and beam. (Couser, Harries, & Tillig, 2011)

Figure 12 shows the process of developing different design response surfaces. A set of different standard ship shapes is developed in a parametric approach, i.e. different stern shapes because of different propulsion, bow shapes and mid shapes. Put the standard shapes together and run it through a CFD process that changes the global variables. The objective of the response surface could be resistance but also stability limits as seen in figure 11. This is a good way of using standard shapes in early design and to gain knowledge of variation. It could also be used in learning processes to give knowledge of how global parameters and shapes affect the performances. The visualization gives an extra dimension in a learning process and thus also in a value rate of CFD described further in Chapter 9

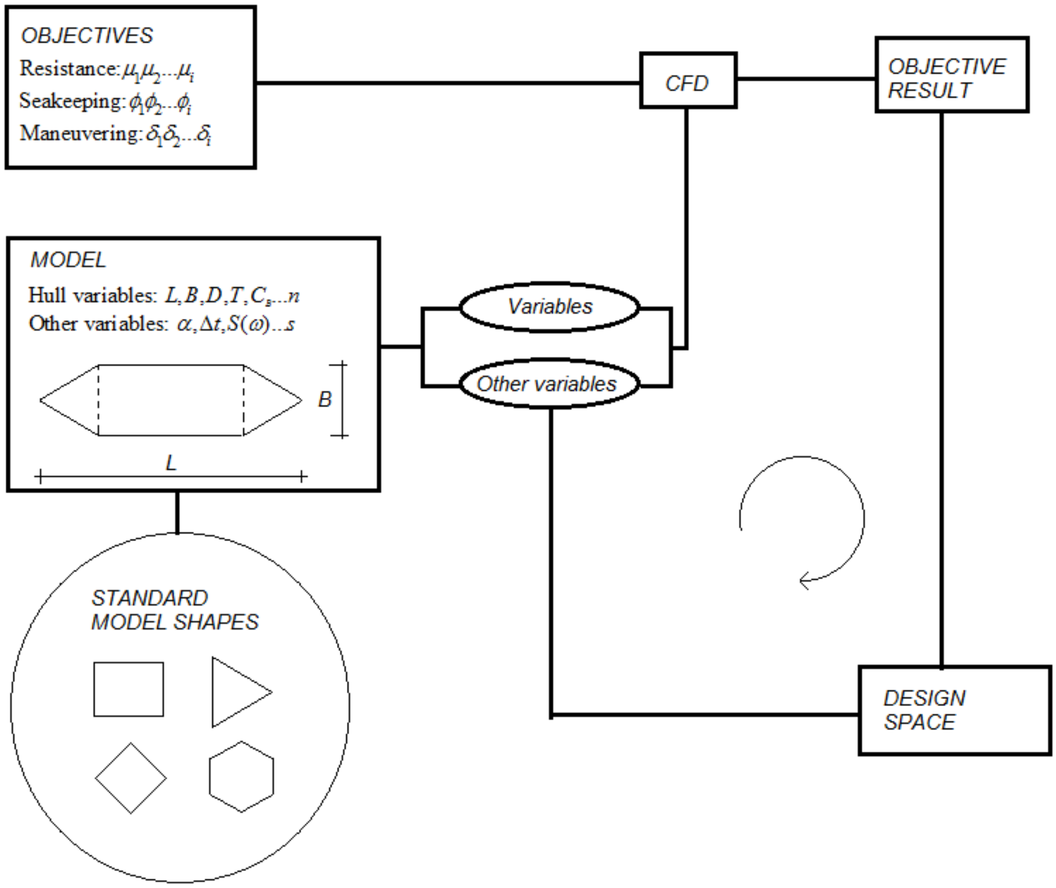


Figure 12: How the design space or response surface(s) is developed by using CFD and simple standard model shapes

6 UNCERTAINTY AND ERROR IN CFD

Is the CFD result correct? Or how much can I trust the results? These are questions that remain after a run with a CFD code. If a designer is going to use the result in a design process the designer needs to have a reliability check. In engineering the validity of a computer design is often checked by model tests to see if the computer design result differs from the model testing. CFD is still a computer program tool that contains more errors and uncertainty than a model test, but in developing new design it seems that model testing is outdated. Model tests will be to verify design and not to develop new design. That is too expensive both in cost and time.

Uncertainty is defined as (Slater, 2011):

> *A potential deficiency in any phase or activity of the modeling process that is due to the lack of knowledge*

The definition indicates that there may or may not exist a deficiency. In modeling it is sometimes hard to see if there exists a deficiency in the design. The designer does not have the needed knowledge in the physical process that is needed in building the model and lack of this knowledge leads to uncertainty.

Error is defined as (Slater, 2011):

> *A recognizable deficiency in any phase or activity of modeling and simulation that is not due to lack of knowledge.*

Different from uncertainty is that error is a recognizable deficiency upon examination which has nothing to do with the lack of knowledge of the designer, but has to do with mistakes or deficiencies that are there because of simplification or approximations.

The definition of error here is different from that of an experimental physics, which is 'the difference between the measure value and the exact value'. In CFD the exact value is typically not known, and errors in CFD have to be treated with uncertainty according to what is known.

Error can be further broken down in acknowledged and unacknowledged error. Acknowledged error is error that is identified and put through a procedure like elimination or listing. Unacknowledged error is error which is not treated within a procedure.

6.1 Acknowledged error

1. *Physical approximation error: Physical modeling error and geometric model error*
2. *Computer round-off error*
3. *Iterative convergence error*
4. *Discretization error: Spatial discretization error and temporal discretization error*

1. A physical approximation error is an error that is due to the uncertainty in the formulation of the model and simplifications done in the modeling. This is not a part of the discretization of the model, this error deals with the continuum model only. It's about the choice of the governing equations which are solved and the properties of the fluid and solid. These errors occur because of the uncertainty in the physical models and of lack of knowledge in the phenomenon. Simplifications are then introduced because experimental confirmation is not possible at the time or cost. Physical modeling errors are examined by performing validation studies that focus on certain models (i.e. inviscid flow, turbulent boundary layers, real-gas flows, etc.)
2. Computer round off errors develop within how the computer stores floating point numbers. It is not a significant error because 64-bits are now a standard way to store numbers and it is neglectable compared to the other errors.
3. Iterative convergence error exists because there must be a stopping point in the end of an iterative method used in a simulation. The error scales to the variation in the solution at the completion of the simulations
4. Discretization errors are errors which represent the governing flow equations and physical models as algebraic representations in time; finite difference, finite volume, finite element. Discretization errors are also called numerical errors. As the grid points or number of volume goes to infinite the discretization error will go to zero. This convergence is also present in the time stepping errors. The discretization error is of most concern for the user during an application. The error all depends on the quality of the grid that is developed by the user or by an automatic grid generation. In the beginning of a simulation the quality of the grid and accuracy is difficult to indicate. There are many things that affect the quality of a grid like; resolution, density, aspect ratio, stretching, orthogonally, grid singularities and zonal boundary interfaces. To deal with discretization error different runs with different degrees of complexity are investigated in order to see if the solution converges.

6.2 Unacknowledged error

1. *Computer programming error*
2. *Usage error*

1. Programming errors are 'bugs' within the writing code of the program. Validation studies are made to get rid of such errors and are the responsibility of programmers, but there will always exist some probability of programming mistakes or errors.
2. Usage error is an error that the designers are responsible for. The designer may not have the right knowledge and user interface which are needed. User error may come up as a modeling or discretization error. If the user input is not properly accurate the results will also be inaccurate. The wrong conclusions may also be drawn from the results. The potential of user errors increases with increasing options available in the CFD code. It is minimized through smart programming and interfacing the codes, proper training and accumulation of experience. A part of user errors is intentional errors in order to make the model easier. This may be proper in a conceptual study of the design. To make a model easier will often give a higher discretization and physical approximation error.

7 DESIGN METHODOLOGY – TWO METHODS

‘Conceptual design is about sequentially identifying the problem and analyzing the relevant information and consequently formulating relationships between design parameters and functional attributes, and acquiring a comprehensive discourse about the achieved solution, the principles of the model’s the functionalities, and the searching process’ (Brinati & de Conti, 2007). There are many ways of approaching conceptual design. I will try two divide them into two approaches; point-based design and set-based design. How ship design companies approach a new ship design or how they design new ships is as diverse as there are different ship design companies or ship yards. To narrow it down to two methods is to simplify it, and to get a general overview of design methods. The reason why I implement this overview is that CFD will be used differently according to what method that is used, and to investigate the design process.

There are many interpretations of what conceptual design contains. My interpretation blends somewhat into the definition of preliminary design. In this research conceptual design is defined as the first stages of the design, it is here that all the main parameters are decided, like engine type, propulsion, systems onboard, hull shape, main dimensions etc. Figure 13 shows examples of what I consider as conceptual design. The ship hulls shown are in the conceptual stage and are a result of imagination, innovation and experience but have not been proved and tested. They differ from normal hull of a ship and may be an improved hull shape for the ships operational profiles.

Figure 13: Ulstein design of a container vessel to the left (Ulstein Group) and STX OSV design of a PSV vessel to the right (STX OSV)

Conceptual design may imply the designing standard ships or new innovative ships. Designing standard ships will not demand much time in the concept stage, the ships will nearly be finished already as a concept. In such a process no big innovative solutions are needed, and it is not the customers demand to get such a ship. However there has been a tendency to have an assortment of different ships that the customer may pick from, like a commercial off-the-shelf (COTS) design product. Here an innovative solution is important when selling the product. In this thesis I will focus more on how to make innovative solutions, like new COTS design or a new ship design, and not how to standardize production design.

'Design in engineering is a decision-making process that leads from a set of given product requirements to a product definition with all salient features for design assessment and production. Design is a synthesis process. The number of free design variables will be greater than the number of equality constraints. Thus the solution is not uniquely determined by the set of requirements. There will therefore exist a solution space of many feasible solutions unless conflicting constraints prevent a solution. The designer will then by some synthetic judgment pick the most favored solution, either by direct intervention or by declaring a measure of merit to define a yardstick for the best design. This is how design differs from pure analysis' (Nowacki, 2010). This is how Nowacki describes the design process and its complexity.

Figure 14 shows what areas that usually are investigated in ship design. They are all linked to each other by the global parameters and dimensions. It is because they are so much linked to each other that it is difficult to start out from one place and end with one as Nowacki describes it, and therefore different methods have been developed to create a fast reliable way to get to a solution with the respective constraints that are investigated. It is with resistance and propulsion, maneuverability and seakeeping that CFD will play its role. Structure, stability, cost and volume of the ship are also coupled up with what kind of hydrodynamic performance the ship will take.

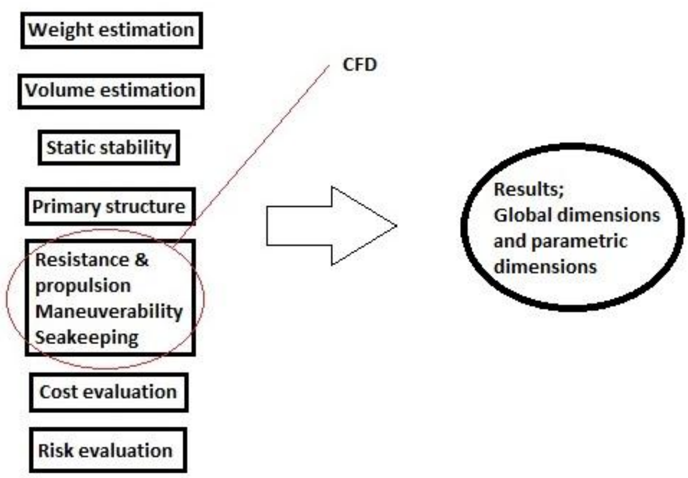


Figure 14: Areas that usually are investigated in a conceptual design stage

7.1 Time and knowledge

There are three factors that identify a general design process; the evolution of a product's cost, management's ability to affect these costs, and the evolution of the designers' knowledge about a design problem. The factors are important to understand when in the process CFD can be used most efficiently.

1. The first factor is production cost. When a new product is developed, the designers will make decisions affecting the life time cost of the product: How expensive will it be to manufacture? How much of the price of the product will be earned as a profit? How much will it cost to maintain and operate the costs? The difficulty with production costs is that the largest impact is done in conceptual design, which is almost sixty percent (Anderson, 1997), with the least data of knowledge of the product. The decisions made very early will have a long-lasting consequence on the total cost of the system, while late decisions will have less impact.

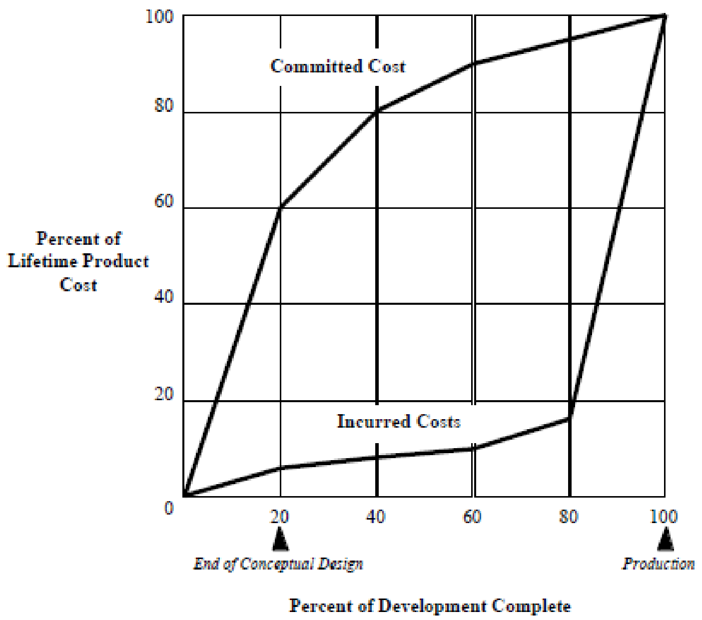


Figure 15: Production cost that is in the design process (Bernstein, 1998)

2. As illustrated in figure 16 the second factor is influence of the management to the product. It is at its largest in the very beginning of a design process. In addition to this, the cost of making changes in the design variables rises exponentially in the development cycle. The further one goes into the design process the harder it will be to add additional needs. The reason is that every decision made by engineers constrains the options of available future

decisions. So the later you are in a design process most of the life-cycle cost has already been decided for and the ability to change this cost declines rapidly.

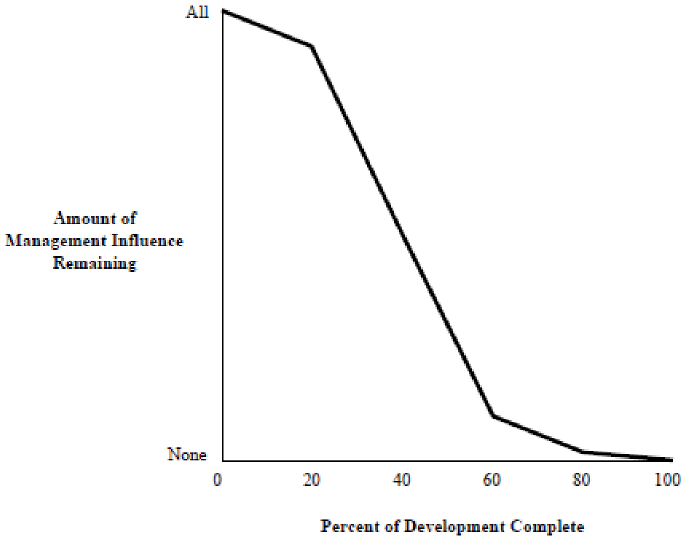


Figure 16: Management influence on the design during the design process (Bernstein, 1998)

3. A third factor is the lack of knowledge that the engineers and managers have in virtually every aspect of the product in its conceptual design stage. As the design process goes forward more information is gathered of the customer's needs and constraints. More knowledge will give better trade-offs in decisions and solutions to the problem, but as explained the decisions are made early with little knowledge. This is some of the reason why ship design has relied on experienced designers to not make the wrong decision early in a project.

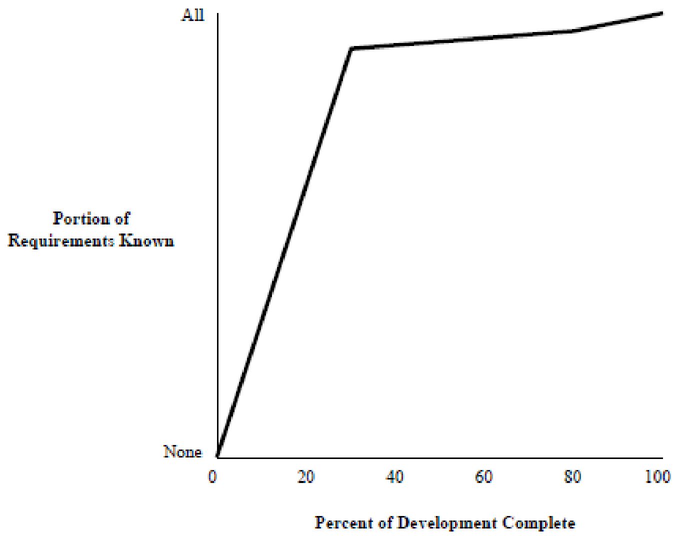


Figure 17: Knowledge of the requirements and the design during its development (Bernstein, 1998)

In summary time and knowledge are the key factors. There is not the time to gather all the knowledge in a conceptual design stage and one has to rely on experience to make good decisions. Or in some cases just rely on what has been done in the past and trust that it is good enough. Knowledge is therefore crucial, but getting the knowledge is difficult. There is much uncertainty in the variables chosen because of inaccurate methods used in this stage of the design. Even if it is in the conceptual design the biggest impact on the life-cycle cost is influenced it is here that the amount of time spent of the whole project is shortest. Because time is so scarce in conceptual design, easy methods have been used to find the performances of the design like regression models based on model tank testing. These models are now considered as outdated because they are all based on old ship hull shapes (Bertram V. , 2002).

New methods like CFD can help to get better results in some parts of the design process, but there is still a time issue as described in Chapter 5. Also optimization processes have been favored in many areas in conceptual design to identify good solutions fast and letting a computer search for solutions. It is used as a decision support tool in many cases. It is clear that optimization with CFD can be a good support tool to make better decisions in conceptual design, because it gives more knowledge of the problem. In Chapter 11 I will use three examples of how optimization can be treated in a concept stage to gain knowledge of the problem.

Decision in ship design is much linked up with the trade-offs which have to be done. “A trade-off between two opposing things is a compromise or balance between them” (Clue, 2010). How much balance that will be given to each opposing thing is the designer’s decision. Complex engineering is filled with compromises or trade-offs that have to be decided in the design process. When and how decision is made, is how a designer can make things wrong or right when making a product. To make right decisions you would need to use the knowledge and experience that you have at hand. To have reliable tools to increase the knowledge will give better decisions.

7.2 Point based design

The best in design is to have as many concepts as possible to consider the alternatives. But having many different alterative designs takes time; it is better to find a solution fast that can satisfy the requirements. This is the strategy of an iterative process or point based design. Point based design is essentially Evans’ (Evans, 1959) design spiral or other spiral design methods where a single design or point based design is found when the design satisfies all the constraints in each step of the spiral.

This model emphasizes that there are interactions between each step and it must be considered in a sequence or in a spiral. It is an iterative process, and the theory is that the design will be better in each pass around in the spiral. The result is a design that can be developed further or used as the start point for various trade-off studies.

Some disadvantages of the point based design are that it will not create a global optimum in terms of the ship design measure of merit, such as the Required Freight Rate (RFR). The number of iterations will also have a tendency to be limited, because of time and budget, so the design will not be adequately finished and converged.

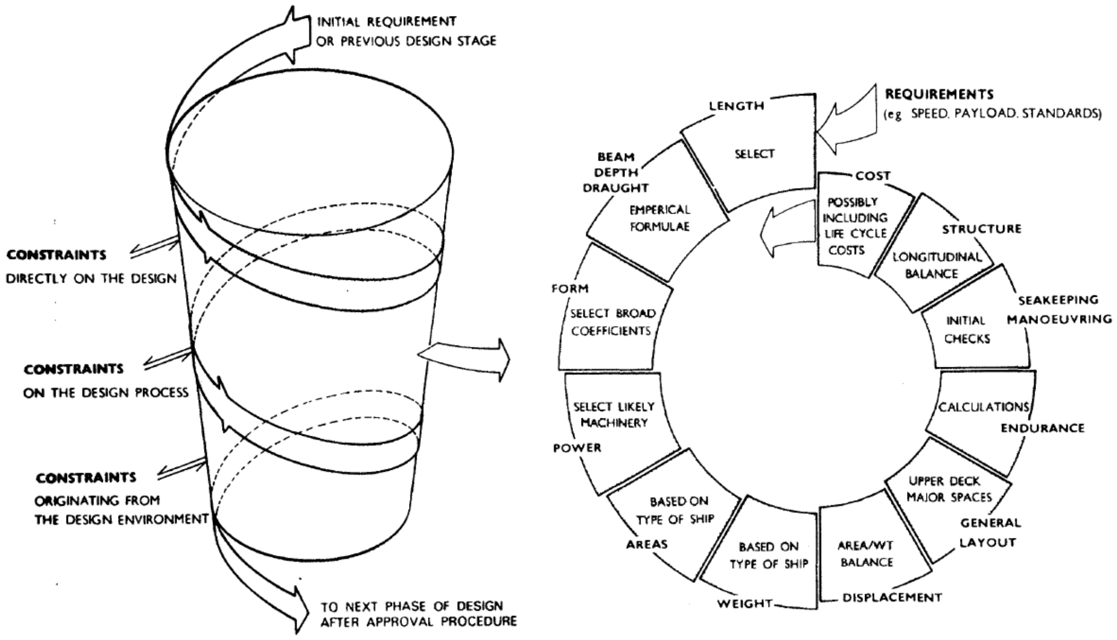


Figure 18: Point based spiral iterative design (Andrews D. , 1997)

In general a point-based strategy consists of five basic steps (Singer, Doerry, & Buckley, 2008):

1. First, the problem is defined
2. Engineers generate a large number of alternative design concepts, usually through individual or group brainstorming
3. Engineers conduct preliminary analyses on the alternatives, leading to the selection of a single concept for further development
4. The selected concept is further analyzed and modified until all of the product’s goals and requirements are met.

5. If the selected concept fails to meet the stated goals, the process begins again, either from 1 or 2, until a solution is found.

7.2.1 Communication

In complex design, as a ship design, the products require a wide and diverse set of skills of the designers, which tends to be beyond the grasp of one individual. The design process is therefore a set of groups that past knowledge over to one another. Transformation of knowledge is therefore an important aspect to look at in complex design so that no knowledge is mistaken or lost in the design process.

In point based design there is a tendency to have an ‘over the wall’ knowledge transformation. What is meant by this is that the product development is done in stages at a time where one has to build the design in stages in a sequence. The knowledge in each sequence is just handed over to the next sequence, or design team, which has had no influence on the requirements and parameters up till now. The parameters and requirements are established already, but can be changed so that they also will satisfy this stage in the design process. And so will the design process continue in a circle till all design sequences are satisfied. Some of the problem is that when one design team is changing the requirements and parameters and that they don’t know exactly how the change will affect the objective of the other stages in the process. An example would be two groups of design teams; one is the structure team and the other is the hydrodynamic team. The hydrodynamic team passes over the optimized hull to the next team which is the structure team. The structure team looks at the hull and decides that there are some difficult shapes that are difficult to make in the production, or that it will be cheaper if the hull is made by changing the hull in more standardized building blocks. How will the change affect the hydrodynamic performance of the ship, what is the trade-offs? A way to deal with this ‘over the wall’ communication is to work more in a concurrent engineering team. This will improve the design process and mitigate the errors due to limited intra-team communication caused by distance. This is a step towards a set-based thinking of design, but is still point-based because they have not erased the iterative way of going towards the final design and the theory of establishing early constraints.

In point based design much effort is used to establish ‘hard’ constraints as early as possible. What is meant by ‘hard’ constraints is that it is a deterministic number i.e. speed is put to 14 [knots] or loading line is 10.3 [m]. There is however constraints that has to be ‘hard’, like port constraints. The theory in point based design in establishing ‘hard’ constraints is that the design team will not use time outside the design space. But this leads to some contradiction in point based iterative design as explained under.

Bernstein did an extensive analysis of set-based design and how it could be integrated in the aerospace industry. In his thesis he mentions two paradoxes in point-based design (Bernstein, 1998):

The first paradox; system design methods emphasize establishing requirements early, but iterative methods imply that they will change in the course of the iteration. It means that though point-based design tries to establish the requirements early to save time, the iterative approach will force the requirements to change in the process and actually more time could be made searching over and over again in the process.

Second paradox; do it right the first time, or establish early requirements and constraints, mentally actually to decrease the cost effectiveness of the design process by degrading the amount of information which the process produces, because success design is maximizing the information by an adequate failure rate. This is on the basis that the purpose of design process is to generate information cost efficiently; to gain as much knowledge in one test.

7.2.2 CFD in point based design

CFD in point based design will be used to find a point solution of the performances used by the set of requirements that are put up. The global parameters are usually decided before the hydrodynamic performance part of the design process take place. There are some 'rules of thumb' about hydrodynamic performances that are used in deciding the global parameters, so that some knowledge is based on the decisions that are made. The change in geometric shape with respect to the hydrodynamic performance will be done by local geometric shape, i.e. bulb change. CFD is used to verify and find the performance, and not to base the decisions on the result that CFD provides. But if the result of the performances is not satisfactory a new run in the iterative process is needed. CFD in point based design is often coupled up to an optimization algorithm that search for the best point solution in the design space. But this is usually a local geometrical change search that is used.

7.3 Set based design

Set based design is a concurrent design process that differs from iterative point-based design. It is a different way of approaching the design process. This way of designing complex systems was a result of a study (Sobek, 1997) of how Toyota did their production and designing of automobiles. The

Toyota process of designing cars was investigated because they managed to produce world-class automobiles in significantly shorter time than required by other automobile manufactures.

Set based design could be summarized as a method for engineers and product developers to get to the design solution(s) by reasoning, developing and communicating about sets of solutions in parallel and relatively independently. What it means is that a set of designs or groups of design alternatives is established early and narrowed down by gradually eliminating the alternatives until one option remains. It could also be several options i.e. two prototypes that meet the demands as a result of this process. The theory is that this method will provide a global optimized solution to the design problem. The trade-offs and decisions are delayed and that more knowledge is provided before the decision is made.

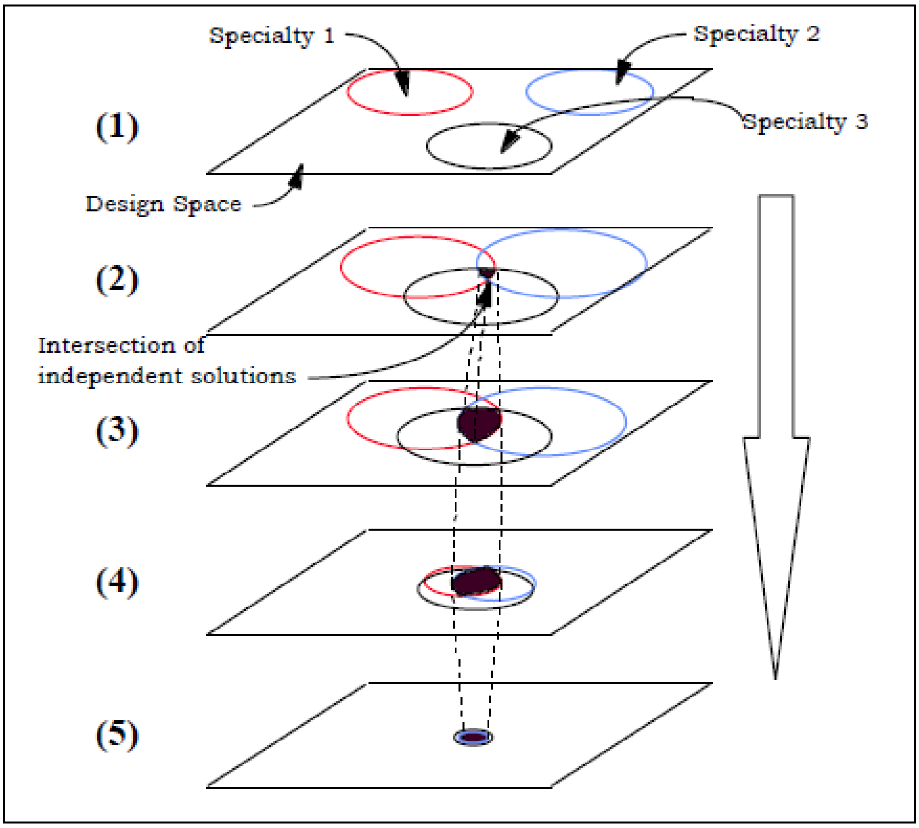


Figure 19: Set based design approach towards the final solution (Bernstein, 1998)

In general a set based strategy consists of four steps (Singer, Doerry, & Buckley, 2008):

1. A broad set of design parameters is defined to allow concurrent design to begin
2. These sets are kept open longer than usually to more fully define trade-off information

3. The sets are gradually narrowed until a more globally optimum solution is revealed and refined
4. As the sets narrow, the level of detail (or design fidelity) increases.

7.3.1 Communication

Communication and collaboration is done in another way in set-based design. The solutions or sets of solutions are created by more individual groups of experts. The different groups should develop a solution of the problem by their own perspectives. The interaction is done after each group has reached their solution and the process of trade-off and narrowing down starts. An important aspect of this way of dealing with design is that the design variables must not have a single value but a range of values with max and min value, i.e. length of the ship and should be between 90 [m] and 110 [m]. This also yields for the constraints of the system; the constraints are 'soft'.

In contradiction to point-based design, set based designs initial development typically seek to define regions of the design space instead of several solutions.

An example from the automobile industry is the competition for volume under the dashboard that might arise between an audio system and a heating system. Instead of specifying in advanced the envelope each of the systems must fit in, the designers can design a range of options so that the design teams can see the differences in cost and performance between these two competing items.

Some of the problems that can occur in set-based design are that it can take a longer time to find a possible solution than point based design. Another problem with set based design is that not always will the independently solutions fit into each other. The solutions may be so different that only one of the solutions is developed further, time and money is then spent on a design that will be scraped. It is therefore important for the management of a set-based design team to ensure that a narrowing process does in fact occur.

7.3.2 CFD in set based design

CFD in set based design will be to develop design spaces and regions where the solutions can be found as explained in Chapter 5 and response surfaces. It is used to quantify the performances in a very early stage of the design. It is used to gain knowledge so that better decisions can be done.

The process of narrowing the design sets, the design also gets more detailed. One must note that the design in early stages, with little detail, should be tested with simple and quick methods. It should not be a complex test of design but just enough to expose the problems. The models of the design should be easy in the beginning and then get more complex as the design develops. CFD models should therefore be easy and robust in the beginning of a set-based design approach and evolve to get more complex as the design develops.

8 OPTIMIZATION PROCESS IN DESIGN

Optimization and CFD have gained much attention the last decade. It is natural to think that CFD coupled up to a design hydrodynamic performance optimization process is a nice way in letting the computer search for the optimal result in a hull design process. The improvement in design can be lesser resistance. In many researches there have been produced more efficient hull shapes in an optimized CAD – CFD framework with 2 – 6 % less resistance (FRIENDSHIP SYSTEMS, 2009). This is more a detailed design where small variations in the bulbous area are changed.

Coupled with design is the design optimization, or the selection of the best solution out of many feasible ones. Optimization can be used to help the designer to reach a solution, but because the ship design is such a complex problem the best solution may not be found but a very good one may be found. A systematic approach of how to look at ship design may be to divide it into a variety of complex subsystems and their components. A subsystem could be power generation, cargo handling and storage, accommodation of crew and passengers, and ship navigation. Each of the subsystem forms a complex nonlinear optimization problem for the design variables, with a variety of constraints and objective functions.

Inherent to the design optimization is the conflicting requirements put up by the various ship design stake holders; Ship-owners, operators, ship builders, classification society, regulators, insurers, cargo owners, forwarders, port operators etc. Overall requirements for all stake holders would be economy and safety. But how important each of the requirements is will change for each stake holder.

The initial set of ship design requirements is the outcome of compromise in discussions between highly experienced decision makers, mainly of the ship design and the shipbuilding side, and end users who will try to articulate their desires and trade-offs which they are willing to allow. Optimization can be used to shorten the time to find the solution(s), but it will also give the assurance that improvements no longer are feasible. It can give the designer an insight in how the constraints are governing the solution, and the designer may 'soften' the constraints that are given.

Some basic elements in optimization:

1. Optimization objective: This refers to a list of mathematically defined performances or efficiency indicators that may be eventually reduced to an economic criterion, namely the profit of initial investment, or safety criterion. There are several objective function goals and

one of them would be the hydrodynamic performance of the ship. A way of couple hydrodynamic performance to the overall requirement could be the installed power for economy and stability for safety. The hydrodynamic performance can again be broken down in basic elements and objectives.

2. Constraints: This refers to mathematical inequalities or equalities. Normally they are put up by regulatory frameworks pertaining to safety (SOLAS and MARPOL) or physical impossibilities (material strength). These defined objectives may be extended by a second set characterized by uncertainty with respect to their actual values and being determined by the market conditions (supply and demand), cost of major materials (steel price and fuel price), anticipated financial conditions (interest rates) and other case-specific constraints. The uncertainty of the latter set of objectives may be assessed on the basis of probabilistic assessment models as mentioned in Chapter 10 and robust ship design.

3. Design parameter: This refers to a list of parameters characterizing the design. For a ship this includes the main dimensions (length, width, draught and depth). It may be extended to the hull form, the arrangement of spaces and outfitting, structural elements, percentage factors, coefficients and networking elements.

4. Input data: This includes first the traditional owner's requirements. For a merchant ship this may include cargo capacity (deadweight and payload), service speed, range, etc. In a more global optimization model it may be complemented by a variety of further sets of data affecting the ship design and its economic life, like financial data, market conditions, and cost of materials. It may also include a more general type of knowledge like type of ship and propulsion or drawings of general arrangement. This has to be translated for inclusion in a computer-aided optimization procedure and parameterized. As with the constraints these input data may be of an uncertain character, like the speed of the ship and a probabilistic approach could be used.

5. Output: This is the values of the entire set of design parameters for which the specific optimization objective are in an extreme value. Trade-offs in a multi objective is often done by rating the objectives and implementing them into the model or by creating a Pareto front (Erfani & Utyuzhnikov, 2010).

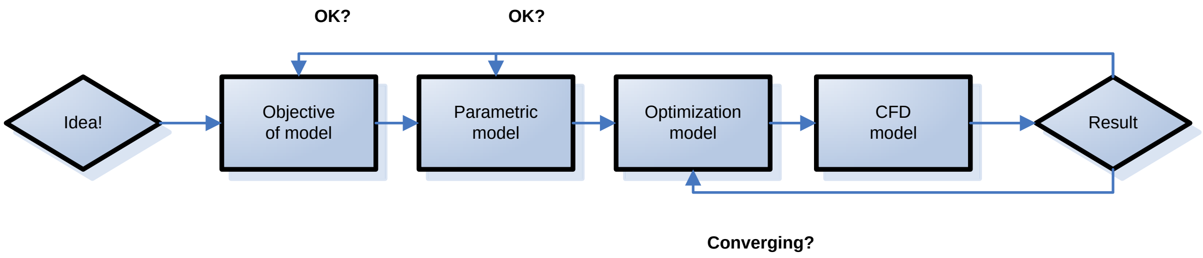


Figure 20: Simulation driven optimization design

An optimization process starts with ideas of how to model the design as seen in the figure 20. An idea could be robust ship design with different draught and speed. The next step is to find an appropriate objective(s) of the model; single or multi-objective. The geometric shape is then generated in a CAD computer program. Parametric modeling is done so that the optimization process is done with lesser variables. The optimization model is chosen most out from if it is a multi-objective problem or a single objective. There is a diversity of optimization search algorithms and they act quite differently from each other so care should be made when choosing the optimization algorithm. The CFD model or method is chosen from time perspective and robustness compared to the geometrical change in the CAD model. In every step of the process the result must be checked if it is in order of the objective and the parametric model, and if it in fact converges to a solution.

As described optimization can be defined in some basic elements, but how these elements are treated could be different. I would say that there would be a difference in how to treat the basic elements when speaking about point-based method versus a set-based. How the optimization is done lies much in where and when the trade-offs are done.

8.1.1 Optimization in point-based design

In point-based design optimization is done with hard constraints. That means that when looking for solutions there will not be a solution that lies outside the constraint boundary, i.e. a ship that has a speed constraint or a space constrain. The optimization process tries to converge the solution as quick as possible to the final point design in the solution space. The trade-offs in point based design is often done along the design process, while in set based the trade-offs are done in the end of the design process. A more point based approach in optimization would be to quantify the trade-off by a factor (see Section 10.2.1)

8.1.2 Optimization in set-based design

Optimization in set-based design is more an integrated part of the design process than a search process in a computer model. The focus of use of computer models is to search for design spaces and the trade-offs is done by human decisions and the problem gets narrowed down. Attempting to integrate the solutions by the different groups will then lead to optimization of the system. The different groups that have created their solution to the problem will all have their expertise in one of the objectives of the design although they must take into account the other objectives of the design the group will try to optimize it with their expertise, i.e. the group that is experts on hydrodynamic performances will try to optimize the ship for lower resistance, motions and efficient propulsion while the steel construction group will try to minimize the weight and complexity of the construction.

9 CAD AND CFD

Computer Aided Design (CAD) is a method for the designer to visualize the forms and shapes. It is simple to use and programs that have been developed are user-friendly. The designer can change variables and make different shapes. Because of this tool it has made innovation more possible in engineering, but also it has made it easier in implementing the different systems and space arrangements.

The benefits of CAD for consumers and customers are the rising quantity and quality of new products at lower cost along the diversity of product variants, and reduced time to the market.
It is important to establish the function of the product developed in the CAD system and look at the quality. The quality of the product could be in ship design reduction of fuel consumption, less noise or less weight (Harries & Abt, 2008).

Visualization will enable the following:

- Have a physical representation of the system being optimized
- Improved understanding of the trade-offs that are made in the optimization design process
- Assist in the early stages of optimization problem formulation and implementation
- Easily to determine the path taken by the optimizer through the design space and interrogate individual design options
- Verify assumptions regarding the physical configuration of the system. In many cases, the designer may not be aware that these assumptions were made
- Simplify the downstream management of the assembly and manufacturing tasks by identifying problems upfront. For example, the optimizer could decrease the thickness of a part to unreasonable value that would prevent the part from being manufactured. Monitoring the design evolution would help identify this more quickly.

CAD is a well-established tool in the engineering design environment, but CFD has not been implemented that much in early design, because it consumes a lot of time and is not as user-friendly. But even though it seems that the time decreases for CFD simulations it is a complicated tool that would need expertise.

To make a good CAD design that interacts well with CFD, the designer needs to have knowledge of modeling, fluid dynamics and programming. The challenge for a designer is to have a well functioned coupling between CAD and CFD. It still exists many problems in the diversity of grid generation and the inputs from the CAD model. This is a programming error that the software company must address. But still the designer must have knowledge in programming so that he or she does not get too much error and problems with the software. For a designer and not a programmer the main challenge is to narrow down the free variables without narrowing the design space. The designer needs to make an intelligent *parametric* model of the design

Parameter definition (Definitions online):

- *Parametric quantity (a constant in the equation of a curve that can be varied to yield a family of similar curves)*
- *Parameter ((computer science) a reference or value that is passed to a function, procedure, subroutine, command, or program)*

As described in the definition a parameter is a variable that affects many other variables. It is a way to narrow down variables to make it easier to make geometric change.

There are different stages of parameterization (Harries & Abt, 2008):

1. *Conventional variation:* The shape is defined by independent variables that do not bear any problem-specific information. A B-spline surface described by points is an example of conventional geometric modeling.
2. *Partially parametric variation*s: the shape is defined by conventional description, but changes applied to the shape are given by means of parameters. These parameters are associated with problem-specific properties. An example is the Lackenby shift function (Lackenby, 1950).
3. *Fully parametric variations:* The geometric model is entirely described by high-level parameters that reflect the characteristic of the product. An example is a cylinder which is represented by two parameters height and radius.

The efficiency of the three different methods of parametric modeling is described in figure 24. A fully parametric model requires expertise of the designer because the model will very quickly lose flexibility when narrowing down the variables from conventional modeling. The reason to do such a high level of parametric modeling is described by the cost per new high-quality variant graph. The cost of change is much less, and this explains why a fully parametric model is favored in optimization.

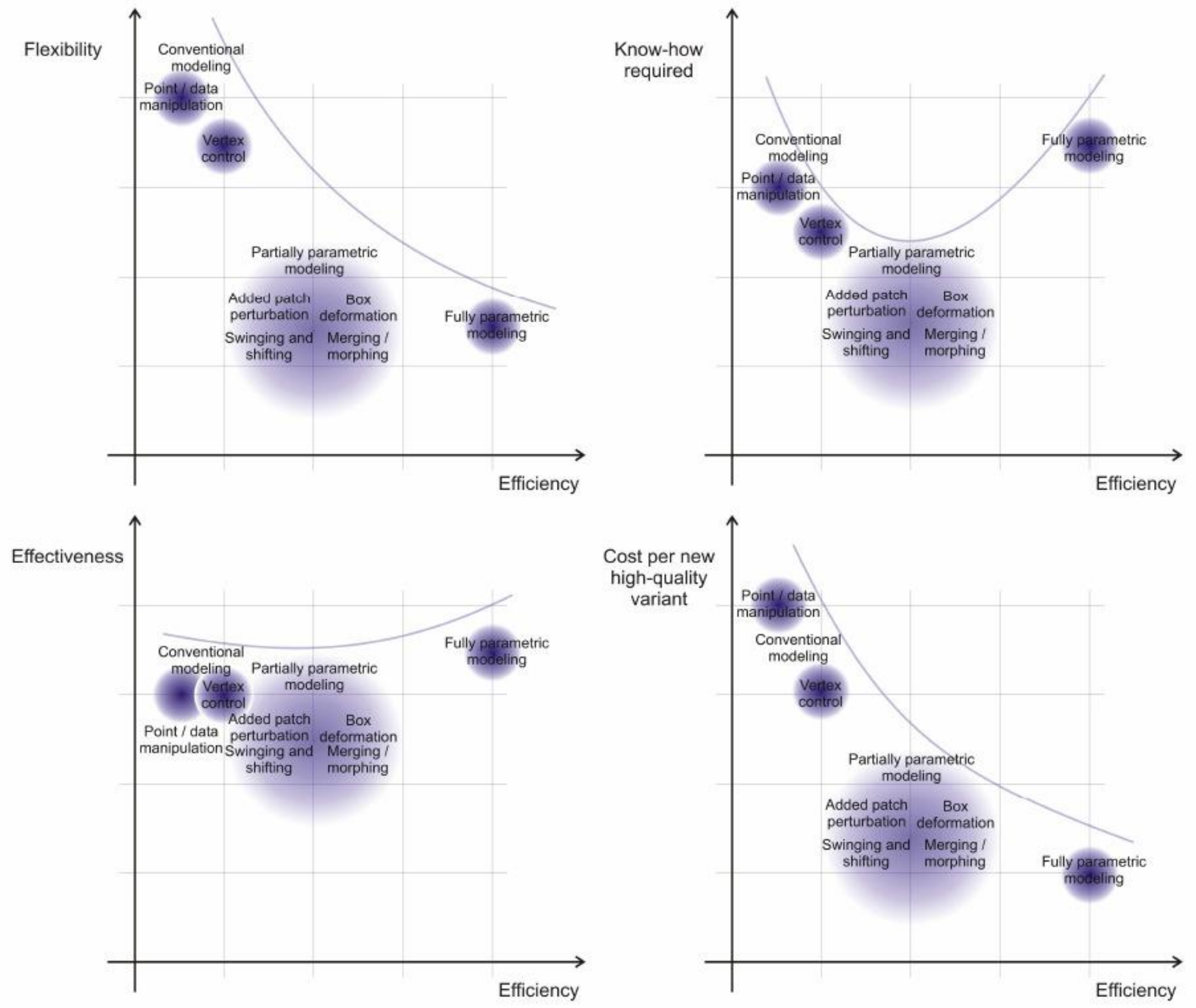


Figure 21: Assessment of the different geometric modeling techniques (Harries, 2008)

Parameterization is very useful for optimization purposes where you would want as few variables as possible to make the process simple. But there are many difficulties involved in parameterization of a design. Much experience is needed and you could lose much information by narrowing down the variables too much. When seeking optimization of a design, parameterization is a designer's most important task.

9.1 Value rate of modeling and simulation

'Purpose of design process is to generate information cost-efficiency' (Reinertsen, 1997). Cost efficiency would intend how long time one design process takes or how long each test would take and how much information that is gathered in that amount of time. When conducting tests or design processes, more information is contained in the result of failure then with success. A design process is than a success when ‘an adequate failure rate to generate sufficient information is gathered’ (Reinertsen, 1997). So how to gather the most information in as short time as possible will be a function of what tools that are available and what kind of design method that are used. Tools like CFD can generate information, but there are such a wide user-profile and complexity so that the tool must be divided into different areas of use. When and where to apply the different CFD methods in a design process will depend on how cost efficient the method is. However there is no ‘one-answer’ to this, it will be different from design to design and what the intention of the design process is.

There are degrees of complexity of modeling in design. In ship design research there have been a tendency for more complex models and optimization techniques in hull optimization, but the question remains if the time put down in the complexity of the models pays off.

Venkataraman and Haftka identify three types of complexity in modeling (Venkataraman & Haftka, 2002):

- Model complexity, inherent to the size of the problem, and related to the number of design variables and constraints
- Analysis complexity, related to the level of fidelity of the models used. Fidelity ranges from low-fidelity, empirical models to medium-fidelity models based in a simplified approach, such as beam structural models or panel methods for fluid dynamics.
- Optimization complexity, related to the type of optimization: linear or nonlinear, deterministic or probabilistic.

This is the overall model complexity of a design problem. CFD complexity is much related to what method that is used. Figure 25 shows that the complexity increases when going from Potential Codes to RANSE Codes. RANSE Codes has also a much wider area of complexity with its turbulence models and grid generation. It is the complexity of the model and its features that tells something about the value rate of the process. Increasing complexity will increase the time, and thus affects the value of the process.

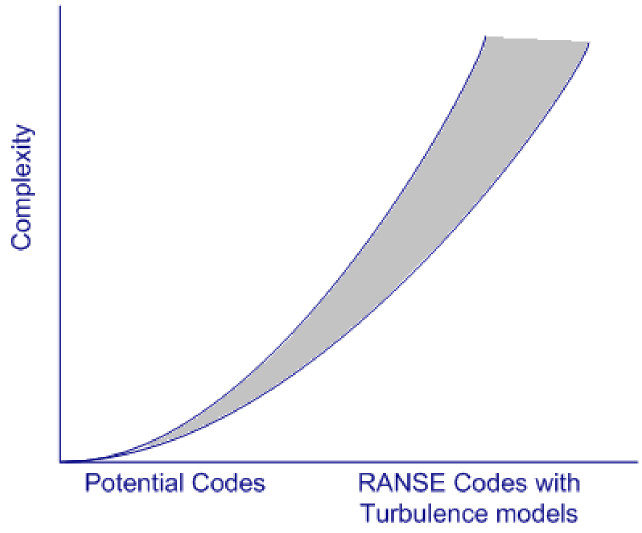


Figure 22: Complexity and the two CFD methods

The value of a model leads to a trade-off between the need for information and the time needed to do a model run. Increasing complexity will increase the time of a model run. A way of rating value in modeling Rubbert in his chapter; *On the pursuit of value for CFD,* evaluates the quality of a design solution by the amount learnt over a certain period of time (Rubbert, 1999):

$$(Rate\ of\ \ Learning) = \left(\frac{Learning}{Cycle}\right) \cdot \left(\frac{Cycle}{Time}\right) \qquad (9.1)$$

The first term represents the amount of information that is given in a cycle. The second term is the number of cycles that can be carried out in the specific time available. However, more of importance is the increment of information that the design will give in the amount of time or the added value in each cycle. Rubbert represents the ratio of incremental value to the value of previous information as:

$$\frac{\Delta Value_{\text{mod}}}{Value} = \left(\frac{\dfrac{\Delta Learning}{Cycle}}{\dfrac{Learning}{Cycle}}\right) + \left(\frac{\dfrac{\Delta Cycle}{Time}}{\dfrac{Cycle}{Time}}\right) \qquad (9.2)$$

Both of the terms in the added value represent the complexity of the model. The first term represents the complexity in the sense of how easy the model is to be understood. The second term represents complexity in the sense of how long time it takes to run (the longer run the higher is the complexity). Increasing complexity would therefore indicate increasing time per cycle and a more difficult model with more variables. Accuracy is not taken into account in this value term, but one could couple it to the cycle per time term. In CFD higher accuracy would intend more time spent per cycle.

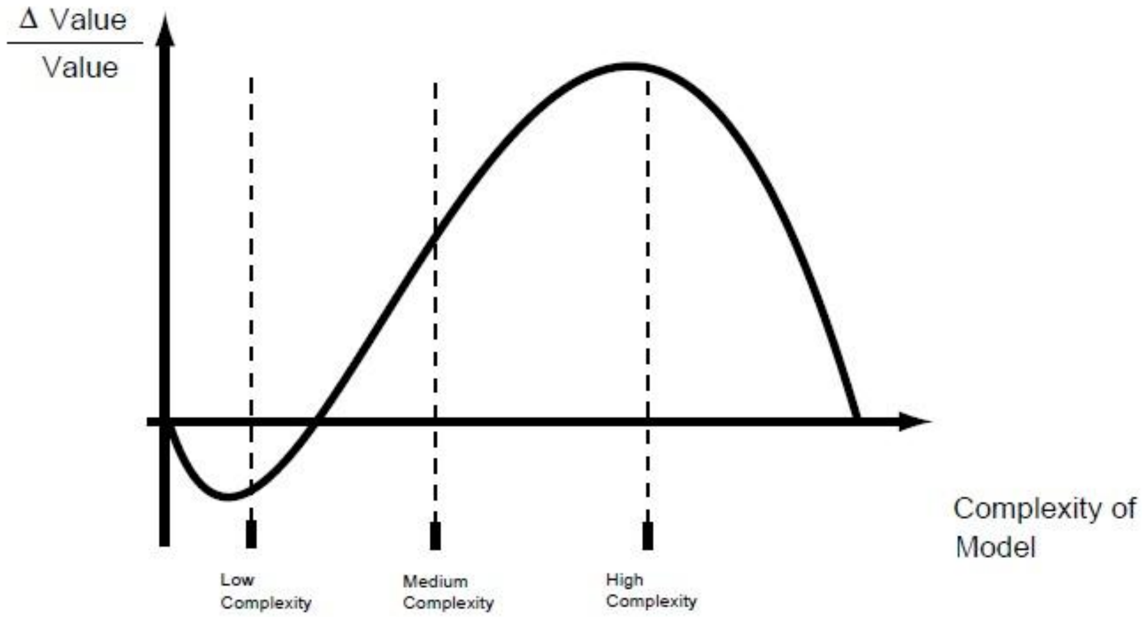


Figure 23: Value rate of modeling (Deremaux, Willcox, & Haimes, 2003)

Figure 26 shows the balance between having a model, CAD and CFD, which balances accuracy and computation time (represented by complexity). Such a model is accurate enough to represent reality and gives added value to the designer without using too much time.

1. *Low complexity*; the model encapsulates only general ideas of the design at its simplest level. Typical when talking of CFD modeling that would intend just potential linear codes and with a very easy CAD model with very few variables. The time spent in modeling will give no new answers to the designer.
2. *Medium complexity*; the model encapsulates the details of the design that are necessary for physical understanding. The CFD method is still potential codes but with nonlinear wave making codes. Fully or partially parametric modeling is considered. This is a typical way of doing exploration search in an early design stage, but the model is not expected to generate any new unexpected results.
3. *High complexity*; the model is an exact representation of the physical system, including details such as fillets. The CFD codes will have accuracy, typical RANSE codes for turbulent

flows in the aft ship. Fully parametric modeling. To have such a high rate of complexity and a high added value rate would intend that not many RANSE computations are done. Potential codes could be used in such a high complexity and having just as high value rate by weighing up the lack of accuracy with more computation in a time period then RANSE Codes.

There is however a possibility of having a model that is very complex but does not give any added value. An example would be an optimization model that is very complex, and then the answers will give new questions to the problem and no answers to the problem. Modeling CAD-CFD is most about raising the right questions, i.e. in seakeeping small local change may not be important for the motions. A very complex model of the ship is not necessary, because the value of the time spent is not gained in the value of the model answers.

How much value that is gained is also a function of the designer's experience and knowledge in simulation driven design. The designer must have experience in parametric design, optimization, CFD and also an innovative set of mind. How good the rate of value as a function of complexity is then much up to the designer and the task at hand.

10 ROBUST DESIGN

Much often in design the designer assumes that the design parameters or input environmental variables are deterministic. This is to make the design process easier for the designer. But in the resent years in ship design there have been more focus on the robustness of a design. This is because the operating profile and operating life of the ship is not influenced by constant variables, and there seems to be economic profits in making a ship more robust.

In smaller product design, i.e. electrical engineering, robustness has been applied in many decades but not in much bigger design products like airplanes and ships. The car manufacturing industry has been more innovative in its design approach, i.e. set-based design and Toyota, and some car manufacturers like Nissan have tried to implement robustness strategies in the design.

10.1 Robust design definition

There are some different definitions on what robust design is, here are two definitions taken from (Wang, Wu, & Lust, 1997) and (ASI):

> *Robust design is designing products and processes that are minimally impacted by external forces such as environmental, customer use or manufacturing conditions.*
>
> *Robust design is a design whose performance is not unacceptably compromised by expected variations in parameters which are known to affect its performance, and is more tolerant to unexpected variations.*

The first definitions intend that robust design has to do with external variables that influence the product in its life time and that the product should be minimally impacted by these external variables. The second definition says that the variation in the external variables should not give a bad impact on the products design performance, i.e. higher resistance for ships because of big variation in sea state, and that it can tolerate the variance in the external variables in the best way.

The approach for dealing with robust design was pioneered by Dr. Taguchi who made a general model in how to make a design more robust. His intention was to improve the fundamental function of the product or process, thus facilitating flexible designs and concurrent engineering. Taguchi developed the theory in the post second world war, and introduced statistics to improve the manufactured goods. He realized that much of the industrial production was concentrated of producing the product on target, i.e. a specific diameter, a cell for a specific voltage, or in ship designs a specific speed. He realized that a poorly made product was a product that could not satisfy its intention of use because of variation of the environmental conditions. To make a quality product he therefore introduced noise factors (environmental variation, manufacturing variation or component deterioration) and cost of failure functions in the phases of design. The noise factors are simply factors that can influence the performance of the product. The cost of failure is a products cost of rework or scrap when it deviates from its specific target usage. This is the general idea of cost functions but Dr. Taguchi broadens the idea of cost functions to also include the *cost to society*. A nominal product would result in some loss to the customer or to the wider community. These losses are external losses and are usually ignored by the product developers and manufactures, which are more interested in their *private costs* than *social costs*. Taguchi argued that such losses would find their way back to the originating corporation that produced or designed the product (in an effect similar to the Tragedy of the Commons (Hardin, 1968)), and that by working to minimize them, manufacturers would enhance brand reputation, win markets and generate profits (Taguchi, 1993)

In summary robust design is about identifying the input variables that are uncertain and give them probabilities and introduce penalties of failure. Managing the uncertainty and giving the product the ability to handle such uncertainty is robust design.

10.2 Robust ship design

In ship design a ship is robust when it can handle many different situations that can be expected. When looking at ship hull design there are many uncertain variables see figure 21.

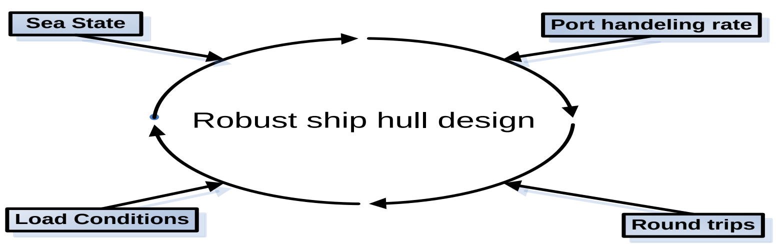


Figure 24: Some uncertain input data in ship design

To give the external and internal variables probabilities would also intend to have some deviation of expected values, this could be of difficulties in ship design because operating and environmental conditions are ‘intrinsic’ stochastic functions, whose expected values and standard deviations can neither be influenced by the designer nor by the manufacturer. A standard ‘deterministic’ approach to manage different operating conditions in design optimization is to take into account an aggregate objective function (AOF) as a linear combination of the system performance, evaluated in different operating points (Diez & Peri, 2010). The focus is on the uncertain variation of the operating conditions, addressed from a stochastic point of view. The optimization process is to minimize the effects of the uncertainties involved.

There exists much information of robust design strategies and it is an interesting topic with respect to ship design. In ship design there is not much literature on the topic, but as referred to before (Diez & Peri, 2010) has a good introduction in their paper. Other literature is (Hannapel & Vlahopoulos, 2010).

10.2.1 Aggregated objective function (AOF)

As explained an aggregated objective function is not a fully probabilistic approach in design optimization and methodology, but it is a way to simplify it to a more deterministic point of view. Equation 10.1 is an example of such an approach. Taken into consideration is the different operating conditions represented by different draughts T_{nm} ; in the equation represented by N . The resistance $R_{T_{nm}}$ is calculated in different operating draughts which has its probability k_n , see figure 25. The probability could be in the ships lifetime or in one year. k_n could also represent the trade-off between the different objectives. M in the equation 10.1 represents the different design lines or the different geometrical shapes that is investigated during the optimization process. The objective is then to minimize the resistance. In one of my research examples I will use AOF as a way to simplify the multi objective problem.

$$\min Z = \sum_{n=1}^{N} \sum_{m=1}^{M} R_{T_{nm}} k_n \qquad (10.1)$$

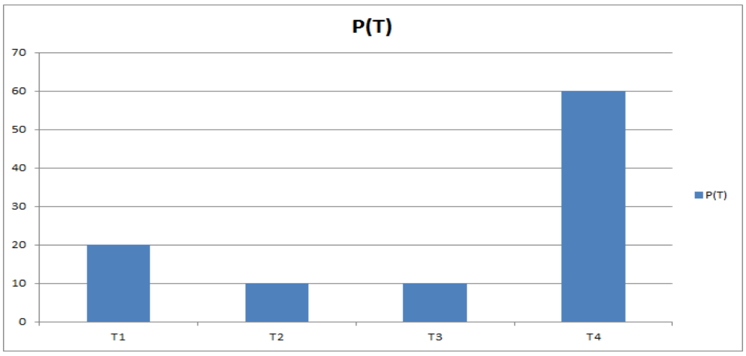


Figure 25: Probability of four different operating draughts

Figure 26 shows three different design lines and its resistances in different draughts. It is an illustrated example of how it could look like. In an optimization process there could be thousands of different designs that are evaluated.

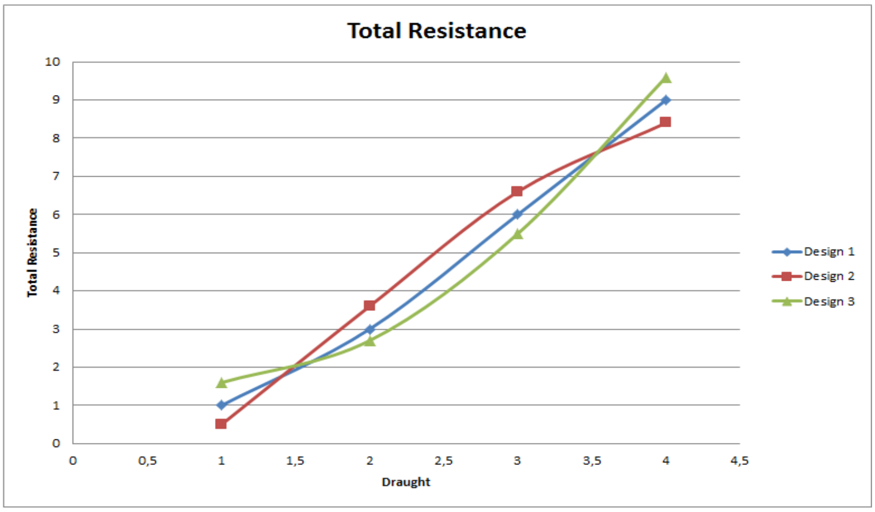


Figure 26: Example of different design lines and the resistance in different draughts. Which design that is chosen will be very much affected by the percentage factor

There could also be another approach by having an original design and look at the change in geometry with respect to the original hull. The objective will then be to minimize the deviancy from the original hull and the other geometrical shapes, see equation 10.2.

$$\min Z = \sum_{n=1}^{N} \sum_{m=1}^{M} (R_{T_n} - R_{T_{nm}})k_n \qquad (10.2)$$

11 RESEARCH EXAMPLE

In my thesis I have written about different design methods or different approaches of how to obtain a conceptual design. I have also mentioned some new trends in ship design; robust ship design, optimization design and CFD. I will make use of some examples that will use all of the above trends. I will make use of a framework that contains CAD modeling, different optimization methods and CFD methods (FRIENDSHIP SYSTEMS, 2009). This framework is tightly coupled with the CFD program; SHIPFLOW. SHIPFLOW contains different approaches of CFD called XPAN, XBOUND and XCHAP. XPAN is a potential code, XBOUND is a boundary turbulence approach and XCHAP is a RANSE code.

The idea of the model is to get knowledge in early stages of design where little information is present of how the geometric shape will change with a robust design approach. In a design group the model represents the hydrodynamic contribution's point of view. As explained one would favor a robust investigation of the hull in the beginning of a design stage where most of the decisions are made. Later in the design process there would not be the change of a robust hull investigation.

I will try to make a very easy model of the ship so it is robust for a large geometric change. I will try to make a model with low complexity and with a medium CFD accuracy so that I can generate many sets of design in an optimization process. My value rate of each cycle will then be in the medium complexity range as described in figure 26. The reason why I didn't try to get a higher value rate was because of the errors that are contained in the simulations. To make a high value rate model, I would have to be more experienced in CAD modeling and with the CFD method.

I have divided the research example in three different simulation runs; One that search for the midship geometric shape with a multi objective approach and one with an aggregated function (AOF) approach to the same problem but with a single search optimization method. The last simulation run I have chosen to investigate a simple geometric change in the bow part of the ship. The latter was the most challenging when thinking of simple parametric modeling.

11.1 Robust ship design

As mentioned, robust design is about dealing with uncertain input variables. These variables could be uncertainties in the building process or having to do with the operation of the ship. My input variables for this specific model are different draughts. To see how robust my hull shape is would intend a multi objective process. Here the objectives are resistance with respect to two different draughts. The goal here is that there may be an economic gain in having more resistance in the design line but better in the other loading conditions. How much time the ship is spending in the different conditions will be the decisive variable, and this is also not a deterministic variable but probabilistic. But because this research intention is not to go deep into probabilistic ship design I will say that my input variables are deterministic. I can still have a robust design approach by using an aggregated objective function (AOF) approach explained in Chapter 10 or by just minimize each objective in a multi-objective search algorithm.

11.2 The parametric CAD model

I would need to start out from a very simple model to identify how the ship hull will change. The CAD model must be a trade-off between simplicity without losing flexibility. The goal of my model is to identify changes in the geometric shape so that I can have an understanding of how to make a more complex parametric hull shape of a robust ship.

When modeling, I must have in mind four things:

1. What kind of optimization process am I using? This is not the most important thing to have in mind but important when modeling is if you have a one objective or a multi objective optimization. You have to allow for trade-off in the process.
2. What kind of CFD method? What is the value of the result you get? This will intend how much time you would want to use, how accurate results you want and how complex the CFD method is.
3. What is my intention of the model? What kind of shapes do I expect, and can the model change into this shape. If having a multi objective approach the hull must be able to be 'optimized' in more than one area, i.e. the hull must be able to create a hull kind of shape for bigger velocities and a round shape in low velocities. This enables trade-offs in the optimization process.
4. Errors that may occur in the process. Make your model easy to understand so that errors when running the model may be easy to find. 'It's not hard to make it difficult it's more difficult to make it simple'.

When having these four things in mind, the parametric shape may take form.

11.2.1 Hull 1

Its purpose is to investigate the midship section. I have created a simple NURBS (Non-Uniform Rational B-Spline) line which can be changed by two points (p_{z7} and p_{y6}). The theory is that it will create possibility for geometric change because of two different draughts (T_{dwl1} and T_{dwl2}). p_{y6} can only have a variable change in the Y-direction and p_{z7} can only have variable change in the Z-direction. To couple these points to the global parameters I have chosen a percentage approach; p_{z7} is a variable percentage of the global depth parameter and p_{y6} is a percentage variable of the global breath parameter. The global parameters represent then a fully parametric variation and the points represent a conventional variation of parametric modeling. The variable range is decided in the optimization model, see Section 11.4.2

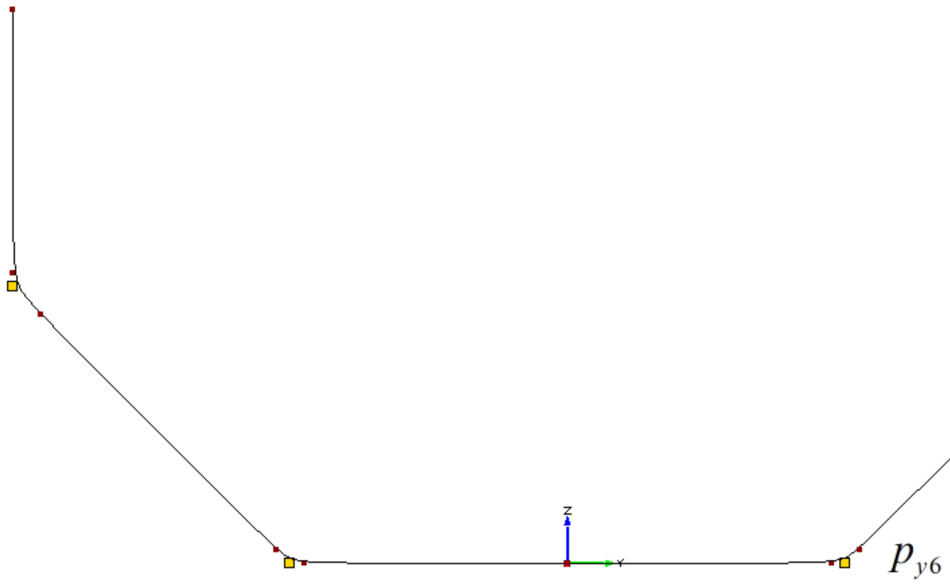


Figure 27: Midsection of the hull

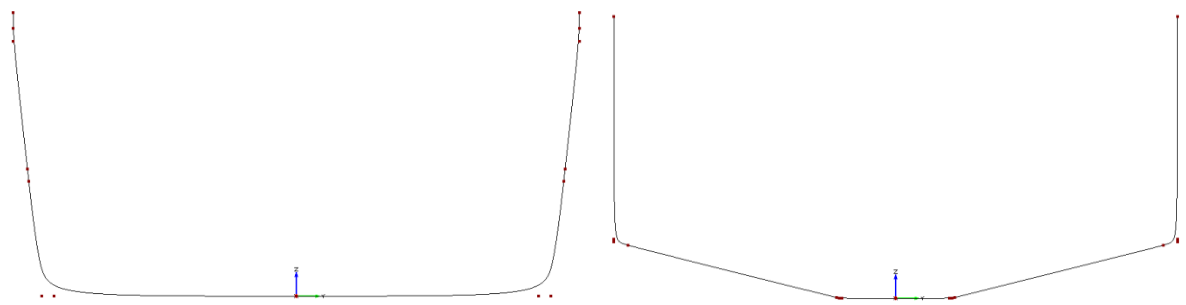


Figure 28: Outer range of the geometric change in the hull

The rest of the hull is created very simply by two other lines that represent After Perpendicular (AP) and Forward Perpendicular (FP). The surface is then created by creating a lofted surface between the lines.

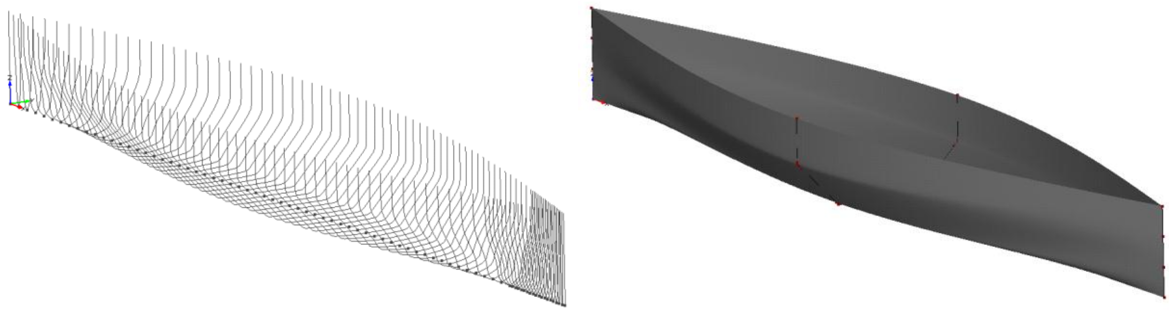


Figure 29: To the left the offset sections for the auto meshing and to the right the representation of the hull

Global parameters:

Length: 180 [m]
Breath: 30 [m]
Depth: 15 [m]
Dwl1: 10 [m]
Dwl2: 5 [m]

11.2.2 Hull 2

The purpose of this hull shape is to investigate change in the bow section. This is a difficult area to approach, because you would want to create a big variety of geometry with the same parameters. You would also use few parameters so that the optimization process does not have so many free variables and so that the model does not get too complex. I tried many different approaches, but ended up with this one. It is maybe not a complete model of bow geometric search, but I have had some problems in creating the offset sections which is the input of the hull to the CFD program. This have led to a more simple model.

The bow section is created by two curves; one in the X-plane and one in the Y-plane (figure 30). The idea is to have a simple variation scheme. The curve in the X-plane can be changed by three points and has five free variables that change in the optimization process. The curve in the Y-plane is the stem profile of the bow shape and can change with one point that represents the tip of the bulb, and has two free variables. The free variable is parameterized by a percentage factor of the global dimensions similar to hull 1. The theory is that it will create possibility for geometric change because of two different draughts (T_{dwl1} and T_{dwl2}). The variable range is decided in the optimization model; see Section 11.4.2

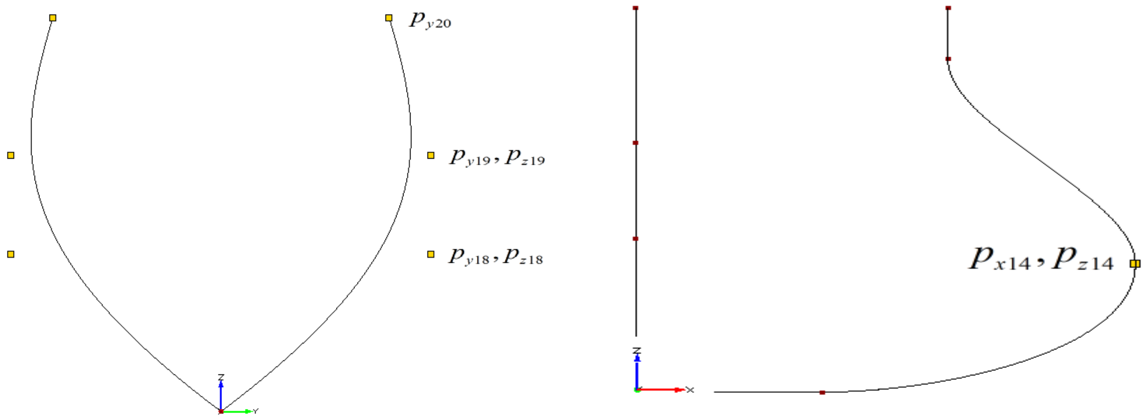


Figure 30: Curve to the left is in the X-plane and the curve to the right is the stem profile in the Y-plane

The rest of the hull is created by a line that represents the AP and a midsection. The bow surface is created by a lofted surface and the aft part is also created by a lofted surface. Between these two surfaces there is created a fillet surface that is tangent to the two surfaces.

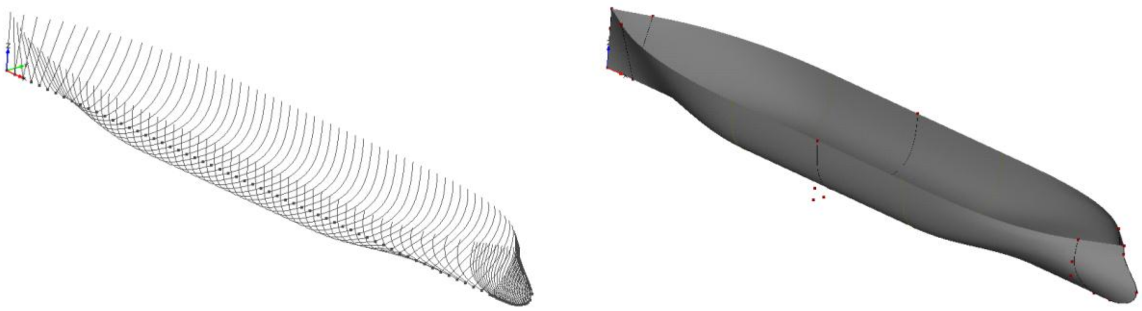


Figure 31: To the left the offset sections for the auto meshing and to the right the representation of the hull

Global parameters:

Length: 180 [m]
Breath: 32 [m]
Depth: 15 [m]
Dwl1: 10 [m]
Dwl2: 5 [m]

11.3 CFD method

The CFD method is based on how long one run takes. Because I am running an optimization process I would like one simulation to take as little time as possible in order to increase the value of the process. To use a CFD method that is using little time would intend a potential code method. The method is also based on the objectives that I want to find; wave resistance.

The CFD method is a non-linear free surface method with free sinkage. It is called XPAN SHIPFLOW and is developed by (Flowtech int., 2010). The method calculates the wave resistance coefficient C_W for the hull and also the area of wet surface S. The wave resistance is much dependent on the bow section and the midsection, and as explained that this will be my focus of geometric change.

The total resistance of the hull is expressed often as coefficients (Steen, 2007):

$$C_T = (1+k)C_F + C_W \tag{11.1}$$

Where the total resistance is:

$$R_T = C_T \frac{1}{2} \rho V^2 S \tag{11.2}$$

C_W is the wave resistance coefficient and C_F is the frictional resistance coefficient, k is a form factor that takes into account the 3D effect of the wetted surface. The form factor is difficult to decide in an easy CFD model and will be neglected, but it could be found by using different empirical formula that is based on towing tank tests. Because I neglect the form in the frictional term C_F will then be dependent only on the speed of the ship if using a friction line, i.e. ITTC-57 (Steen, 2007). To start with I will not have speed as a variable, but only the draught, that means that I can simplify and neglect the frictional term. The resistance will then be dependent on wave coefficient and the wetted surface:

$$R_W = C_W \frac{1}{2} \rho V^2 S \tag{11.3}$$

The input variables for the program are the offset sections which are divided into bow section and main hull section. I have not added other sections i.e. stern part because I will not do any geometrical change in this area. But some ship models need to have a divided group of stern parts like transverse stern or boss to get adequate results. Other input variables are the speed and what kind of hull it is. I have chosen a speed of 18 [knots] for hull 1 and 14 [knots] for hull 2 and a mono hull shape application in the CFD code.

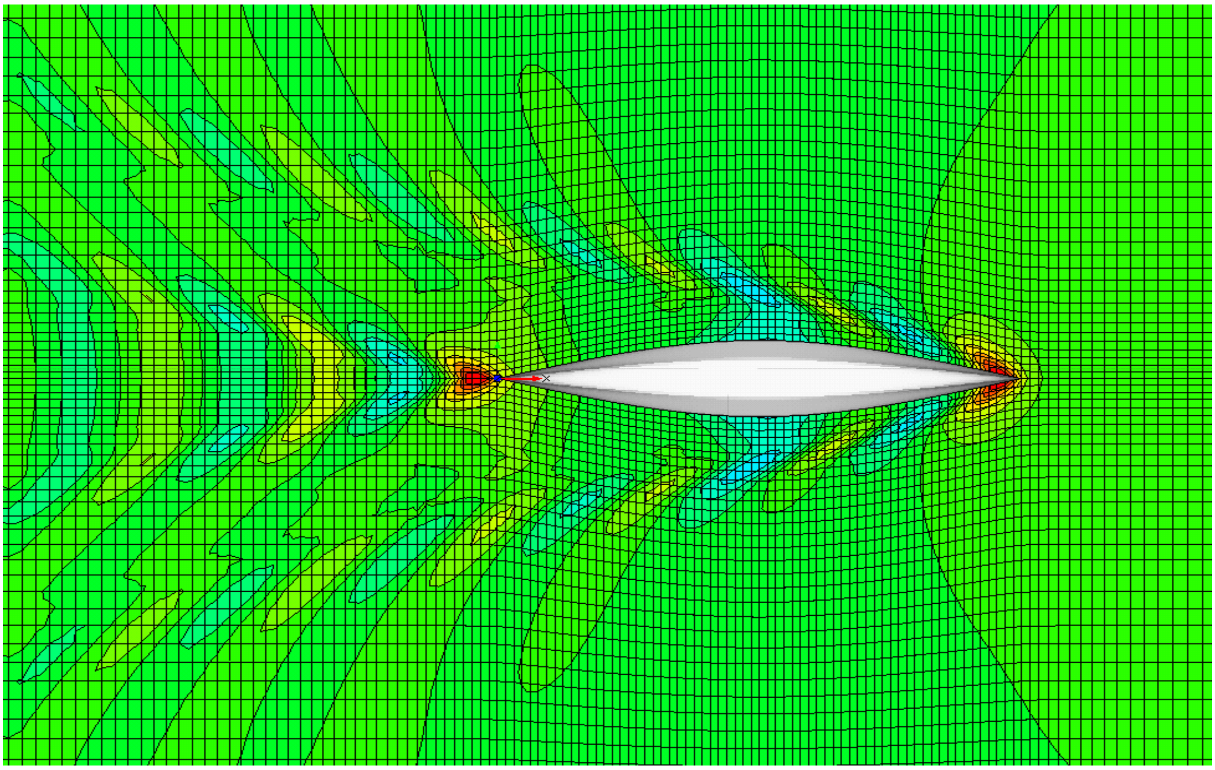


Figure 32: The wave elevation shown in colors and the mesh grids of the free surface

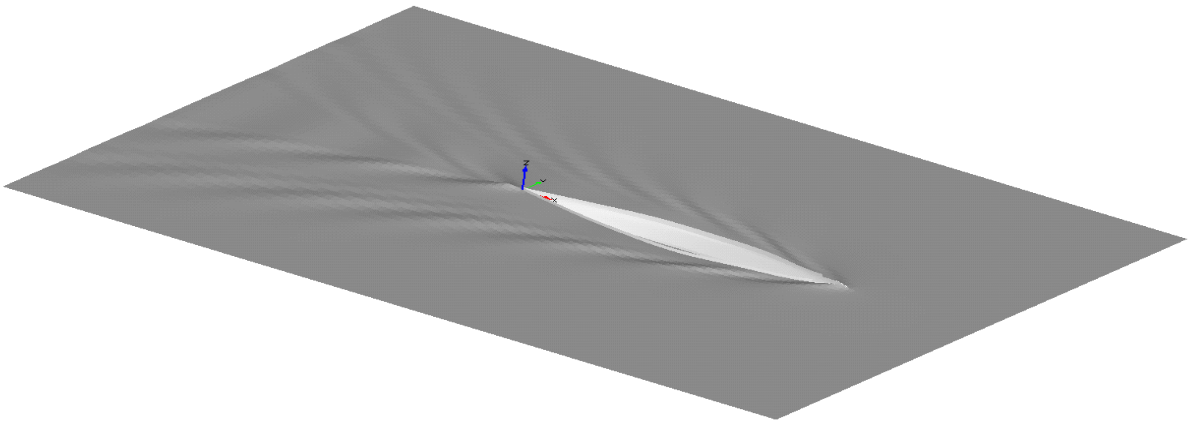


Figure 33: The CFD program computes the wave elevation and the wave propagation for hull 1

11.4 Optimization model

The choice of optimization model is based on if it is a multi or a single objective function. I have done three runs; two with multi objective search and one with an aggregated objective function (AOF) which I have treated as a single objective search.

11.4.1 Optimization method

For the multi objective search I have chosen a NSGA-II (Non-dominated Sorting Genetic Algorithm-II) method. This is an evolutionary algorithm that can be used with inequality constraints. It creates a random selection of the variables called a population. The individual of variables that has the best objective(s) will have stronger possibility to survive, and so the iteration starts and goes one. I selected a population size of 15 and a generation of 18 for hull 1. For hull 2 I chose a population size of 20 and a generation of 20, which give 400 simulating runs. But because I calculate two draughts in one simulating run the CFD calculations will be 800 runs. The NSGA-II algorithm also needs input of probabilities of mutations and crossovers. A mutation is a totally new individual of variables while crossovers are the pairing of two individuals. I have put the probability of mutation to 10 % and crossovers at 90 % for both hull 1 and hull 2. The crossover probability affects the process of mixing up the genes of the parents while creating a child, e.g. if this value is zero the genes of one single-parent are just copied to the child without crossing with respect to the other parent.

For the single objective I have chosen a Tangent Search method. The Tangent Search Method promises to be a reliable solver for small scaled optimizations problems with inequality constraints. The major features of the Tangent Search Method are to detect a descent search direction in the solution space, to ensure fast improvement in the promising search direction, and to keep the search in the feasible domain. Within the permissible solution space the Direct Search Method is applied which consists of exploratory moves that start from a so-called base point along the variable axes followed by global moves in the descent search direction found in successful exploratory moves. If a constraint bound is approached a tangent move in hyperspace is conducted tangential to the constraint either to keep the search in the feasible domain or to bring it back to the feasible domain. The method is capable of detecting a local minimum of the solution space which is of dimension $N*V$ according to the number of free variables. A descent search direction is determined by at most $2*N*V$ function evaluations. Free variables are subject to explicit bounds, i.e. a lower and an upper bound. Satisfactory results are usually obtained by setting the initial step size to be 5% to 10% of the respective variable range. The minimum step size is about 5% to 10% of the initial step size. For this method I have chosen max 150 iterations and a variable tolerance at 0.001.

11.4.2 Mathematical models

The Tangent search Method for hull 1 with the aggregated objective function:

$$Min\ Z = 0.8C_{w\_dwl1} + 0.2C_{w\_dwl2} + 0.8S_{dwl1} + 0.2S_{dwl2} \tag{11.4}$$
$$0.2 \le p_{z7} \le 0.9 \qquad [\text{-}] \tag{11.5}$$
$$0.2 \le p_{y6} \le 0.9 \qquad [\text{-}] \tag{11.6}$$
$$\nabla \ge 20000 \qquad [\text{m}^3] \tag{11.7}$$
$$KM_{dwl1} \ge \frac{D}{2} \qquad [\text{m}] \tag{11.8}$$
$$KM_{dwl2} \ge \frac{D}{2} \qquad [\text{m}] \tag{11.9}$$
$$\left\{\text{Start: } p_{z7} = 0.9 \quad p_{y6} = 0.9\right\} \tag{11.10}$$

This model is to investigate how to use an aggregated objective function compared to a multi-objective optimization search. The optimization model starts to search out from the initial values (eq. 11.10). Also I have chosen to say that the ship is operating 80 % of its time in dwl1 and 20 % in dwl2. Except from the change in optimization process and the objective function the ship model and mathematical optimization model is the same as the NSGA-II search for hull 1.

The NSGA-II multi objective search for hull 1:

$$Min:\ C_{w\_dwl1},\ C_{w\_dwl2},\ S_{dwl1},\ S_{dwl2} \tag{11.11}$$
$$0.2 \le p_{z7} \le 0.9 \qquad [\text{-}] \tag{11.12}$$
$$0.2 \le p_{y6} \le 0.9 \qquad [\text{-}] \tag{11.13}$$
$$\nabla \ge 20000 \qquad [\text{m}^3] \tag{11.14}$$
$$KM_{dwl1} \ge \frac{D}{2} \qquad [\text{m}] \tag{11.15}$$
$$KM_{dwl2} \ge \frac{D}{2} \qquad [\text{m}] \tag{11.16}$$

I have only chosen two free variables for this simple search of midship geometric shape. Stability (eq. 11.15 and 11.16) and displacement (eq. 11.14) are normal constraints in ship optimization process. Because I not know where my center of gravity KG is present I estimate it to be half the depth of the ship, thus the KM must be bigger to have a positive GM value. There is two different draughts and the global parameters are not taken into consideration. Typically in global search the length will be decided by the volume displacement constraint and the stability will decide the breath variable. The resistance search optimization process will then try to make the ship longer while making it narrower. This is common knowledge for a ship designer and to make such a model will be in the low complexity and negative value area of the figure 26 in Section 9.1.

The NSGA-II multi objective search for hull 2:

$$Min: C_{w\_dwl1}, C_{w\_dwl2} \tag{11.17}$$

$$1 \leq p_{x14} \leq 1.1 \qquad [-] \tag{11.18}$$

$$0.1 \leq p_{y18} \leq 1 \qquad [-] \tag{11.19}$$

$$0.1 \leq p_{y19} \leq 0.5 \qquad [-] \tag{11.20}$$

$$0.1 \leq p_{y20} \leq 0.4 \qquad [-] \tag{11.21}$$

$$0.1 \leq p_{z18} \leq 0.4 \qquad [-] \tag{11.22}$$

$$0.4 \leq p_{z19} \leq 0.65 \qquad [-] \tag{11.23}$$

$$1 \leq p_{z14} \leq 1.4 \qquad [-] \tag{11.24}$$

$$KM_{dwl1} \geq \frac{D}{2} \qquad [\mathrm{m}] \tag{11.25}$$

$$KM_{dwl2} \geq \frac{D}{2} \qquad [\mathrm{m}] \tag{11.26}$$

To simplify things I have left out minimizing the wetted surface S and the volume constraint ∇, because this is more a local variable change and there will only exist small changes in volume and wetted surface. There are seven free variables which I have tried to tune into possible ranges so that not too much programming errors will occur.

11.5 Uncertainty and error

The result that I have gotten in these simple examples is not accurate results for still water resistance. What is important in such an optimization process is that they are just as accurate in each run and that they don't contain any programming errors. Results that differ much from the average result in the iteration are an indicator of error in the calculated run. What is an advantage with the NAGA II is that it will not get affected by some error result, but if there exist error result over and over again the optimization process is not going towards a usable result.

In getting reasonable results I have had problems. The problem is getting results for a big variety of the geometric change. The problem is developed in the process of creating the mesh or grid for the CFD analysis. Because it is an automatic grid generator, it is sensible for big changes that differ from the normal. However the CFD method used for wave resistance is very close to the actual wave resistance prediction and wave elevation.

11.5.1 Physical approximation error

The physical approximation errors in this research example are about the simplification that is done in the input parameters of the CFD program. Many of the simplification are done by years of experience of the program designers. One could argue that one of the physical approximation is to divide the resistance into different sections; one of them being the wave resistance. And that the wave resistance is adequately being calculated only by a potential code method.

11.5.2 Iterative error

I have chosen to put a number of how many iteration I want, this means that the search algorithm may not have found the optimum in the design space. But it is to save time and I am only interested in watching the trends and not to find the optimum.

11.5.3 Discretization error

In my model I have chosen a medium density of complexity of the mesh generator. It means that the result could have been more accurate, but it is not my intention to get very accurate results as explained.

11.5.4 Programming error

After using the program I have learned that complex tools like optimization algorithms, CFD and CAD all working in the same Framework is difficult to address for a program engineer. Different codes, input, output etc. is making it difficult to make a rough and robust collaboration. The program that I have been using is a big step towards such a robust tool, but I have had problems with the auto mesh generation for the XPAN free surface calculations and the XBOUND boundary viscous flow computations (I left out XBOUND boundary viscous calculations). This may be because I have not the proper knowledge in the program and that is merely a user error then a programing error. However I have used the program for two months and discovered that the more input that has to be decided before a run the more it constrains the robustness of the program. Take the example of the automatic mesh generator; it needs an input of a sectional model of the hull, which preferably should be divided into different groups like bulb, main hull, boss, stern etc. This constrains the use in a bigger geometrical change in the hull, and thus the program has difficulties developing the automatic mesh i.e. different bow shapes.

11.5.5 Usage error

I have made some simplification that will affect the accuracy of the result. One of them is as mentioned the mesh grid density. My user knowledge of CFD and CAD was very low before getting starting with simulating. This may have affected my process of getting some examples of simulation driven optimization design. The examples are based on the knowledge that I have gotten through this thesis and from a short experience of using the computer tool. The CFD computer tool that I have been using has many options and thus it will increase the user error, but it will also constrain the robustness.

11.6 Acknowledgment

These examples of a simulation driven design with an optimization process is a very simple one, but illustrates some of the results that can come from CFD and optimization. My original idea of how to use the tool was much more complex and the idea was to follow the approach described in Chapter 5 of how the trade-off between time and accuracy could be done by having many simple variants in the beginning and then narrow it down more and more to a model that is complex and with accurate CFD calculations (I didn't get the time to explore the use of RANSE codes in the program). This research example represents only the first stage. The reason for only coming to the first stage was problems with programming error and that I got the software very late in my process of writing the thesis. I had to do many simplifications and also learning the program took time. The programming error led to lesser variables in the optimization algorithm, a simpler hull shape (neglected bow shape variation together with midship variation) and less objectives (neglected speed variation). The main problem has been the coupling between CAD and CFD.

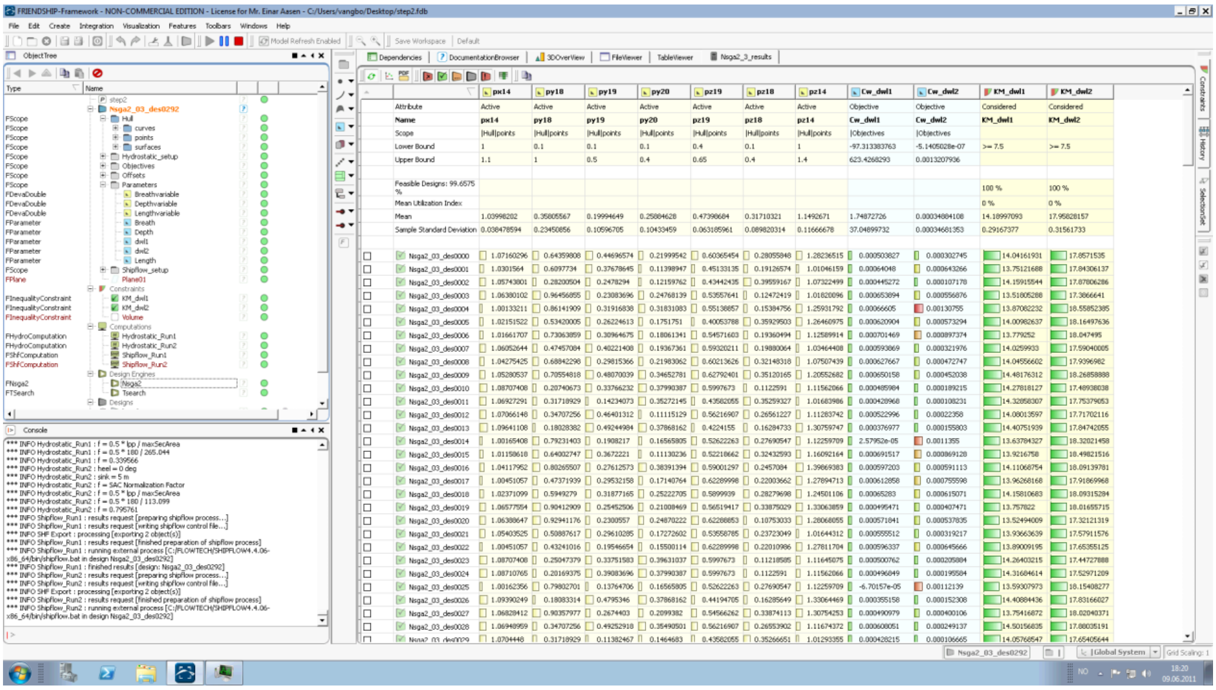


Figure 34: The framework window during the optimization process of hull 2

11.7 Results

The results of these examples are not my main goal of interest, but it is the process of how to get to the results that I have been focusing on. However I will present the results in a couple of 2D graphs and what that can be drawn from them. There are generated much output data from the optimization process. All the data is collected in the CD attached in the cover. The CD also contains the CAD models from the framework program.

11.7.1 Simulation run 1; Tangent search hull 1

In this simulation run the optimization process starts to search for a solution out from the initial values. As expected the process converged quite fast. If looking at figure 35 and figure 34 you would see a strong dependency of Py6 variable to the objective function. I seem that Py6 stopped at Py6 = 0.4 because of the displacement constraint. Going under this value would break the constraint as seen with the red spots in figure 36.

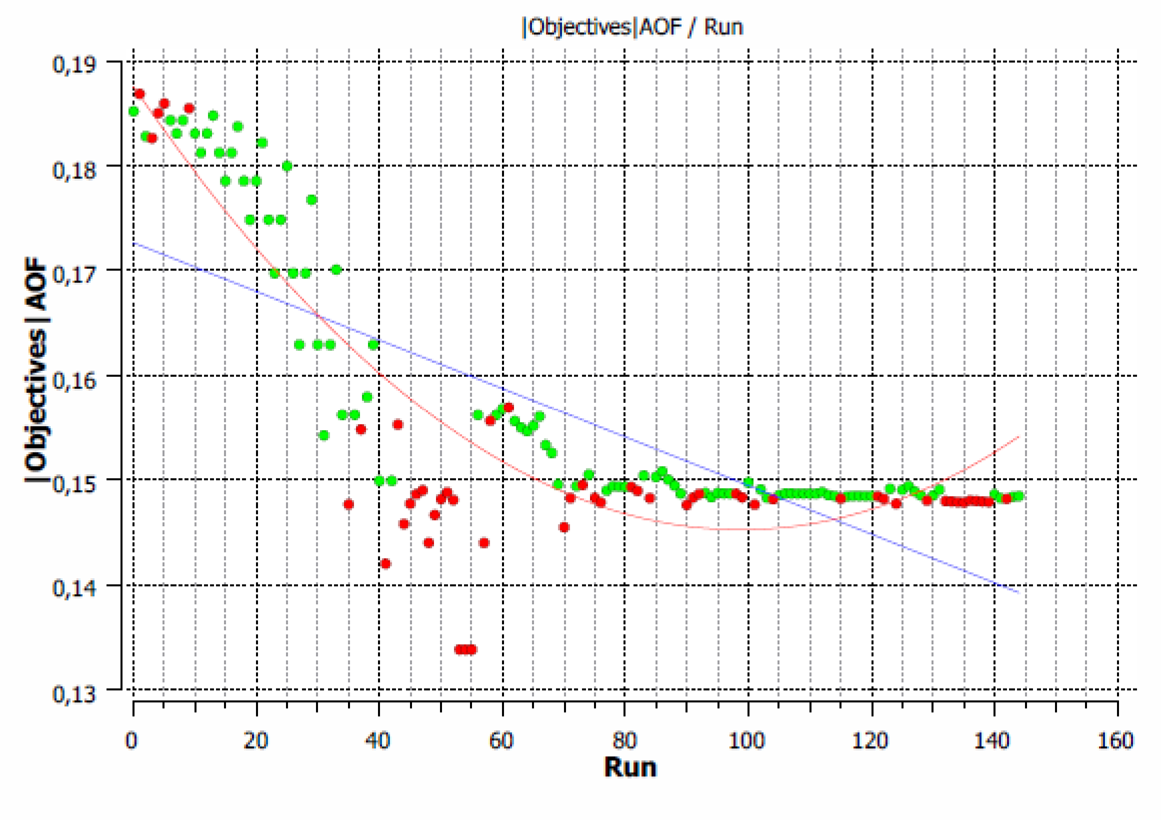


Figure 35: Aggregated objective function (AOF) and the search process

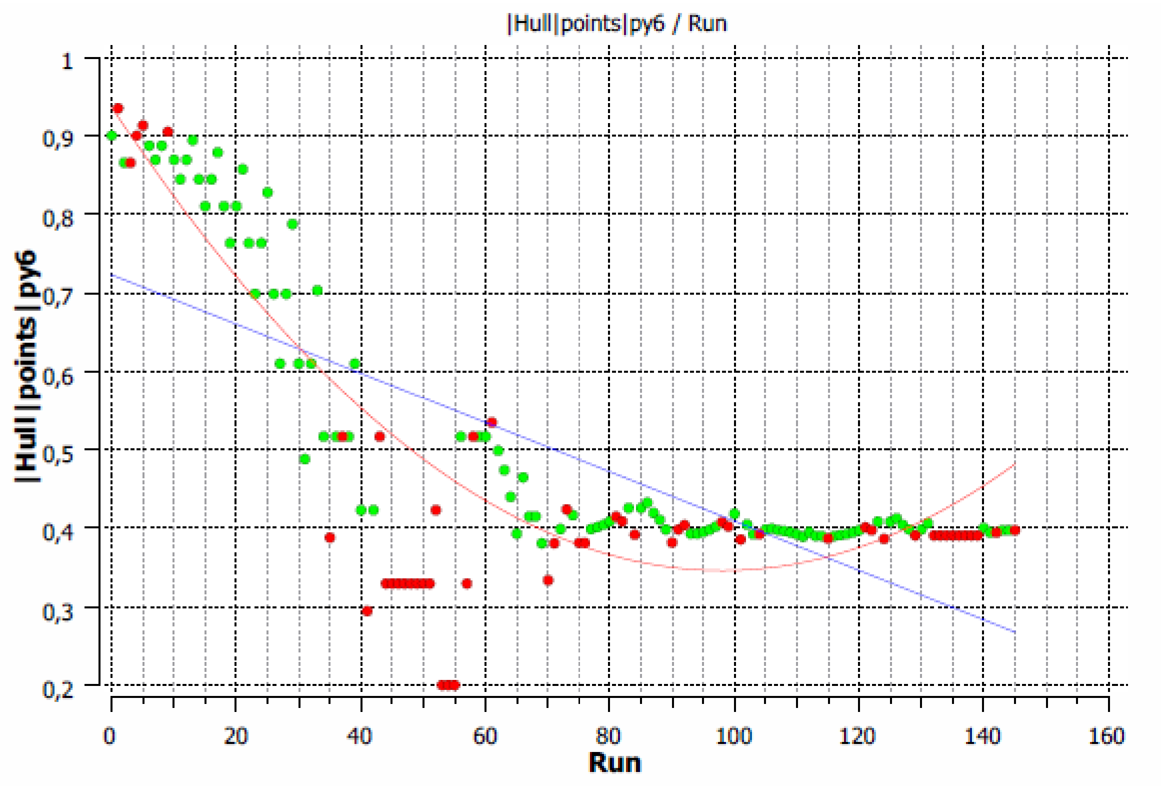


Figure 36: Py6 variable and the search process. It shows a strong similarity to the AOF graph.

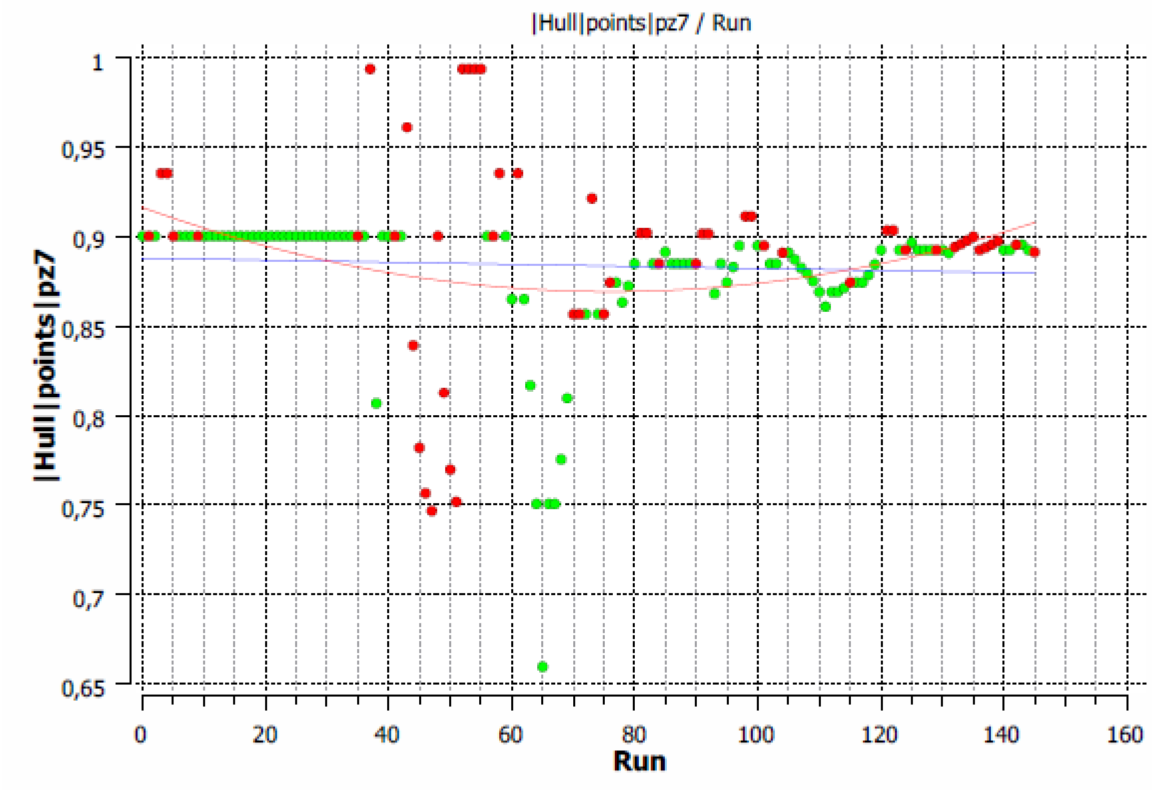


Figure 37: Pz7 and the search process. No strong dependency is found during the search process.

Figure 37 shows the other free variable Pz7. It seems that there is no strong dependency of this variable to the objective. Py6 is the dependent variable. This optimization is a local search process and is very dependent on where the search process starts; it may have looked different if searching from another point in the design space. The general conclusion from this search process is that Py6 is the most dependent variable when looking at the resistance. There may be a better option but the process was 'locked' into this solution because of the displacement constraint. To investigate further one could change the starting point or to change the trade-off factors in the objective. To investigate a more global search approach the next simulation run was chosen with an evolutionary algorithm.

11.7.2 Simulation run 2; NSGA-II search hull 1

Figure 38 and 39 shows the two wave resistance coefficients during the optimization run. Cw_dwl1 has ‘locked into two different solutions and Cw_dwl2 seems to be converging towards a solution. However the two other objectives S_dwl1 and S_dwl2 (see figure 40 and 41) seem not to be going towards a final solution. More iterations should have been done to get to a converged solution. There is however a possibility that there may exists two or more solutions that is close to one another in the design space that the free variables could have more than one solution to it. More than one possible good solution is good when doing trade-off with other requirements in the design process; it allows other options without penalty.

The red dots in figure 40 and 41 shows that there cannot exist solutions under S_dwl1 = 0.164 and for S_dwl2 = 0.094. The constraints in the optimization function do not allow solutions under these values.

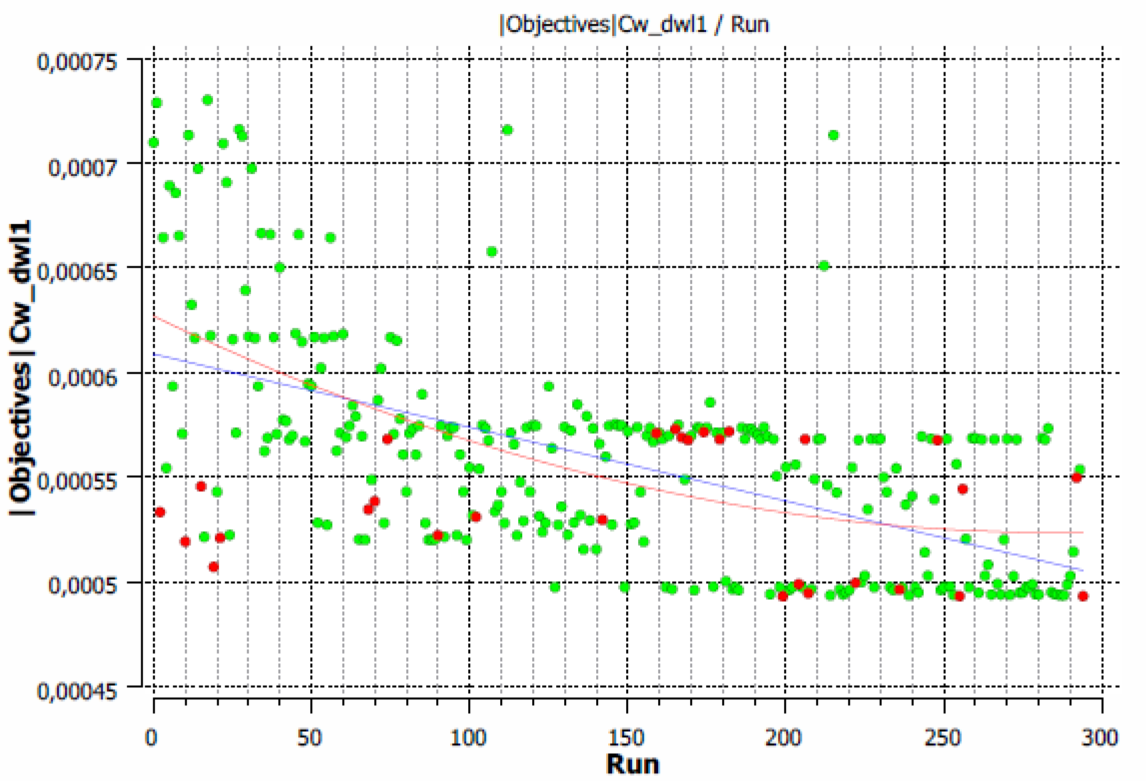


Figure 38: Cw_dwl1 objective during the search process

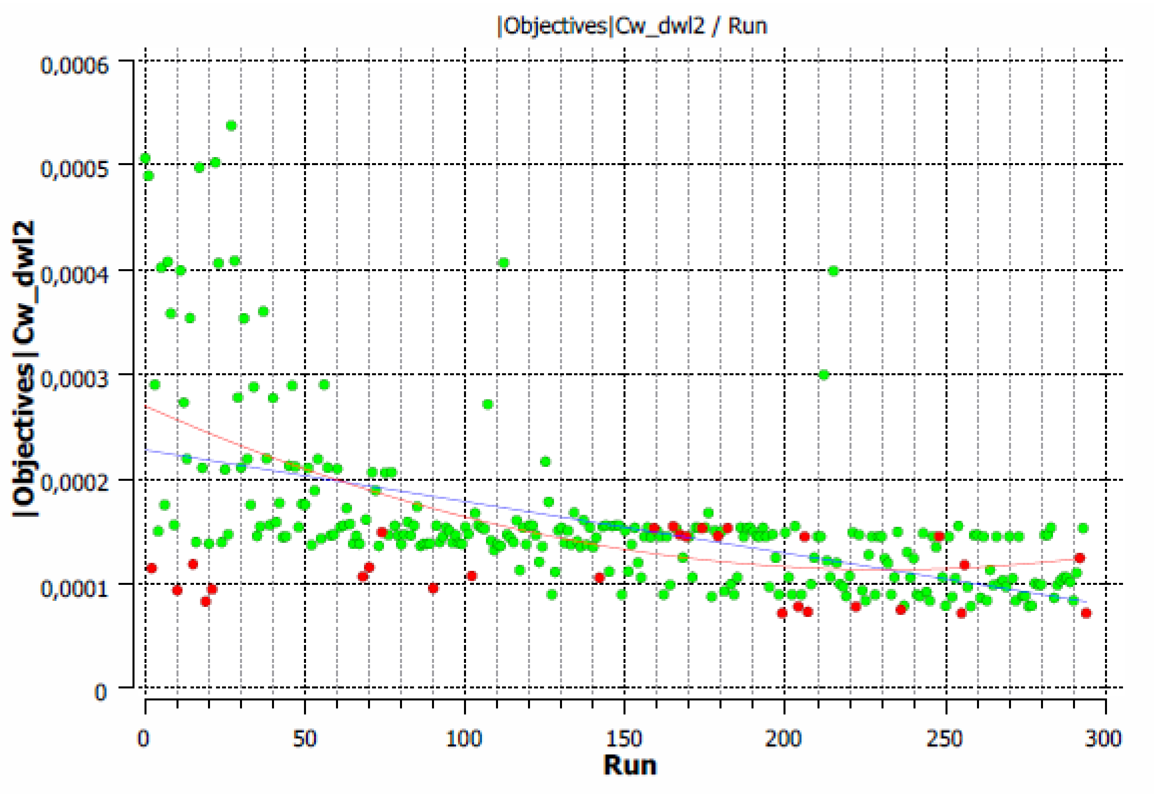


Figure 39: Cw_dwl2 objective during the search process

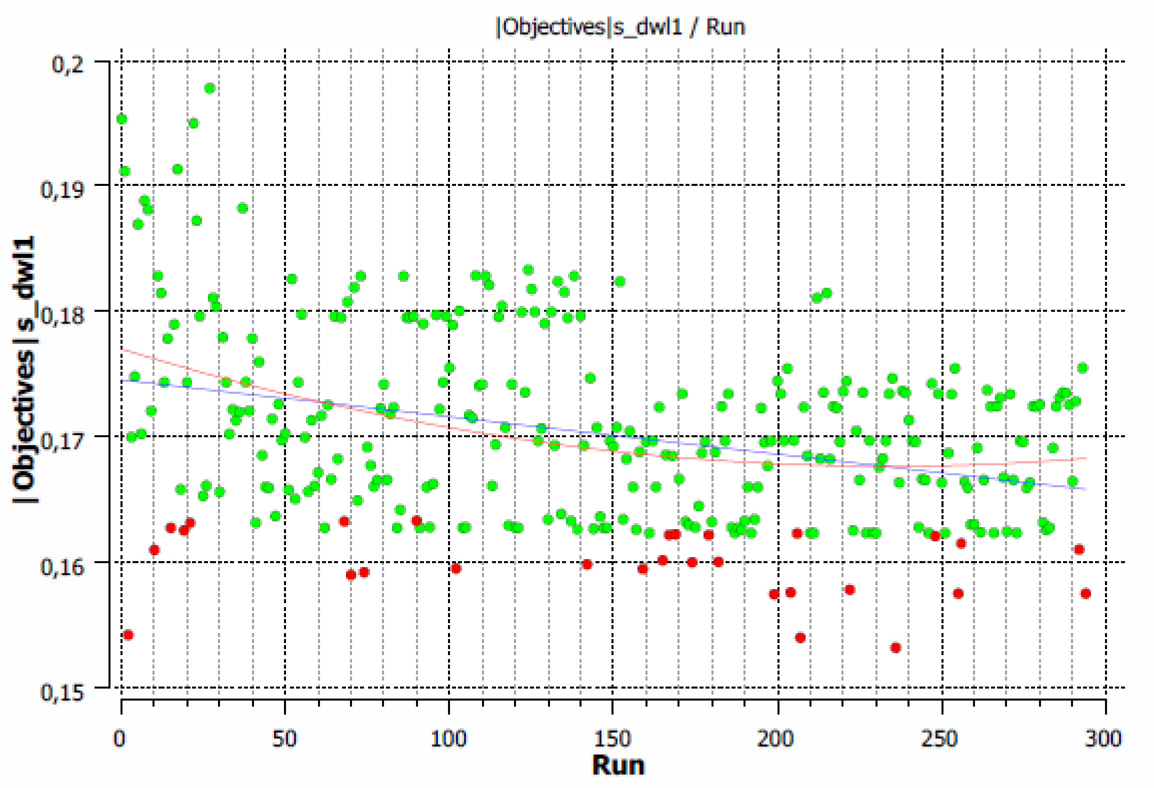


Figure 40: S_dwl1 during the search process

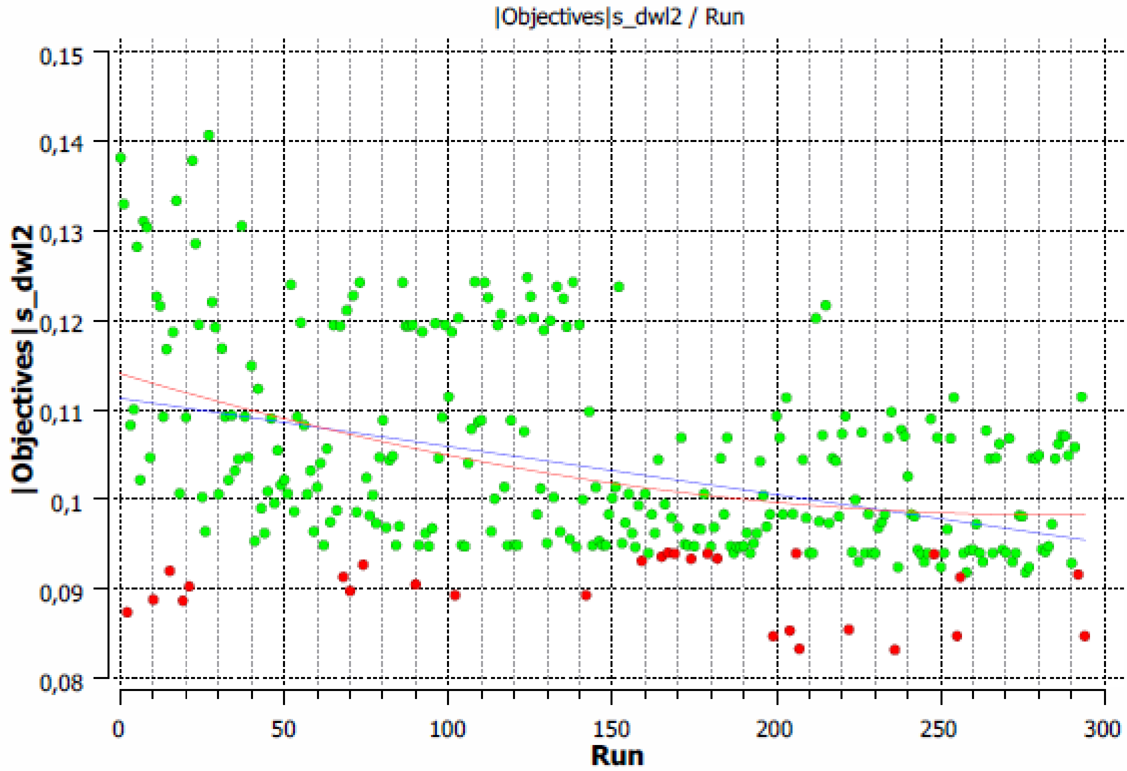


Figure 41: S_dwl2 during the search process

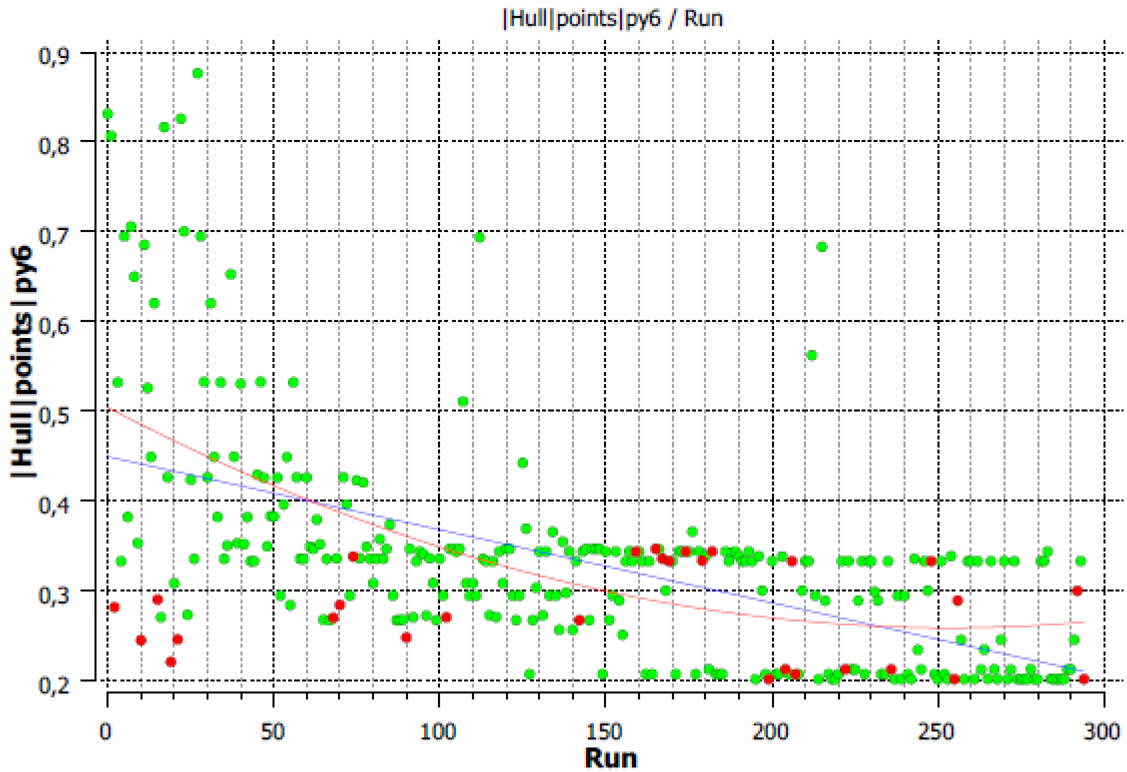


Figure 42: Py6 during the search process

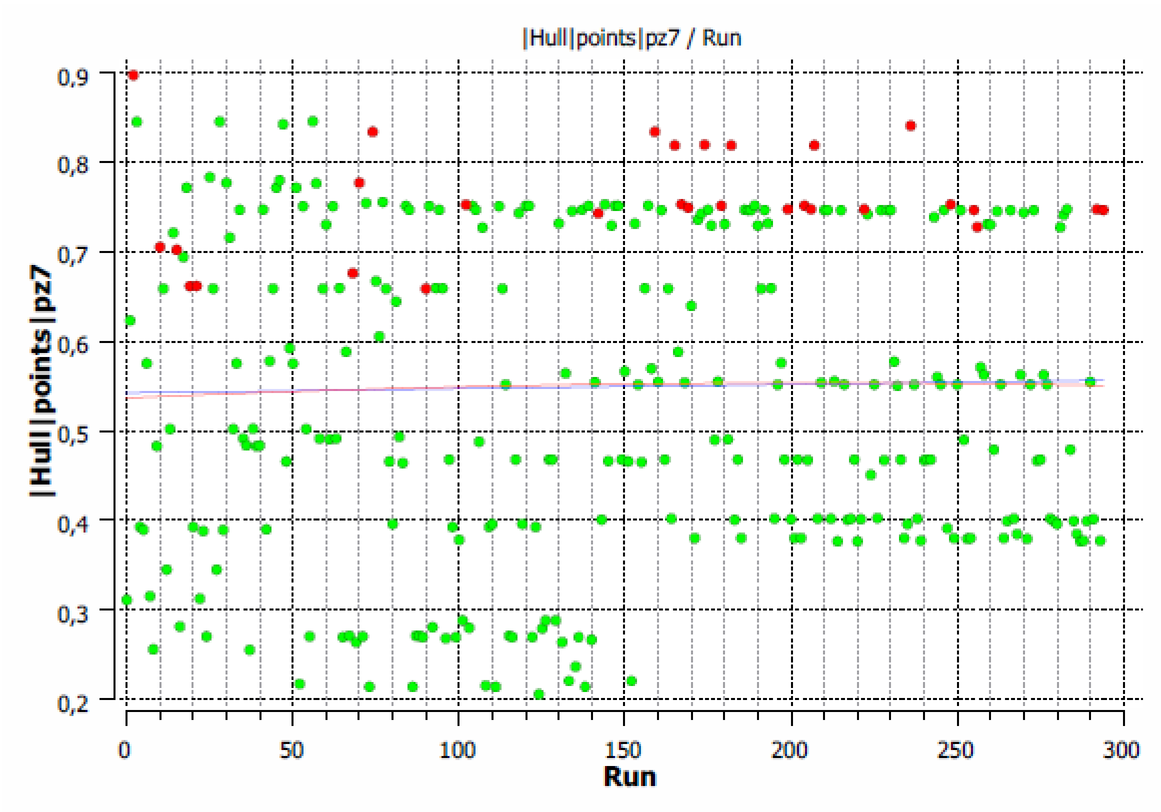


Figure 43: Pz7 during the search process

If the objective results do not converges to one solution, it would also intend that the free variable have not converged as well. Py6 (figure 42) seems to be going between two answers and Pz7 (figure 43) has no convergence. It indicates the same as with the Tangent search process; Pz7 is not a very dependent variable to the objectives. The conclusion is that the free variable is narrowed down further to a solution. A new search process should be done with more narrowed free variables.

11.7.3 Simulation run 3; NSGA-II search hull 2

This search process has seven free variables and contained much more programming error. I have deleted error results such as very high numbers or negative numbers, which counted up to 80 simulations. The result shows no tendency for convergence after 400 simulations, which imply that many more iterations should have been done. However it may also be that the programming error and meshing error in this model have affected the process. If looking at the result of the objectives (see figure 44 and figure 45) it seems it is a great diversity in answers. It could be that the bow shape has such a high influence on the wave resistance.

I will not go through all of the results from this search process (it can be taken from the CD attached), because of the uncertainty and error that the results contains. But some conclusion can be drawn when looking at the results. The variable that decides the bow or the stem profile Px14 could have been leaved out from the search process. It seems that for two draughts a bulbous bow is not needed, though it may have a good affect in one draught. Also it seemed that the optimization process tried to narrow down the bow shape as much as possible by lowering both Py18 and Py19 to its lowest limit which makes sense in a hydrodynamic point of view.

Figure 44 has also pointed out the ten best design solutions in accordance to the wave resistance in water line 1. The design solutions have also been pointed out in figure 45.

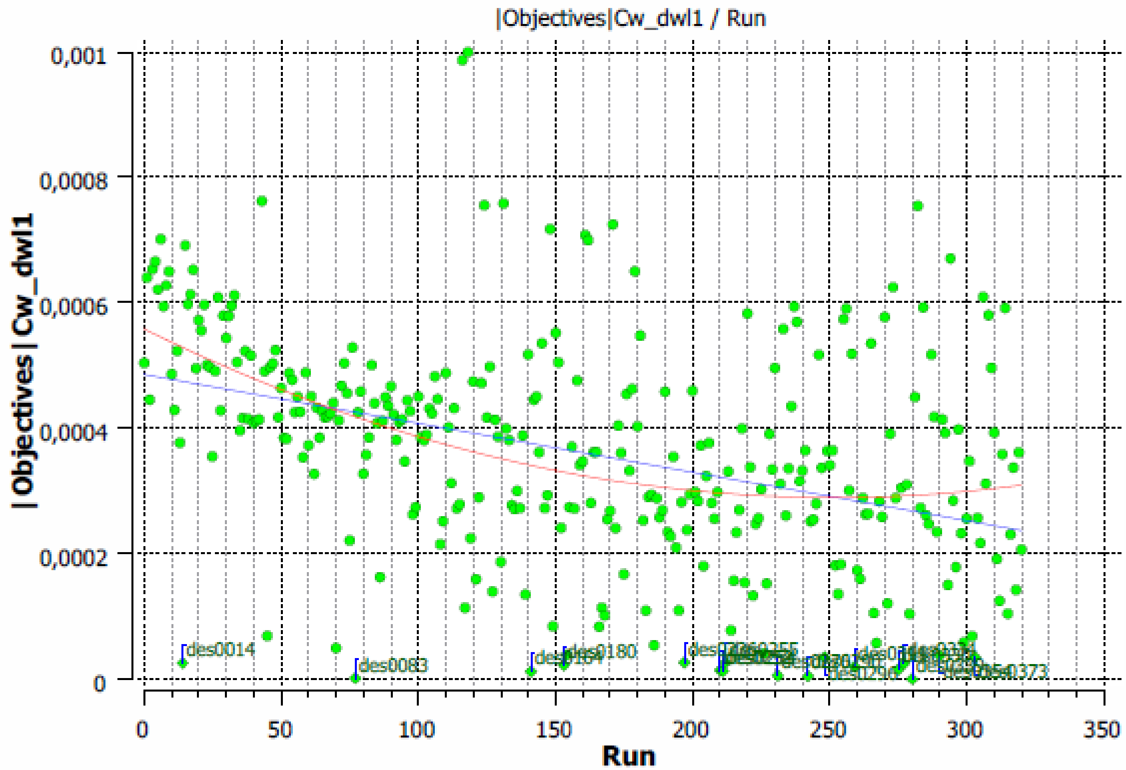


Figure 44: Cw_dwl1 objective during the search process

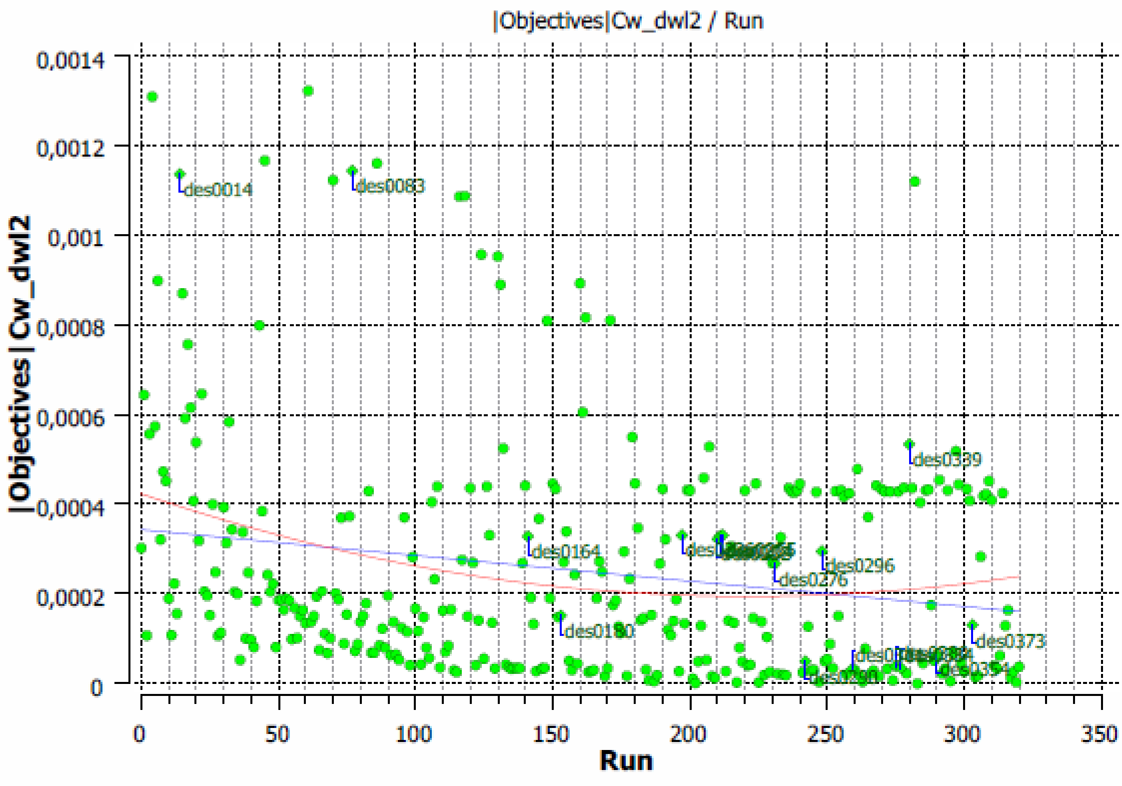


Figure 45: Cw_dwl2 during the search process

11.7.4 Time

Because the CFD method is a nonlinear one, the simulation time would be different from design to design. On simulation could be from 1 minute to 10 minutes depending on how fast the solution converges.

I used a Desktop Server Computer with a CPU at 3.33 GHz (2 processors) and an installed memory of 32 GB. The operating system was Windows 7 64-bit. The Computer divides the processor speed in 24 sections which I was allowed to use six of.

The first simulation took around 6 hours.

The second simulation took around 15 hours.

The third simulation took around 50 hours.

11.7.5 Discussion of the hull

Because of lack of time and that it is not the main goal of this thesis to try to find an optimized hull I will just present the hull how it will look like after the conclusions from the three simulated runs. It represents just a step on the way. More investigation should be done.

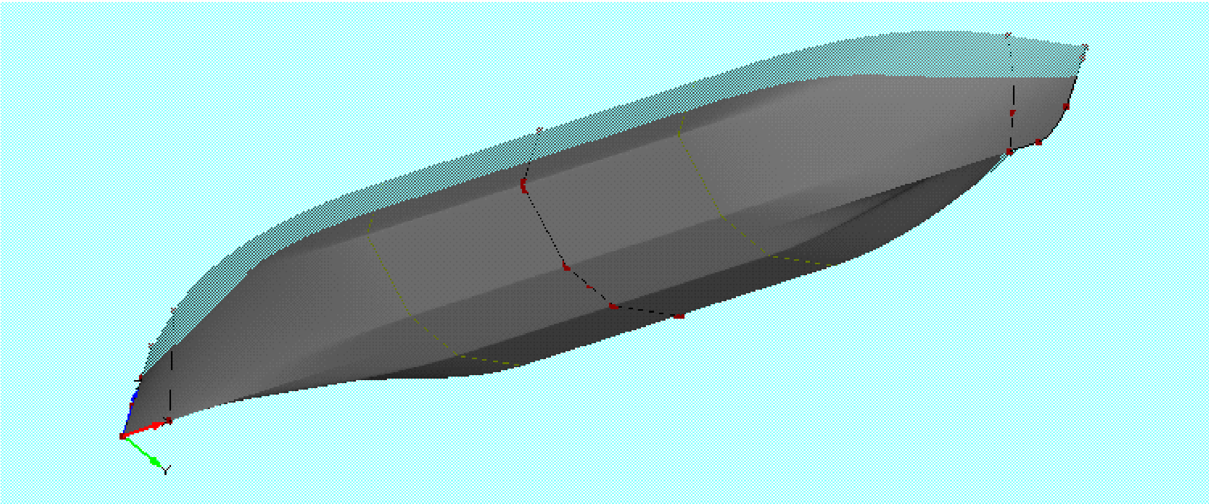


Figure 46: A good solution of the hull when looking at two operating draughts

$$R_{T_{dwl1}} = (C_{f_{dwl1}} + C_{w_{dwl1}}) \cdot \frac{1}{2} \cdot \rho \cdot V^2 \cdot (S_{dwl1} \cdot Lpp^2)\ [kN] \tag{11.27}$$

$$R_{T_{dwl1}} = (0.001593 + 0.0003849) \cdot \frac{1}{2} \cdot 1.025 \cdot (14 \cdot 0.51444)^2 \cdot 0.1987 \cdot 180^2 \tag{11.28}$$

$$R_{T_{dwl1}} = 338.506\ [kN] \tag{11.29}$$

$$R_{T_{dwl2}} = (C_{f_{dwl2}} + C_{w_{dwl2}}) \cdot \frac{1}{2} \cdot \rho \cdot V^2 \cdot (S_{dwl2} \cdot Lpp^2)\ [kN] \tag{11.30}$$

$$R_{T_{dwl2}} = (0.001567 + 0.00003153) \cdot \frac{1}{2} \cdot 1.025 \cdot (14 \cdot 0.51444)^2 \cdot 0.1269 \cdot 180^2 \tag{11.31}$$

$$R_{T_{dwl2}} = 174.722\ [kN] \tag{11.32}$$

In eq. 11.27 and 11.30 which represent the resistance in the two operating draughts I have left out the form factor to simplify things. It seems that frictional resistance C_f is more important than wave resistance C_w in a speed of 14 [knots] which is the case of this hull. However because of less wetted area there is a great difference in resistance estimated between the two operating draughts.

If this were to be an optimized hull with respect to the two operating draught it would be a hydrodynamic point of view of how the ship will look like in an early design process.

In a point based approach one would before starting this process already decide for many of the variables. The hull that is created here is therefore not representative in a point based design approach. If designing a bulk ship for example, the economy of the ship would have decided some of the hull shape variables already like much of how the block coefficient would be (much higher than with this hull) and the speed of the ship. This is because it is an iterative approach and the hydrodynamic estimations would have been later in the process.

But let's say that you were to design a bulk ship with a displacement of 20 000 [m3] and with two operating draughts of 10 [m] and 5 [m], this hull would be a starting point if favoring the hydrodynamic performances in a set based design approach. The decisions to making the ship shorter and with a higher block coefficient would be later in the process where more information of the problem is gathered.

11.7.6 Discussion of the process

The thing about optimization search processes is that it takes time to model. More experience will lower the time it takes, but still it takes time. However when running the optimization process you don't need to do anything. The process runs itself over one night or two days. Other work can then be done when running the optimization process or it can be run over the night from one working day to the other.

Make your model easy with as few free variables as possible. More variables will give the model too much complexity, and the time spent modeling and simulating will give low value as described in Section 9.1. Start out with investigating only one geometric change to identify how it will act on the objective(s). I got more knowledge from the first hull model than the second hull model. Fewer thoroughly though of variables will give more information to the designer than many variables. Many variables will give new questions and there is sometimes difficult to see how the interactions are dependent on the objectives. Also many variables will often make it difficult to identify errors in the results and where it origins from.

CFD coupled up to a CAD optimization program is a powerful tool to use. It can generate much knowledge to the problem, and also allows more innovation. The knowledge can be used in making better decisions and the innovation can be done because of the low cost of creating many simulations.

12 CONCLUSION

CFD have now arrived to the point where it is not used only as a modeling verification of the estimated performance before model tests, but is used to search for possible designs; simulation driven design.

It is clear that in ship design much of the decisions are based on experience. After discussing the three factors containing general product design, time and knowledge came out as important. You have little time and little knowledge to evolve the product design. It seems that the time factor can't be done much with. The market demands faster and faster ship design processes. However the knowledge can be done something with by using better tools like CFD. During my investigation I have identified three ways of how CFD could be applied in design. The two first represent design exploration and the last is the opposite of simulation driven design where CFD is used to verify the modeling.

Three ways of applying CFD:

1. Simulation driven design
2. Response surface design
3. Modeling design

The first method is used with optimization to explore different variation of shapes. It could be done with potential codes or RANSE codes. To use the time in the best way potential codes is the favored choice if doing many simulations. RANSE codes are used when turbulent areas are to be investigated, but because of time big steps in the variation should be done. In conceptual design this method is a powerful tool if there is time for design exploration. In ship design much of how the global parameters affect the hydrodynamic performance are already known. However when looking at robust hull shapes not much knowledge has been gathered and this method would fit well into a robust hull exploration. The research examples of this thesis have tried to show some kind of way it could be done, but is not a complete exploration.

The second method is used to find design space for certain objectives put up by the designer. Resistance, seakeeping and maneuvering response surfaces could be made by changing global parameters. The work that is laid down in creating these surfaces should be used later in future design processes or during a design processes like set based design. In set based design the parameters have ranges when designing. Response surface will fit well into such a design approach.

The third method is used to verify the model that has been developed during the design process. Typically the best CFD method is used to find the performance. Earlier in the design process easy empirical methods have been used to estimate the performances. This is more the old way of using CFD. It fits well into the point based approach of designing ships.

CFD is a computer tool and there should be some care in how to use the tool. Many of the researches I have read trough have used optimization and CFD to do the whole design exploration with many variables and the complexity is high. Lost information during the process is inevitable. CFD should be used as a supporting tool to gain knowledge before doing trade-off decisions and not ways of letting the computer find the final solution in the design space. It is in the trade-off decisions that a designer drives the solution to a good one. To have a good human computer synergy will give a better background for making better decisions. The decision should be based on the experience from the designer *and* the knowledge that is gathered during a CFD model exploration. In CFD exploration simple models that are; easy to understand, have a high CAD model complexity and a high level of parameterization will give good control of the process.

In short CFD should be used differently according to what that is designed and what method that is used. If designing new innovative hulls it should be used earlier in the design process and with a simulation driven design approach. If designing a more standard ship, CFD should be used in a modeling design approach to verify the estimations that have been done earlier in the process.

12.1 Further work

Two points of further work could be of interest:

1. Set based ship design
2. Robust ship design

The set based design approach has been thoroughly investigated, but how it could be applied in ship design has not been investigated much. I think that because of tools like CFD and other computer tools, set based design have been more feasible in naval architecture. I have only touched the subject and there seems to be advantages especially in new developing designs. I suggest if this is to be investigated it should be in collaboration with a ship design company or several companies. This is to investigate how the process is done now a day and make a comparison. A good start would be to read up on the literature; (Bernstein, 1998) and (Sobek, 1997). Also an interesting topic is to investigate further how CFD could be used to produce response surface and what response surfaces that will be of most interest in a conceptual design stage.

Robust ship design has barely been touched in research. There are many ways of approaching this but I will suggest two ways of further work. The first is to investigate how Taguchi's method or other probabilistic methods could be applied in ship designs. The focus could be to design a robust hull shape and not as a complete robust system. The second way could be to narrow it more down to an example of robust bulk ship and to design it for more than one speed and more operating draughts. This could be done in collaboration with a shipping company that operates bulk ships to see how the operating profiles may look like. Take an example of existing bulk ship and try to investigate the robustness of it. Try to investigate a new hull shape that is optimized and compare it.

The program that I have been using in this thesis will fit well into calculating the performances in both of the suggested further work.

13 BIBLIOGRAPHY

Abt, C., & Harries, S. (2007). A new approach to integration of CAD and CFD for naval architects. *International conference on computer applications and information technology in maritime insustries.*

Abt, C., Bade, S., Birk, L., & Harries, S. (2001). *Parametric hull form design - A step towards one week ship design.* Elsevier.

Anderson, D. (1997). *Agile product development for mass customization.* Irwin Professional Publishing.

Andrews, D. (1997). A comprehensive methododology for the design of ships (and other complex systems). *Proceedings:Mathematical, Physical and Engineering Sciences, Vol. 454, No.1968.* The Royal Society.

Andrews, D., Keane, R., Lamb, T., Sen, P., & Vassalos, D. (2006). Design methodology. *interantional marine design conference.*

ASI. (n.d.). *The American Supplier Institute (ASI).* Retrieved from http://www.amsup.com/robust_design/index.htm

Bernstein, J. (1998). *Design methods in the aerospce industry: Looking for evidence of set_based practices.* Master of Science Thesis: MIT.

Bertram, V. (2000). *Practical ship hydro dynamics.*

Bertram, V. (2002). *Practical Ship Hydrosynamics.* Butterworth Heinemann.

Bole, M., & Forrest, C. (2005). Early stage integrated parametric ship design. *ICCAS.*

Brenner, M. (2008). *Integration of CAD and CFD for the hydrodynamic design of appendages in viscous flow.* TU Berlin.

Brinati, H. A., & de Conti, M. (2007). Learning aspects of procedures for ship conceptual design based on first principles. *International conference on engineering education.*

Campana, E., Peri, D., Tahara, Y., & Stern, F. (2006). Shape optimization in ship hydrodynamics using computational fluid dynamics. *Elsevier*.

CFD Technologies. (n.d.). *CFD technologies*. Retrieved from http://www.cfd-technologies.co.uk/

Choi, J., Hsiao, C., & Chahine, G. (2005). Design trade-off analysis for high performance ship. Hull with air plenums. *International symposium on seawater drag reduction.*

Choi, J., Min, K.-S., Kim, J., Lee, S., & Seo, H. (2010). Resistance and propulsion characteristics of various commercial ships based on CFD results. *Elsevier*.

Clue. (2010). *Clue for windows version 8.4.* Clue international corporation.

Couser, P. (2002). Use of computers in the design of high-speed craft. *High-speed craft technology and operation.* RINA.

Couser, P., Harries, S., & Tillig, F. (2011). *Numerical hull series for calm water and sea-keeping.*

Definitions online. (n.d.). *Definitions*. Retrieved from http://www.definitions.net/definition/Parameter

Deremaux, Y., Willcox, K., & Haimes, R. (2003). *Physically-based, real-time visualization and constraint analysis in multidisciplinary design optimization.* American institute of aeronautics and astronautics.

Diez, M., & Peri, D. (2010). Robust optimization for ship conceptual design. *Elsevier*.

Duffy, Harries, Marzi, Petz, & Wu. (2009). Virtue: Integrating CFD in ship design.

Erfani, T., & Utyuzhnikov, S. (2010). Directed Search Domain: A Method for Even Generation of Pareto Frontier in Multiobjective Optimization. *Journal of Engineering Optimization*.

Evans, J. (1959). Basic design concepts. *Naval architects journal*.

Feng, B., Liu, Z., Zhan, C., Chang, H., & Cheng, X. (2009). *Ship hull automatic optimization techniques research based on CFD.* Elsevier.

Flowtech. (n.d.). *Flowtech*. Retrieved from http://www.flowtech.se/index.php?option=com_content&task=view&id=16&Itemid=16

Flowtech int. (2010). SHIPFLOW 4.4 Users manual.

FRIENDSHIP SYSTEMS. (2009). User Guide FRIENDSHIP-Framework.

Grigoropoulos, G., & Chalkias, D. (2009). Hull-form optimization in calm and rough water. *Elsevier*.

Hannapel, S., & Vlahopoulos, N. (2010). *Introducing uncertainty in multidiscipline ship design.* American Society of Naval Engineers.

Hardin, G. (1968). Tragedy of the commons. *Science*.

Harries, S. (2008). Serious Play in Ship Design. *Tradition and Future of ship design in Berlin*.

Harries, S., & Abt, C. (2008). Tight and loose coupling of CAD and CFD. *RINA Marine CFD.*

Harries, S., Tillig, F., Wilken, M., & Zaraphonitis, G. (2011). An integrated approach for simulation in the early ship design of a tanker.

Kolev, P., Damyanliev, T., & Georgiev, P. (2005). *Optimization and robust investigation in ship design.* Taylor & Francis Group plc.

Lackenby, H. (1950). On the systematic geometrical variation of ship forms. *RINA Transactions*.

Larsson, L., & Baba, E. (1996). *Ship resistance and flow computations, advances in marine hydrodynamics.*

Moctar, B. e. (2008). *State-of-the-art CFD simulations for ship design.* Springer proceedings in Physics.

Nowacki, H. (2010). Developments in marin design methodology: Roots, results and future trends. *IMDC.* Trondhem.

Palluch, J. (2011). Coupling of CAD, simulation and optimization. *The integration of CFD into the product development process.* NAFEMS.

Papanikolaou, A. (2009). Holistic ship design optimization. *Elsevier*.

Percival, S., Hendrix, D., & Noblesse, F. (2001). *Hydrodynamic optimization of hull forms.* Elsevier.

Ray, T., Gokarn, R., & Sha, O. (1994). A global optimization model for ship design. *Elsevier*.

Reinertsen, D. (1997). Managing the design factory: A product developer's toolkit. *New York: The free press*.

Rubbert, P. (1999). On the persuit of value for CFD. In M. H. D.A. Caughey, *Frontiers of computational fluid dynamics.* World Scientific publishing.

Sariöz, E. (2006). An optimization approach for fairing of ship hull forms. *Elsevier*.

Schneekluth, H., & Bertram, V. (1998). *Ship design efficiency and economy.* Butterworth Heinemann.

Singer, D., Doerry, N., & Buckley, M. (2008). *What is set-based design?*

Slater, J. (2011). *Uncertainty and error in CFD simulations.* Retrieved from NPARC Alliance CFD verification and Validation web site: http://www.grc.nasa.gov/www/wind/valid/tutorial/errors.html

Sobek, D. (1997). *Principles that shape product development systems: A Toyota - Chrysler comparison.* University of Michigan.

Steen, S. (2007). *kompendium: Motstand og Propulsjon, Propeller og Foilteori.* NTNU.

Steen, S. (2011, Juni 01). CFD. (P. Vangbo, Interviewer)

STX OSV. (n.d.). Retrieved from http://www.stxosv.com

Taguchi, G. (1993). *Taguchi on Robust Technology Development: Bringing quality Engineering upstream.* New York: ASME Press.

Ulstein Group. (n.d.). *Ulstein group*. Retrieved from http://www.ulsteingroup.com

Vangbo, P. (2010). *Hydrodynamic ship hull optimization.* NTNU.

Venkataraman, S., & Haftka, R. (2002). Structural optimization: What has Moore's Law done for us? *Structural dynamics and materials conference.*

Wang, W., Wu, J., & Lust, R. (1997). *Deterministic Design, Reliability-Based Design and Robust Design.*

Wikipedia. (n.d.). *Wikipedia*. Retrieved from http://en.wikipedia.org/wiki/Taguchi_methods

Xu, L., & Wang, Y. (2001). The fine optimization of ship hull lines in resistance performance by using CFD approach. *Elsevier*.

Zhang, B., & Kun, J. M. (2008). The optimization of the hull form with the minimum wave making resistance based on rankine source method. *Elsevier*.

Zhang, P., Zhu, D.-X., & Leng, W.-H. (2008). parametric approach to design of hull forms. *Elsevier*.