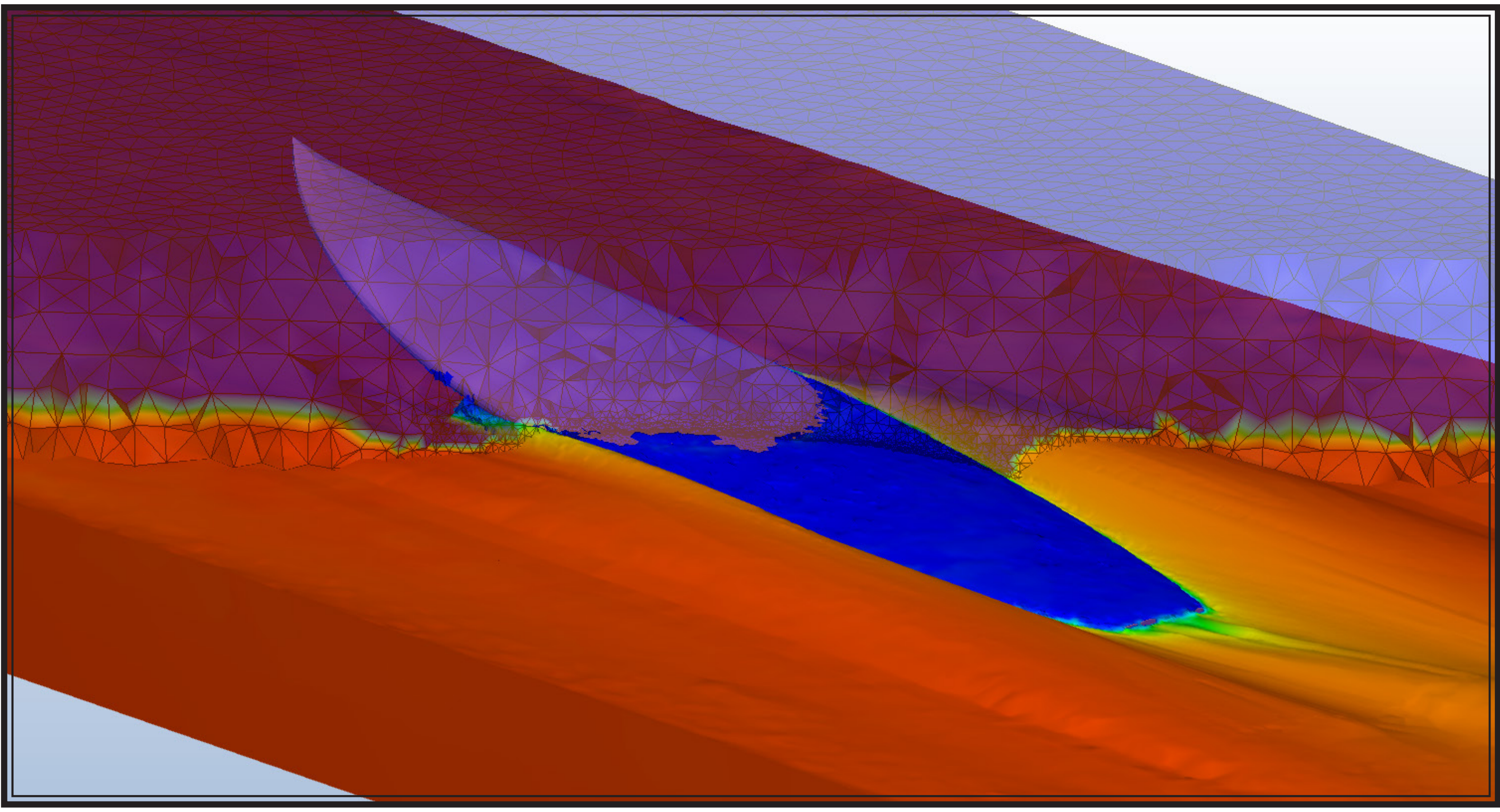
UNIVERSITY OF TWENTE.
Faculty of Engineering Technology
Internship project for Master Mechanical Engineering
Thermal and Fluid Engineering

February 2, 2018
Tweed Heads, NSW, Australia



Validating CFD surfboard models to experimental results

As part of the Peau Pty. Ltd. Surfboard Performance Analyser Project

Author:
Jason Duesmann
student nr.: s1361201

Supervisors:
Adj. Assoc. Prof. Steven Schmied
Peau Proprietary Limited
Prof. dr. ir. H.W.M. (Harry) Hoeijmakers
University of Twente

Preface

This internship is done as part of the University of Twente master program Mechanical Engineering, specialization Thermal and Fluid Engineering. The activities are performed from early November 2017 until early February 2018 for the Australian based company Peau Proprietary Limited and conducted at the Aku Shaper factory in Tweed Heads, New South Wales, Australia.

Professor H.W.M. Hoeijmakers is the supervisor from University of Twente and he assesses this report. Adjunct Associate Professor Steven Schmied is the external supervisor from Peau Pty Ltd.

It is assumed that the reader of this report has a basic understanding of computational fluid dynamics.

All intellectual property generated by the student during and regarding the assignment shall be owned by Peau Proprietary Limited. The report remains confidential for a period of twelve months after submission to the supervisors.

This report contains both an abstract and a summary. The abstract was required for the university, but the summary gives a bit more information.

The author would like to apologize in advance for any remaining incorrect grammar and spelling or strange sentence structures still present, unnoticed due to English not being his native language.

Abstract

As an important step in developing a surfboard performance analyser tool, this internship tried to validate a surfboard CFD model that could be used for a parametric analysis. Simulations were to be compared to towing tank experiments performed in 2011 in which a surfboard in a fixed orientation is moving through water at different velocities. An initial CFD surfboard model was developed in Autodesk CFD, which was improved by investigating simulation results in sub-studies. These helped to draw conclusions on the to be used proper domain, board mesh density, wall layer thickness, turbulence model and advection scheme. Surface roughness was found to be of influence, but could not be applied confidently. Mesh independence was not reached due to limited computing power. This - and incorrect wall layer thickness and transition - were thought to be the main reasons for having up to 60% lower drag in the simulations than in the experiments. To improve this, it was recommended to investigate how to correctly implement surface roughness and also improve the mesh by adjusting the board CAD model and using more computing power to solve for smaller elements. To make the model more suitable for a parametric analysis and a more realistic surfing situation, modeling waves and motion driven flow might have to be implemented.

Summary

A surfboard performance analyser tool was desired by the employer, eventually to be implemented in surfboard design software. This tool must assist board designers and shapers to choose between the numerous alternatives to better fit their customers' needs for maximal performance, progression or fun. An important step in the development of the tool is finding empirical relations for suitability of combinations of board geometry, wave properties and surfer experience from a parametric hydrodynamic analysis using computational fluid dynamics (CFD). At first a CFD model had to be developed and validated. This is done during this internship by trying to match simulated to experimental data.

Experimental data was available from a previously conducted study under the same supervisor. Three similar boards, only differing in tail shape, were in a fixed orientation towed in a straight line at four different velocities through a towing tank at the Australian Maritime College (AMC) in Tasmania. Drag forces were measured and pictures were taken of the wake directly behind the board. Drag was found to increase almost linearly with velocity. The differences in tail shape did not result in significant drag differences.

Research questions were proposed. Answering these must give insight in how closely the model was matching the experiments and what had to be taken into account and improved for a future parameter study. An approach was suggested that should form the pathway for developing the CFD model and answering these questions.
Firstly, the proper setup and settings had to be found that gave realistic results. To find these, the experiments were mimicked as far as possible. Also, literature was consulted, test simulations were performed and support through a forum was asked.
Secondly, the proper setup and settings should be used to simulate the experimental runs.

An initial model was developed that looked a lot like the experimental situation, but of which results were still different.
Of two boards the geometry was digitally available in board design software AKU shaper. The file format had to be converted and the mesh density had to be decreased in order to import the geometries into the software Autodesk CFD. The board submergence was estimated from photographs, since this was not clearly documented for the experiments. The computational domain was similar to the towing tank size, except in the movement direction. A lot of information was found online about recommended settings for this kind of analysis. Drag forces were monitored to judge if simulations were converged.

Several sub-studies were performed in order to find out how the model could be improved. Some sub-studies were run in parallel and therefore not always the latest settings were used in every simulation.
The domain size was decreased and new boundary conditions were applied, since the old domain was unnecessarily big and boundary conditions could not be applied realistically.
It was tested what the influence of different submergence length was on the drag, because there was uncertainty about the exact submergence length. No useful results were found, probably due to a poor mesh and other incorrect settings.
The influence of the board mesh density was investigated. It turned out drag forces did not differ noticeably. Slightly different wake structures were seen for the higher board mesh densities. A board with 4000 faces was thought to be a good compromise.
Since surface roughness is of influence on the skin friction drag, it was investigated how results would differ in case surface roughness was applied to the surfboard surface. It was seen that drag forces were much lower when there was surface roughness, so the difference with the experiments was even bigger. No drag-roughness relation could be determined. Uncertainty about correctly implementing surface roughness in combination with the turbulence model remained.
Several combinations of turbulence models and advection schemes were tested, because some literature recommendations were conflicting. The SST k-omega SAS turbulence model with the ADV5 advection scheme and 10 wall layers was preferred because it gave best converging results, however at the cost of longer computation time. But drag still was 61N lower than the experimental 104N.
A mesh independence study was performed with five meshes, each finer mesh having half the element size nearby the board than the coarser mesh. The finest mesh had 16 million elements and with several weeks computation time, this was the limit of the available time and computer power. Drag was not converged for smaller element size, so no mesh independence was reached. Also, some badly meshed areas were still seen but could not be improved in Autodesk CFD.
Drag forces were still over 30% off, but possible improvements - except for the mesh - were becoming more limited. To find out if everything so far was setup correctly and the differing drag was indeed due to the poor mesh, support from Autodesk CFD specialists was asked through their forum. Their conclusion was that the problem indeed was in the meshing. Besides decreasing the element size, it was also recommended to make the board surface meshing more uniform and rectangular in order to have better prismatic wall layers and transition to the tetrahedral elements. It was tried to improve the board Computer Aided Design (CAD) model surface meshing, but without success.
To capture the boundary layer and therefore drag correctly, the prismatic wall layer thickness is of great importance for the used turbulence model. It was investigated what the first layer element height had to be and it turned out this needed to be about 40 times smaller than in the often used mesh MN. It was not known how to apply such a thin layer in Autodesk CFD and simulate this with the current computer power.

Although drag was still found to differ a lot from the experiments, it was hoped maybe a similar drag-velocity relation could be found. The final runs were trying to mimic the experiments as good as possible

with the available knowledge and resources. Domain D20 was used with suppressed board S00 and mesh MNb. The SST k-omega SAS turbulence model, AVD5 advection scheme and advanced wall layer settings were used. 2000 time step size of 0.005s with 10 inner iterations were run.
Drag was found to increase with velocity, but in a different way than in the experiments. The simulated wakes did not join as was seen on the taken photographs during the towing tank tests.

Because no matching results were obtained, no conclusion could be drawn on how the CFD model must exactly be setup. Although many things were applied correctly, a few things still need improvement. Based on the results from the sub-studies, it is likely that problems are caused by improper meshing. Elements are probably too coarse, there is a non-smooth transition from the prismatic wall layer to the tetrahedral elements and the wall layers are too thick. To resolve these problems, the surfboard CAD model needs to be adjusted and more computer power is needed. Also, it was unsure how surface roughness had to be correctly modeled.

A lot of sub-studies were performed with unsuitable settings and improper meshes. Care had been taken when drawing conclusions from these results. It must also be noticed that the submergence length might have caused differences, just like any other possibly incorrectly interpreted methods used in the experiments. Besides, it was not guaranteed that the digital board files accurately represent the actual boards used.

For future investigations in the Aku Shaper Performance Analyser project it is recommended to improve the mesh and wall layers, using more computing power. Adjusting the board CAD model might also be necessary for achieving this better mesh. Besides, surface roughness must correctly be applied. Above that, a motion driven model might be developed to use for the parameter study, as well as implementing fins and modeling waves in order to come closer to a realistic surfing situation. Other software, capable of exporting list results, would be useful for determining empirical relations. Possibilities to discuss CFD results in person with colleagues would increase progress and avoid mistakes.

Contents

1 Introduction

1.1 Motivation

Surfboards can be found in many different designs, suitable for different types of surfers and waves. The latest designs are a result of decades of trial and error, in which the surfboard shaper receives feedback from the surfer. The shaper gains experience over the years and applies this knowledge in order to improve the performance of the boards, or to match a board with his customers' wishes.
In the past years the surfboard industry also discovered the use of computers to aid them in the design process. Boards can be designed digitally and can even be cut out of a blank by a specialized Computer Numerical Control (CNC) machine. One such a software program to design boards is Aku Shaper, made by a company that also produces cutting machines. It should be noted that after the shape is cut out, still some hand work like sanding and glassing is needed.
Even though the shaper has experience on what the effects on the performance shall be when he or she makes certain adaptations, it would be a great advantage if the software could hand the shaper feedback on the current design. This saves time, material and therefore costs for producing a board and interpreting qualitative feedback from the customer. In an even better scenario, the software would not only give feedback on the design, but also give suggestions on how to improve it. This would take out the guess-work and assist the surfboard shaper by designing a board that fully matches his customers' needs. This may make it possible to design even higher performance boards for professional surfers, while on the other end of the spectrum more forgiving boards can be made for beginners. Supervisor Steven Schmied and I are both surfers and applied our fluid engineering knowledge to get one step further in developing this software tool by setting up a Computational Fluid Dynamics (CFD) surfboard model. By taking out the guess-work and replacing it by research based knowledge we hope to let the surfboard innovation thrive, with more fun for more people. Findings of this project may later even be applicable outside of the surfboard industry as well, possibly other planing hull hydrodynamics such as high speed sailing.

1.2 Report structure

Now that the motivation for this project has been explained, this report starts by defining goals to be achieved during this internship (chapter 2). Some general information about surfing is included in chapter 3. Also, the components of hydrodynamic drag experienced by a surfboard are explained, because drag is a much monitored quantity throughout this study. Subsequently, a summary is given about the experiments (chapter 4) of which the results are tried to be matched by CFD simulations. After that, in chapter 5 it is time to define research questions and suggest an approach to find answers. The approach starts by setting up an initial model (chapter 6) that could subsequently be improved in chapter 7 by doing various sub-studies. These findings could then be used to perform the same runs as in the experiments, so that results can be compared in chapter 8. Based on this and the findings in the sub-studies, conclusions can be drawn (chapter 9), certain issues are to be discussed (chapter 10) and recommendations are given (chapter 11). Last but definitely not least, a lot of thanks are given in the Acknowledgements (chapter 12).
The process of this internship project shall be evaluated in appendix A.
Instructions are made for a future student that may be working on this same project, in appendix D.

2 Goal

The final goal of the entire project shall be to have a validated tool implemented in the Aku Shaper software (the *Aku Shaper Surfboard Performance Analyser*) that returns the suitability of a surfboard design for a range of surfer experience and waves, based on empirical relations. These relations are to be determined by a parametric analysis using CFD simulations. This internship shall form the start of the CFD simulations. The scope of work for the multi-year Aku Shaper Analyser project was originally described as follows:

1. Conducting a parametric CFD analysis of surfboards. Parameters including surfboard and fin geometry and material.
2. Determining empirical relations between the surfboard parameters and performance, to define suitability for certain surfers (weight, height, ability) and waves (height, shape).
3. Create a tool to analyze a surfboard design file and return the suitability for a range of surfers and waves.
4. Develop a method to integrate the tool in Aku Shaper.
5. Validate the tool against existing knowledge or tests.

Before the actual start of the internship was decided to try to get as far as possible with steps 1 and 2. It soon became clear however, that validation of a CFD model was needed even before a parameter study could be conducted. In one of the first weeks it was decided that the major goal of this internship besides the educational goals for me - was to deliver a validated CFD model that could be used for the parametric analysis.
A CFD tow tank model shall be set up for which the results could be validated against surfboard tow tank experiments conducted in 2011 at the Australian Maritime College (AMC) by student Sam Schumacher under the same supervisor (Schmied) [29]. The setup and settings of the model shall be well documented to guarantee its future use. This report must give insight in how good the model represents reality and what the limitations are.
It should be noted that the scenario during the tow tank experiments is still totally different than a real surfing scenario, as shall be mentioned in 3.1. For the parametric analysis, a CFD model is needed that comes closer to this real life surfing scenario. Therefore it shall be recommended how the tow tank CFD model can be adapted such that it is more useful for the parametric analysis.
One should be aware that only the CFD tow tank model is actually going to be validated against experimental data. It should be assumed that the adapted model, with similar CFD setup and settings, is suitable for the parametric analysis.

3 Surfing

The focus of this internship turned out to be on CFD modeling, more than on surfboard hydrodynamics. For the future parameter study the focus shall presumably more be on the actual flow changes due to board geometry adjustments.
However, in order to understand some of the considerations made during this internship, necessary for the future parameter study, some basic surfing background is provided here for the non-surfing reader. Also a short overview is given about surfboard hydrodynamic drag, as drag will be the most important monitored value between the experiments and simulations.

3.1 Surfing in general

The idea
Surfing is a popular water sport in which the surfer balances on a surfboard, riding a water wave. The surfer is trying to stay on this wave as long as possible to enjoy the wonderful feeling, or to perform incredible tricks. By shifting its weight in a very controlled way, the surfer is able to make drastic or gentle turns and can ride perpendicular to the wave. This is necessary, because this is the same direction as in which the wave is peeling and the surfer wants to stay just in front of the broken wave (the white water).

The movement
Before the surfer is able to stand on the board, he or she is laying down on the board and paddles as fast as possible to catch up with the wave velocity. Once standing, the surfboard with surfer gains

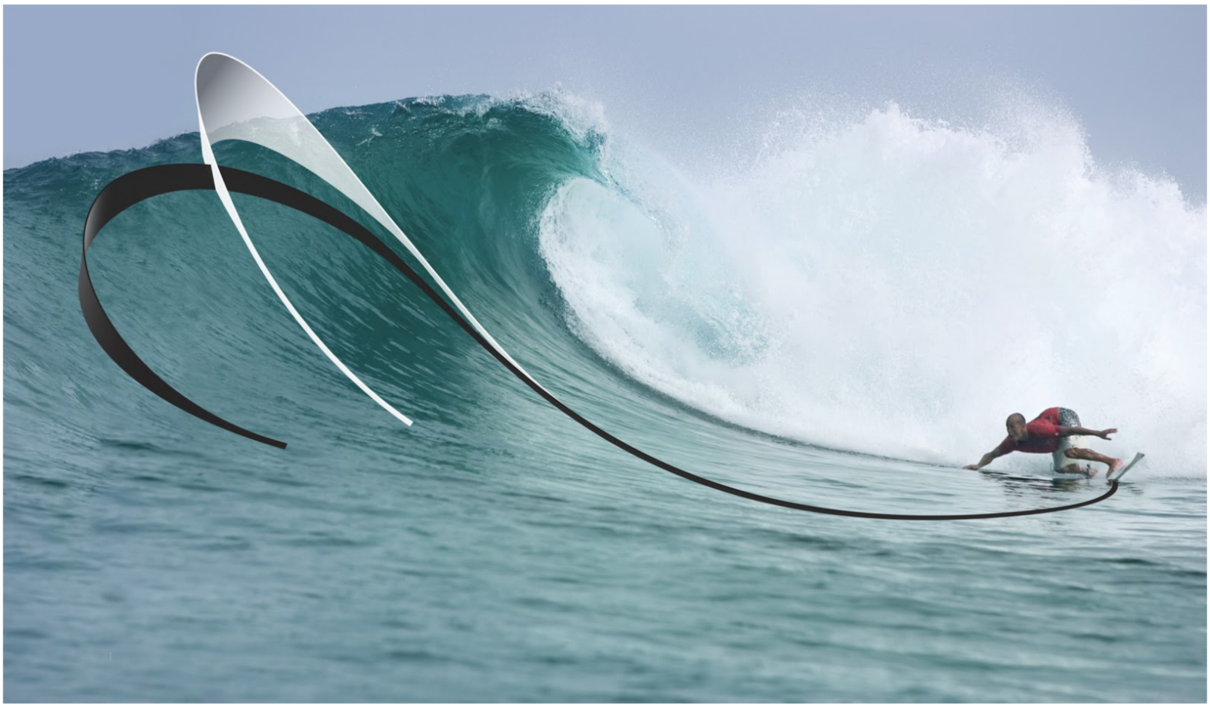


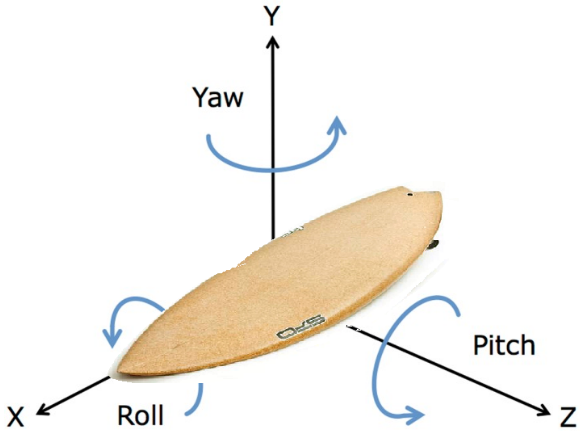
Figure 1: Surfer in a bottom turn. The shown planned route travels up the wave after the bottom turn and allows for a cut back turn at the wave crest.

speed due to gravity as it travels down the wave. The surfboard is partially submerged and makes an angle with the locally inclined free surface, called the trim angle. This trim angle determines the ratio between hydrodynamic lift and drag and needs to be well controlled by the surfer. The lift is necessary to support the surfers weight, because merely the buoyancy of the board is generally not enough, especially not when the board is planing and a only a small portion of the board is submerged. A too small tilt angle increases the chance of making an undesired nose dive, while a large trim angle comes with more drag. Large drag may slow down the board too much, so that the surfer shall be swallowed by the breaking wave. But when there is too less drag, the board gains speed and ends up on the horizontal free surface in front of the wave, losing its gravity potential. When the board still has a lot of speed, the surfer can choose to make a so called bottom turn (figure 1) and afterwards ride the wave upwards for a moment to convert its kinetic energy back into potential energy again. The turn is induced by the roll and yaw angles of the surfboard resulting from the surfers weight shift. See figure 2 for the angle definitions.

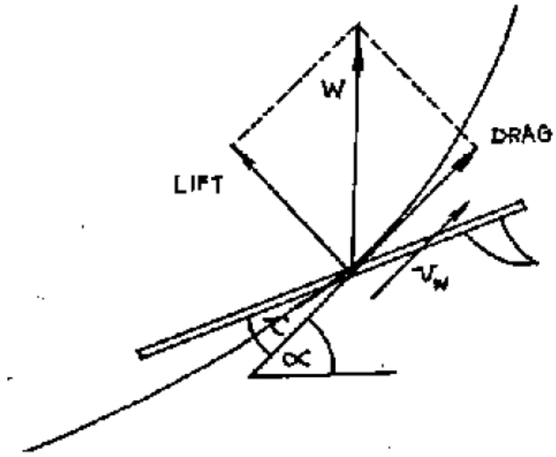
The wave
Although some interesting developments are made in artificially created surfable waves, most surfing is done on incoming ocean waves, breaking and peeling sideways once they reach shallower water nearby the shore. Waves vary in a lot of ways, such as their height, steepness and the way they break.

The board
The front of the board is called the nose and the back the tail. The sides are the rails, the top is called the deck. Fins are mounted on the bottom side nearby the tail and can be placed in a lot of different configurations. The amount of curvature from nose to tail is called rocker.
It can be understood that boards vary in many different ways (just see figure 3 as an example) and not only in shape, but also material and surface finish. Each design being suitable for a certain range of surfers or waves. Although this knowledge is primarily based on experience, rather than scientific research.

The simplification
In this project a partially submerged surfboard shall be moving with fixed speed at a fixed trim angle over flat water, without fins and without a weight applied. It should be realized this scenario is a great



(a) Pitch, yaw and roll.



(b) Side view of board on inclined free surface, making angle α with the horizontal. γ is the trim angle, $\gamma + \alpha$ the pitch. Image from [28].

Figure 2: Angles of a moving surfboard. The pitch angle is the angle between the board and the horizontal. The trim angle is the angle between the board and the inclined water surface, so that is the pitch minus the free surface inclination angle.

simplification of actual surfing. Extra complexities (such as water inclination, fins, applied forces) can be added step by step to the model to make it useful for the parametric analysis needed to make the Aku Shaper Performance Analyser.

3.2 Surfboard drag

A partially submerged surfboard experiences both hydrodynamic and aerodynamic drag, forces opposing its motion direction. The aerodynamic drag is usually much smaller than the hydrodynamic drag due to the huge density difference and is therefore not considered here. The drag experienced by the surfboard while moving through the (viscous) water, consists of two main components, namely viscous drag and wave-making drag [30].

The wake-making drag (and also spray-making drag) occurs for bodies moving at or near the interface of two fluids. A change in water pressure is induced due to the body's motion. The pressure variation in the water relative to the constant pressure above the water surface results in the formation of waves [30], extracting energy from the body. Since this type of drag is dependent on the pressure variations at the board's surface, it is dependent on the board shape.

The viscous drag is composed of both pressure (or form) drag and skin friction drag [30, 28].
Normal stresses due to water particles bumping onto the board surface create a pressure drag. This drag is of course also dependent on the shape [26].
Shear stresses caused by viscous and turbulent effects induce the skin friction drag. The transition from zero velocity at the board surface to flow velocity happens in a thin layer, the boundary layer. The type of flow - laminar or turbulent - in this layer is dependent on many things (flow velocity, length, fluid properties) and is of influence on the experienced friction. For flow over an infinitely long flat plate, the transition from laminar to turbulent happens at a certain Reynolds number, approximately $\mathrm{Re}_x \approx 5 * 10^5$ [3]. For surfboard motion, this transition often happens within a relatively short distance [26] (also see paragraph 7.8.2), so that most of the flow in the boundary layer is turbulent. Surface roughness can cause the shear stress in the boundary layer to increase, resulting in higher friction. Obviously, the total amount of skin friction is dependent on the wetted surface area.

Figure 3: Less conventional boards, just to show variations possible.

Figure 4: Perfect waves approaching the shore and breaking sideways, allowing for a long surf ride.

4 Experiment

In 2011 Sydney University mechanical engineering student Sam Schumacher, under supervision of Steven Schmied, marked the beginning of a full parametric analysis of surfboard hydrodynamics with his surfboard experiments [29] conducted in the towing tank [1] of the Australian Maritime College (AMC) in Launceston, Tasmania. Part of his results are usable to verify the CFD model. To understand what real life situation has to be mimicked by the simulations, a summary of the experiments - including essential information for the simulations - is given in this chapter.

4.1 Introduction

Only certain experiments of Schumacher's research are considered useful for the validation of the CFD model. These are the ones in which three surfboards - only differing in tail shape - were towed in a fixed orientation at four different speeds in the water at rest in the AMC towing tank. Resulting drag forces and wake patterns were recorded when a steady state situation was reached. It was questioned what the drag-speed relation would be, how this would differ between the tail shapes and when the boards were considered to be planing.

4.2 Method

4.2.1 Boards

The three boards used in the experiments and displayed in figure 5 are almost the same, except for their tail shape. No fins were used.
The length of the surfboard is in this study expressed as the length of the projection of the board when laying on the ground, i.e. the straight line length from tail to nose visible when viewing the board from the top or bottom. This length shall be called the Length of All (LOA) and is 1792mm for the used boards.

4.2.2 Fixation

The boards were fixed at a trim angle of 6 degrees, measured between the (horizontal) water free surface and the bottom line of the projection of the board on a side plane, underneath the center of mass of the board (see figure 6). No roll or yaw angles were applied.

The attachment points were fixed at 20% and 70%LOA measured from the tail, as seen in figure 5.

Figure 5: Three boards used in the experiments with wooden blocks fixed to deck for attachment.

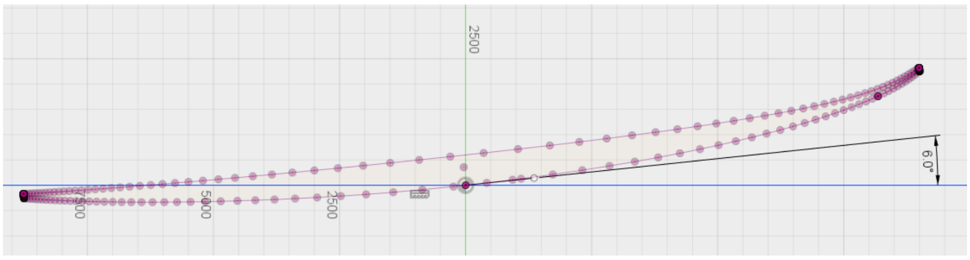


Figure 6: Measurement of the 6 degrees trim angle.

One way to measure the submergence of the board in the water, is by the length from the tail to the water surface at the deck of the board. It was hard to find this submergence, since this was not clearly documented.
There were however pictures taken from the setup with the board partly submerged (like figure 7). Knowing the length of the board and the positions of the attachment points there were three independent reference lengths for determining the submergence from the pictures.
Unfortunately, the light refraction from the water to the air made these reference measurements uncertain. Different pictures and different reference lengths resulted in values for the submerged surface length differing as much as 300mm.
Luckily, a sticker was placed on the top of the surfboard and the waterline crossed the top centerline nearby, as can be seen in figure 7. There were also pictures available of the board out of the water, with the sticker, see figure 5. This solved the problem with the light refraction. Using the three reference lengths, the values for the submergence differed only 14mm. The average value of 767mm was used for the simulations.
This value of the submergence means the waterline touches the top centerline at 37.7%LOA. Knowing that the weight in the experiments was applied at 30%LOA and the back side attachment at 20%LOA, the 37.7%LOA seems to correspond reasonably to the pictures taken, although it is hard to be accurate.

4.2.3 Velocities

The experiments were conducted at four different velocities, namely 3.0, 3.5, 4.0 and 4.5m/s. Due to some control inaccuracy, the actual velocity could be slightly different and was therefore measured for each run. The actual velocities are visible in table 1.

Figure 7: Top view of the attached partially submerged pin tailed board. The waterline crosses the deck centerline just left of the letter M on the sticker. The weight applied at 30%LOA did not play a part in the experiments described.

4.2.4 Measuring

Drag
Drag is defined as the force opposite the motion direction and was measured by a load cell. The output voltage was recorded over time and the unstable part (first and last few seconds) of each run were disregarded since only the steady state situation was to be measured. The load cell was calibrated, so that the voltage was converted into grams. For this project, Newtons are preferred as units for force. Therefore the gravitational acceleration constant in Tasmania $g = 9.804\text{m/s}^2$ [6] was used to convert grams to Newtons.
It was not mentioned what voltage of the stable interval was taken. It shall be assumed that an appropriate choice was made, for instance by averaging the voltage over the stable interval.
Two extra runs were performed for both 3.0m/s cases to get an estimate for the repeatability error. The repeatability error was taken to be 0.55N.
It was assumed that the measured drag was purely hydrodynamic, because aerodynamic drag is considered to be much smaller, i.e. $\rho_{air} \ll \rho_{water}$.

Wake
Photographs were taken from above the tail section of the board to examine the wake pattern. Comments were made about the position along the rail where the wake was generated from and how far behind the board the wakes from both sides joined.
No wave heights or lengths were recorded.

4.3 Results

Drag
Measured drag values are displayed in table 1. The number significance in table 1 might be too detailed, considering all possible factors causing errors (for instance the repeatability error). The numbers are however still displayed like this, but one must consider that they are not necessarily that precise.
Results are almost the same for pin and square tail. Drag increases with velocity, what was to be expected. The relation almost appears linear in this velocity regime, but the slope seems to decrease for higher velocities.

Wake
Photographs of the wakes can be seen in figure 9. It is hard to get full understanding of the wakes from these not very clear images. It is thought Sam Schumacher had more insight in this, probably using other views. Quoting his report on the wakes: *"For higher speeds on all boards, the wake was consolidated into one initial wake wave which joined roughly 30% LOA aft of the tail. The lower speeds saw a semi-planing condition where water would wash over the tail of the board..."* [29].

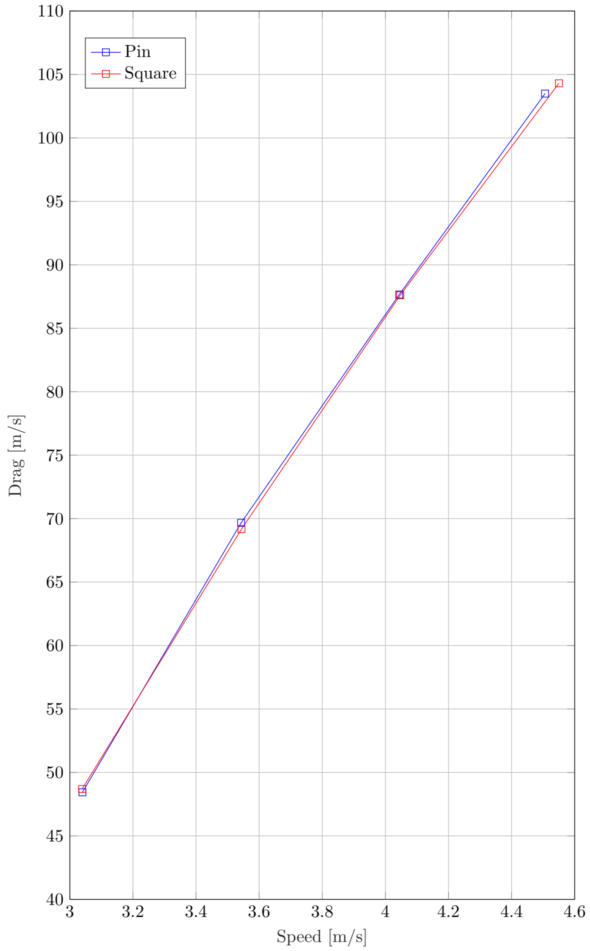


Figure 8: Experimental drag versus velocity for pin and square tailed board.

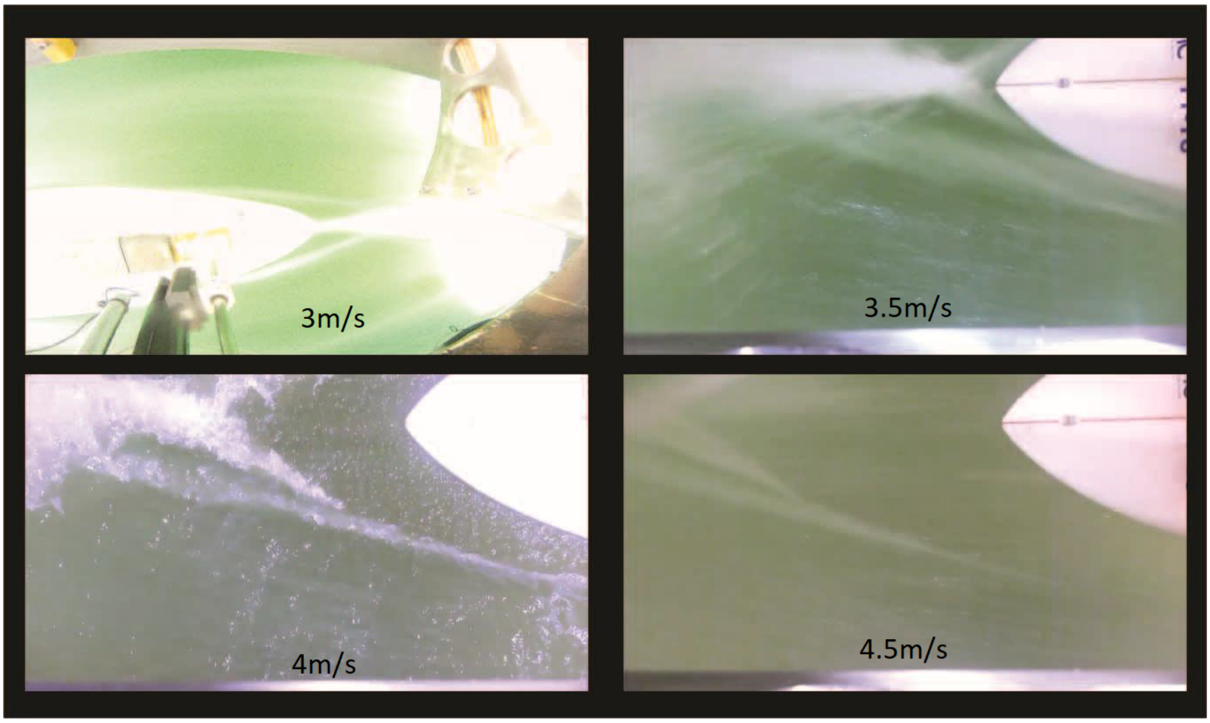
Pin		**Square**	
Velocity [m/s]	Drag [N]	Velocity [m/s]	Drag [N]
3.04	48.44	3.039	48.69
3.543	69.67	3.544	69.17
4.044	87.66	4.047	87.60
4.506	103.49	4.55	104.31

Table 1: Experimental drag results.

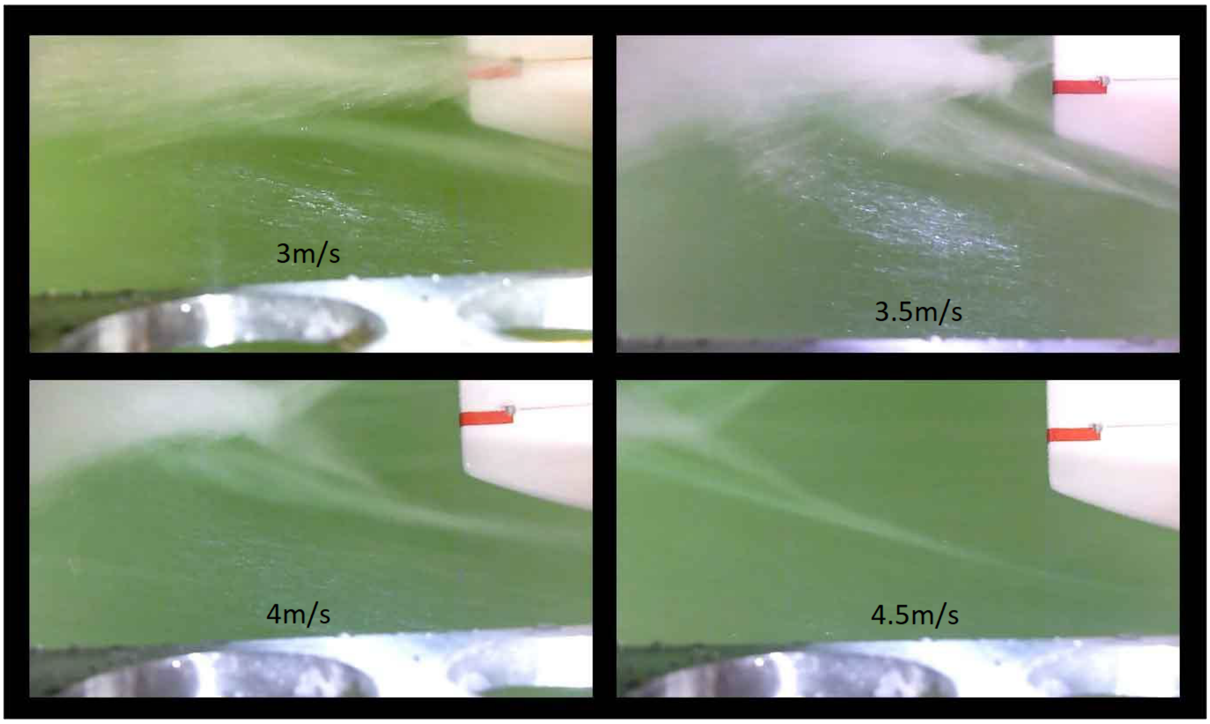
4.4 Conclusion and discussion

The conclusion of importance for this internship, is that drag increases with speed in an almost linear manner. Also, it was concluded the tail shape does not influence the drag significantly for these velocities. It is thought the tail shape influence increases only when roll angles are applied. Wakes generated from the rails join at about 30%LOA behind the tail for higher speeds. For lower speeds, water still washes over the tail.

It is important to consider the uncertainty in the results due to several factors. There always remains the possible human errors in setting up the experiments and doing measurements. Also, calibration errors of the measurement devices should be regarded. The first and last few seconds of each test were disregarded in order to only do measurements while in steady state, but it was not found how drag measurements over time were translated into one drag force. For these reasons, it is expected that matching these experimental results by simulations within a tolerance of a few percent would be acceptable.



(a) Pin tail



(b) Square tail

Figure 9: Photographs of wakes behind board taken during the experiments [29].

5 Problem Approach

In this chapter the goals mentioned in chapter 2 are translated into answerable research questions. An approach is outlined that should form the pathway to the answers. Also a CFD software program is chosen to use for the simulations.

5.1 Research questions

A list of questions is proposed of which the answers are to be found during this internship. The questions are related to the goals (chapter 2) of preparing a CFD model for a future parametric analysis.

- *How much do the simulated drag forces differ from the experimental drag forces?*
- *What is the relation between drag and speed for both tail shapes, and how does this differ from the experiments?*
- *What wake patterns appear during the simulations and what are the differences with the experiments?*
- *How must the CFD model be setup in future surfboard analyses to find realistic results?*
- *In case drag and wake measurements for the model would match the experiments within acceptable tolerance, what limitations of the model - besides limitations imposed by assumptions - need to be circumvented before the model is usable for the parameter study?*
- *What assumptions were made for setting up the model?*

5.2 Approach

As one can see, the research questions are related to a CFD model that simulates the experiments, preferably as good as possible. But the first question is: how is this CFD model going to be realized? This asks for a division of this study in two parts:

- Part 1 : Finding the proper CFD setup and solver settings that give realistic results.

- Part 2 : Actual validation. Simulating the same runs as in the experiment with this setups and these settings. Compare the results and answer the research questions.

The approach for part 2 is relatively straight forward, but for part 1 a plan is needed how to find the proper setup and settings.

To find realistic results with CFD software is not just a matter of letting the software solve the flow around a CAD geometry with default settings. Thought-out decisions must be made for the setup and the settings.
With the *setup* is meant both the CAD geometry of the model (including domain size, board meshing and orientation) and the setup in the CFD software (including material, boundary and initial conditions, meshing). With the *settings* is meant the settings of the CFD solver (advection and turbulence models, time steps, iterations, convergence assessment etc.). Three methods are used to find the suitable setup and settings:

- Method 1 Mimicking the experiments;

- Method 2 Literature recommendations;

- Method 3 Testing different settings or setups and judging the outcomes.

- Method 4 Asking for support through the Autodesk Forum

For certain settings and setups multiple methods are used to make a proper decision, which causes some previous suggestions to be updated. It should also be noticed that a lot of discoveries were made on the way, while trying to run simulations and receiving errors or unphysical results. Documenting all (failing)

runs and explanations for these minor necessary adjustments is considered undesired for this report for readability reasons.
During Part 1 it is important to keep track of assumptions and limitations that are made. They will be summed up when answering the related research questions in chapter 8.

In chapter 6 mainly method 1 and 2 shall be used to make the initial model as good as possible. In chapter 7 it shall be investigated by doing simulations (method 3) how the model can be improved. Also support from the Autodesk CFD forum is asked when the other methods do not provide solutions anymore. In chapter 8 the runs as in the experiments shall be ran with this improved model so that the research questions can be answered.

5.3 Software

Many different CFD software packages are available, all suited for a certain range of applications. In this project Autodesk CFD Motion 2018 (in this report referred to as just *Autodesk CFD*) shall be used. This program has a lot of functionality which is expected to be necessary, especially for future research in this project, such as free surface modeling, solid body motion and suitable turbulence models. Besides, their is good result visualization options. Above that, the software is free for students and educators.

For validation it is good to realize that even the best Autodesk CFD results may still differ a bit from reality (as for any CFD software), as Autodesk showed for several verification scenario's. The scenario closest related to this project is possibly the 'Drag Force on a Cylinder' [22], having a 3.688% error compared to the benchmark in the 2017 version (no data available for 2018 version).

The general work-flow of Autodesk CFD is to first setup the model, let it solve with correct settings and then assess the results. To define the setup, a geometry is imported and materials can be assigned. Subsequently, boundary and initial conditions can be applied, after which a mesh must be generated. The solver settings include parameters such as time step size and iterations, but also the ways how to model advection and turbulence. It is possible to view results on planes and volumes, or create iso-surfaces or iso-volumes for all saved time steps. Quantities can be calculated for walls or parts and flow can be visualized using particle traces. Scenarios can be compared in an environment called the Decision Center.
In one CFD *study* (file), several *designs* can be created, differing in geometry. Each design can have multiple *scenarios*, differing in boundary or initial conditions, mesh or solver settings.

Other programs, such as Autodesk Fusion 360 and Autodesk SimStudio Tools 2016 R3 shall be used to prepare the board geometry - originally obtained from Aku Shaper - for simulation.

6 Initial model

Although this chapter is called Initial Model, it certainly does not describe the setup and settings of the model that was initially used. It however defines a good starting point from which improvements made during the project, can be explained. It has to be noticed that during the first weeks of the internship a lot of effort - circumventing many software and hardware errors, getting more understanding, defining problems - was done in obtaining this 'initial model'.

The model shall be described in the same order as the simulations are performed in Autodesk CFD: setting up the model, defining the solver settings and finally doing measurements from the results.

To keep track of all the runs, they were documented in a Run Sheet. Settings and drag results were saved and notes were made. For the most important setup information (speed, board, domain, mesh) codes were used in the run name. The Run Sheet can be found in appendix B.

6.1 Setup

The setup is started by defining the geometry, consisting of the surfboard and the domain. The materials in this geometry are assigned, after which boundary and initial conditions can be set. Then the mesh has to be generated.

6.1.1 Boards

For the simulations, digital versions of the boards are necessary. It is summarized here how the board files for the simulations are obtained.

Original board files
During the experiments three boards were used, only differing by their tail shape (see figure 5). The geometry of the boards was saved in Aku Shaper compatible .BRD-files that were supplied for this project. Unfortunately the file for the swallow tail did not exist, since Aku Shaper is unable to make this shape. Because no precise dimensions of this tail shape were available, it was not possible to make the geometry in another CAD-program. Therefore it was decided to not consider the experiments done with the swallow tailed board.
It is assumed the .BRD-files accurately represent the geometries of the boards that were used in the experiments.

File conversions for compatibility
The .BRD-files are not compatible with Autodesk CFD. But, from Aku Shaper the board geometries could be exported to .STL-files. It was necessary to than convert the geometry to a body representation and export it as an .IGS-file that could be opened by Autodesk CFD.

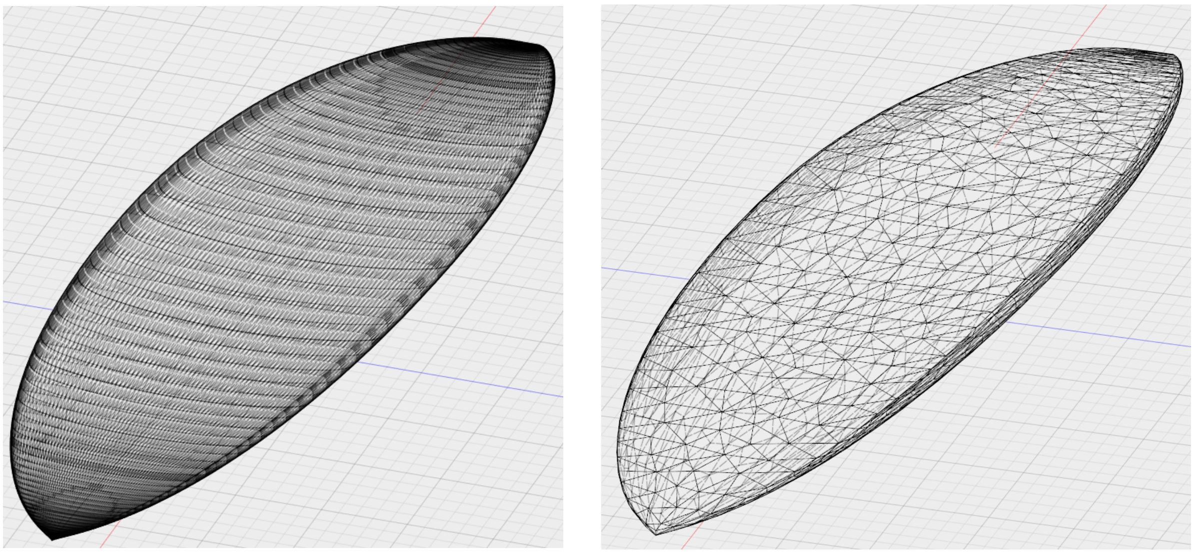
Mesh reduction for software performance
But before exporting the .IGS file, the number of faces had to be reduced. The .STL-files accurately represent the board shape, but have a large number of faces (over 22000) which slows down the software too much (low software performance). A reduction method had been found using the software programs AutoDesk Fusion 360, Meshlab and FreeCad.
It was verified that the board length, surface area and volume only differed less than 1% from the original .BRD-file geometries.

This board mesh reduction and file conversion turned out to be more complicated than expected and it took some time to figure out how it could be done. A detailed description is written for future use during the Aku Shaper project (appendix D).

Boards ready for simulation
The boards that shall be used at first are denoted with a code, the letter relating to their tail shape. The P00 board with a pin tail and the S00 board with a square tail both have a total of 2000 faces. Although mesh edges are clearly visible, the main shape is roughly the same as the non-reduced board shape (see figure 10).
By doing simulations (Part 1) it should be figured out what the effects are of increasing or decreasing the number of faces, so that for Part 2 digital boards can be used with the desired balance between shape representation and software performance.

6.1.2 Domain

Frame of reference
The experiments in the AMC tow tank were conducted in 1.5m deep water, the channel is 3.55m wide and 100m long. The board is moving relative to the water.
For the simulations it was decided to instead let the water move relative to the board. This saves the need for a more complicated Motion study in which the board would be moving inside the computational domain. Also it is then possible to make the domain shorter than 100m, since only the region nearby the board is of interest. Above that the simulated time is not limited, i.e. the board never reaches the end of the towing tank.



(a) Original non-reduced mesh, 23786 faces.

(b) P00, 2000 faces.

Figure 10: Pin board meshing, looking at the bottom, tail and left rail.

A Cartesian coordinate frame is chosen in which x is in downstream direction, the y-axis points upward from the initial free surface and the z-axis is in the boards width direction, positive to the right of the board. The origin shall be at a fixed position in the domain box.

Length conditions

There are still conditions for the length of the computational domain. The inlet and outlet boundaries must be far enough away. The Autodesk Knowledge Network (AKN) recommends to have at least 5 chord lengths upstream and 10 chords downstream of the body for external incompressible flow [13]. Although the model in this project is not surrounded by the water entirely (and therefore not really external flow), the need for boundaries to be sufficiently far away remains. The length in front of the board must be long enough so that the flow can be considered as fully developed once it reaches the board. Besides, the length behind the board must be long enough to capture the wake shape.

Domain dimensions

The simulations shall be started with a domain denoted with code D00 (figure 11), having a 1.5m deep water box and 0.5m high air box. The width is 3.55m and the length 6m. The sides and bottom of the box shall thus represent the walls and bottom of the towing tank, respectively. The horizontal interface between the water and air box shall form the initial free surface (see paragraph 6.1.5).
Note that the length of the domain is not according to the recommendations. The long domain as recommended shall have much more elements and is therefore considered too computationally expensive. It shall be investigated in Part 2 if the length of 6m is sufficient.

Board position and orientation

The board is placed at a fixed position in the domain, easily done in Autodesk SimStudio Tools. The submergence point (were the initial waterline crosses the centerline on the deck of the board) is made to coincide with the origin. The trim angle of the board is set to 6 degrees.

Not using symmetry

One may recognize that the setup is symmetric in width direction, i.e. left of the board is the same as right of the board. This could be used to reduce the full model to a half model with a symmetry boundary condition on the dividing plane, allowing for faster computation. It was however decided to stick to the full model, because a similar model might in the future be used for non-symmetric problems (with yaw or roll angle included) and verification for the full model was therefore desired by Steven Schmied.

6.1.3 Materials

Water
Both the water and air box are assigned the material 'Water' as in the Autodesk CFD material database (viscosity 1.003mPa/s, density 999.875kg/m^3 at -0.15°C). Water properties of the experiment were not documented. Although water temperature in the towing tank had certainly be higher than -0.15°C it is assumed that the differences in water properties between the experiments and simulations are of negligible influence on the results.

Board suppression
The surfboard is suppressed from the simulation. This means there actually only is a hole in the fluid which has the same shape as the board. The fluid experiences the boundaries of this hole as a wall and therefore flows around it, just like as it would flow around the surfboard. The advantage of suppression is that the surfboard does not have to be meshed with solid elements, which saves computation time. Board suppression should give the same results as not suppressing the board when the selected material has no surface roughness, according to the Autodesk Knowledge Network [24]. In Part 2 the influence of not suppressing the board and setting a surface roughness shall be investigated.
Autodesk CFD wants all bodies - even suppressed ones - to be assigned a material. Polystyrene was selected as material for the board.

6.1.4 Boundary Conditions

The Autodesk CFD example of 'How to set up a free surface analysis of a boat' [14] shows how the boundary conditions can be set. Instead of assigning slip conditions to the sides and bottom, no boundary conditions are applied here. In that case, Autodesk CFD will regard these faces as solid walls (just like the faces of the board). In figure 11 can be seen how boundary conditions are set for domain D00.

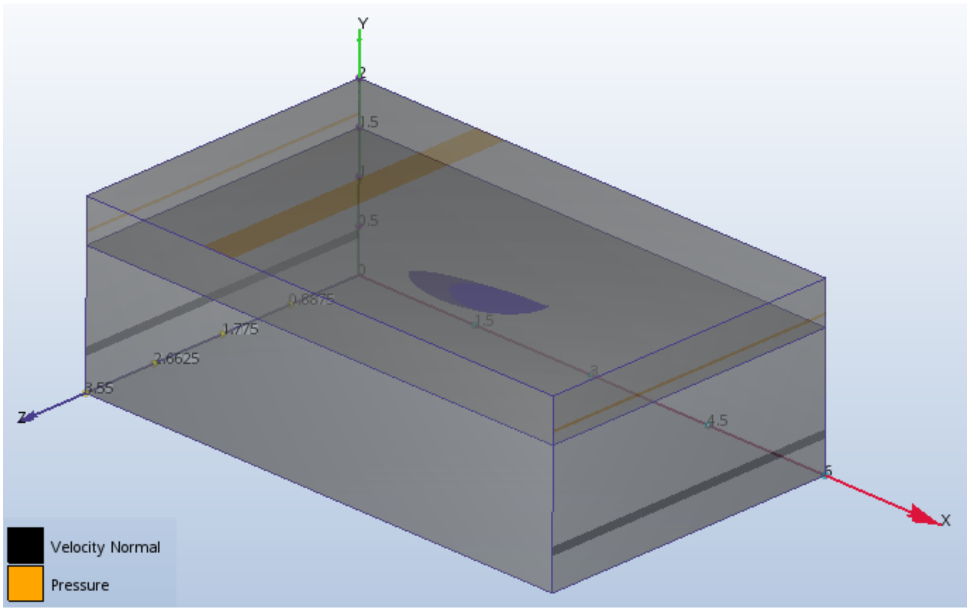


Figure 11: Boundary conditions for domain D00. The pressure applied to the air box is set to 0Pa and the velocity normal to the inlet and outlet of the water box can be adjusted for every run.

6.1.5 Initial conditions

Two initial conditions are to be set [14], both to the water box.
Firstly, the 'height of fluid' is assigned. This way, the total water box shall initially be filled with water.

The air box shall be empty and does actually not contain air, but something like a vacuum. This is the way Autodesk CFD models free surface analyses.
Secondly, the initial velocity of the entire water volume is set to a run specific value, in negative x-direction.

6.1.6 Meshing

For boundary and initial conditions it is relatively easy to judge if they are good, but this is a lot harder to determine for the mesh. The subject is too big and important to be captured in solely this paragraph. Actually, most of the time spent on Part 1 is devoted to finding a good mesh. In this paragraph some important considerations are described, as well as mesh settings that remained the same during this project.

It is fairly easy to obtain results with CFD software. Unfortunately, it is also easy to obtain results that are inaccurate. It is much harder to be sure that results are correct. Any CFD study is incomplete without emphasizing this important realization. One of the major causes for useless results is improper meshing. Also during this internship sub-studies are done on meshes of which results are not independent of the mesh resolution. This was due to running sub-studies in parallel (for there was limited time), without the mesh independence study being completed yet. Drawing conclusions from these results must be done with care.

Mesh independence
Results should not be dependent on the mesh resolution. So, gradients should not seem to align with the element faces. Also, results should be the same when the element size is decreased. This asks for a so called mesh independence study ([7, 15]). Once results are converged (see 6.2.6) on a start mesh, the same simulation is ran on a new finer mesh. Those converged results are compared to the ones obtained on the coarser mesh. If results between the meshes differ more than the specified tolerance, the mesh is refined again and so on until results are within the tolerance. There are several coarsening ratios (element size old mesh divided by element size new mesh) that can be used, just like that there are many results possible to be monitored. In paragraph 7.6 a mesh independence study is performed for Part 1.

Cloning scenarios and interpolation
Autodesk CFD has the option to clone a scenario. When the mesh is then refined, it shall interpolate the previous results onto this new mesh. This is supposed to save computation time, since the initial solution with which the new mesh starts is already closer to the solution that shall be obtained on the new mesh. Sometimes however, computation failed after interpolation. It then turned out to be better to just start solving from the set initial conditions. Both methods have been used throughout the project. It was also thought that cloning results could be useful for obtaining new scenarios in which the speed could be increased. This however turned out to cause incoming waves, resulting in solving errors. Therefore scenarios with different velocities were solved from scratch, instead of from clones.

Diagnostics
The Diagnostics functionality [12] in Autodesk CFD makes it possible to specify what the smallest edge lengths are that should be represented in the mesh. The function predicts if there are potential problems that could cause failure of generating a mesh. It was experienced that setting the Minimum Refinement Length to a minimum often helped to generate a mesh successfully. For this reason and because it was also desired that small edges of the board were represented in the mesh, the Minimum Refinement Length was always set to a minimum in this study.

Wall layers
In Autodesk CFD it is possible to add element layers along wall-fluid and solid-fluid interfaces, which is critical for accurate flow prediction [10, 25], especially of the boundary layer. The wall layer consists of prism elements that are added before generating the rest of the mesh.

Throughout this study two wall layer configurations are used. One is the default (5 layers, layer factor: 0.45, gradation: auto) and the other the advanced (10 layers, layer Factor: 0.6, gradation: 1.25). The advanced settings are recommended for the SST k-omega turbulence model [25].. Also Wall Layer

Blending is enabled, which causes a more gradual transition from the wall layer to the rest of the mesh. This should improve solution stability and accuracy [23] and was indeed experienced.

Manual meshing
Manual meshing shall be used to have more control compared to automatic meshing. Element sizes are assigned to the water and air volumes and to the board's surfaces, refinement regions can be specified. The software then generates an unstructured mesh build up of mainly tetrahedral elements.

Mesh codes
In this project many meshes have been used. Codes starting with the letter M have been used to specify the mesh settings. A complete overview of these mesh codes and settings can be found in appendix C. Generally, a letter further in the alphabet refers to a finer mesh.

6.2 Settings

Solving the simulation can be done in numerous ways. The applied settings are explained here. Non-mentioned settings can be regarded as set to the defaults.

6.2.1 Free surface

Since the water surface is to be captured by the simulation, a so called free surface analysis is performed. The gravity vector needs to be specified and is set to be pointing in negative y-direction.
Free surface analyses do have a few limitations: Air resistance, surface tension and minor sprays are not modeled [27]. Therefore it has to be assumed that these effects have had only limited effect on the experimental results. Furthermore, water is considered incompressible in free surface simulations.
It is warned that free surface analyses are extremely mesh dependent [27].

6.2.2 Turbulence model

Modeling turbulence is a complicated subject. Autodesk CFD comes with a wide range of turbulence models [19]. For some of the first runs, the k-epsilon model was used. Later on, it was replaced by the Shear Stress Transport (SST) k-omega SAS model, as recommended for matching experimental results by [25].

6.2.3 Advection scheme

In Autodesk CFD five schemes are available for modeling advection [11]. The default scheme ADV1 (Monotone streamline upwind) is used for the first runs, because it is numerically very stable and is recommended for free surface analyses [27]. For later runs, the ADV5 scheme (Modified Petrov-Galerking) is also used because it is recommended for the SST k-omega turbulence models [25].

6.2.4 Time steps and iterations

For transient free surface analyses the default time step size is 0.01s, with 3 inner iterations per time step. It is regarded as a good start [27] and is therefore often used.
As a rule of thumb, the time step size is recommended to be at least one tenth [17] or even one twentieth [16] of the time for the flow to traverse the length of the board, but preferably smaller. In this case it takes 0.39s for the water to pass the board in the fastest flowing scenarios, so a time step of less than 0.02 would be advised.
Sometimes - especially for the finer meshes - the solver quit and showed an error message: 'Analysis has stopped because solver got excited unexpectedly'. After trying a lot of possible solutions suggested by Autodesk [2], it was found that decreasing the time step size often helped resolving the problem. It was also mentioned that increasing the number of inner iterations could increase convergence [17]. Often, a

time step size of 0.005s was therefore used with 10 inner iterations.
Autodesk CFD has the feature Intelligent Solution Control [16], which has most of the time been enabled during this project. It automatically adjusts the time step size for better stability and accuracy. However, when the set step size is much larger than the desired step size, the simulation may still fail.

6.2.5 Solver computer

For this project a gaming computer was bought, having more computing power than an average consumer pc. During the first month of the project the computer unfortunately kept showing Blue Screen of Death errors, often when running simulations. This issue could not be resolved.

Luckily, it was also possible to let the simulations be solved by the Autodesk Cloud computer. It turned out this Cloud solving took more time than running on the local machine, but it was now possible to run simulations in parallel. New 'jobs' had to be uploaded to start new simulations and results had to be downloaded when completed. Later in the project - when computation times increased - it was discovered that the Cloud computer had a job time limit of three days. This meant results were downloaded after three days, even when the set number of time steps was not reached. It was possible to submit a new job, starting from the last results, but this limitation cost a lot of extra time managing all simulations.

6.2.6 Convergence assessment

A transient analysis is time dependent and it takes some time before a steady state situation is approached. It is important for the results to be converged, before results can be extracted and conclusions drawn. It is described here how convergence of results is assessed.

Autodesk convergence monitoring
Autodesk CFD monitors the values of the degrees of freedom - such as velocities - throughout the domain for all iterations. It is possible to view several quantities of these degrees of freedom in a Convergence Plot [21], such as the average over the domain, the minimum, maximum, residual (L2 norm) and a fluctuation indicator called DPhi/Phi. The data can also be exported so that values can be compared between runs.
The software also contains a feature called the Automatic Convergence Assessment (ACA) [20]. The ACA evaluates four convergence criteria for all degrees of freedom and stops the simulation when the convergence criteria are below a certain set threshold. Thresholds can be manually set, but default convergence levels are defined as 'Loose', 'Moderate' and 'Tight'. It is however remarked that hydrodynamically induced forces can continue changing even after ACA considers the analysis to be converged. Therefore running additional time steps and manually monitoring these forces (such as drag) is required [20].

Practical
Ideally all data for every run is thoroughly studied to secure the run has converged, but this is not practical to do for many runs. Especially in this study where many runs are performed for sub-studies, another approach was preferred.
For the first performed runs, ACA was enabled and set to 'Tight'. After ACA stopped the simulation, ACA was disabled and extra time steps were run. Average values of the degrees of freedom were monitored in the Convergence Plot until these graphs flattened. Additionally, drag forces were monitored (see 6.3.1). When drag force fluctuations were considered small enough, the run was completed.
For later runs, it was already known how many time steps were approximately needed. Therefore the ACA was always disabled. The average degree of freedom values in the Convergence Plot and the drag monitoring was still used to assess convergence.
For some final runs, a deeper analysis of the convergence was performed.
Of course, the visualization options in Autodesk CFD were also used to judge if no physically unrealistic results were obtained.

6.3 Assessing results

When a run was considered converged, results could be extracted. In this section is described how these are obtained.

6.3.1 Drag

Autodesk CFD contains the Wall Calculator, with which it is possible to calculate forces on faces. By grouping the surfboard, it was possible to view the force exerted by the fluid on the total board. The x-component of the force was taken to be the drag force.

As mentioned previously, convergence of forces had to be monitored manually. Therefore results at intermediate time steps were saved and the drag was calculated for the last five saved results. Typically, the saved time steps were 0.2s separated. Assessing more time steps was desired, but manually extracting the drag for multiple time steps was quite work intensive and considered undesirable to do for all runs. The sample standard deviation of this five point sample was taken. If the standard deviation was less than 1N, the drag force was considered converged. 1N was always in between 1-3% of the average of the five point sample. This value was chosen because it was noticed achieving this minor variation was possible. Besides, getting a smaller variation was considered both hard and non-realistic due to flow variations (wake, bow wave, etc.) happening in the experiments also.

6.3.2 Wake

In Autodesk CFD the free surface can be obtained by generating an iso-surface of the quantity 'Volume of Fluid' (VOF) and setting the value to 0.5. The free surface can be viewed to see the wake.

In the experiments, not much was documented about the wake. Therefore there also is not much to compare quantitatively. The free surface views generated by the simulations shall be assessed qualitatively. It shall be judged if they look similar to the photographs in the experiments.

Wakes of different simulated scenarios can easily be compared in the Decision Center.

The free surface shall also be assessed on any physically unrealistic looking flows.

6.3.3 Computation time

For some runs, the total time between submitting a job to the cloud solver and completion was documented. This time also includes the upload and download time, which could be a few hours for large simulations, but it still gives an estimate of the computation time.

7 Improving model

Once it was clear in general how the model was to be set up and what settings had to be applied, it was still unclear what would be a good mesh. Also, drag forces still differed substantially from the experiments and ways were sought to improve the drag resemblance.
In order to try improving the model, many sub-studies were executed. The ones that were of bigger influence are documented here. Often these sub-studies were run in parallel, because limited time did not allow for awaiting results. For that reason sometimes outdated settings or setups were used. Drawing conclusions from the sub-studies must therefore be done with care.
Not always the same scenario was modeled, i.e. not always the same board for the same speed. It was to be tested if everything also worked for different boards and speeds.

The following sub-studies - not listed in the order they were performed - are discussed:

- **7.1** Domain and boundary conditions

- **7.2** Submergence
- **7.3** Board mesh density
- **7.4** Roughness
- **7.5** Turbulence and Advection model
- **7.6** Mesh independence
- **7.7** Support from forum
- **7.8** Boundary layer

7.1 Domain and boundary conditions

For the first simulations the domain D00 was used, which had the same cross section as the tow tank. There were two problems with this domain.

1. It was found that the velocity boundary conditions in combination with the side walls as applied in figure 11 were not desired. In the experiment the board is moving relative to the motionless water. In the simulations the water is moving relative to the board and the flow distribution in the tank develops due to the tank shape, with zero velocities at the bottom and walls. The boundary condition at the entire inlet surface was a constant velocity normal to the plane. However, the velocity in the simulated tank actually reduces to zero at the side walls and bottom. Therefore some length in stream wise direction was necessary for the flow to develop, while actually this developed flow was not the same as in the experiments. Above that, the extra length was undesired since more elements are necessary and this increases computation time. The constant velocity boundary condition also gave similar problems at the outlet surface.
2. Elements in the bottom and top section were fairly unnecessary, just like elements further to the sides. The bottom section of the domain (the deeper water) did not show much change of flow. This was also seen for the elements closer to the sides of the domain. The higher part of the air box was also of no interest. All this domain space was considered superfluous, see figure 12.

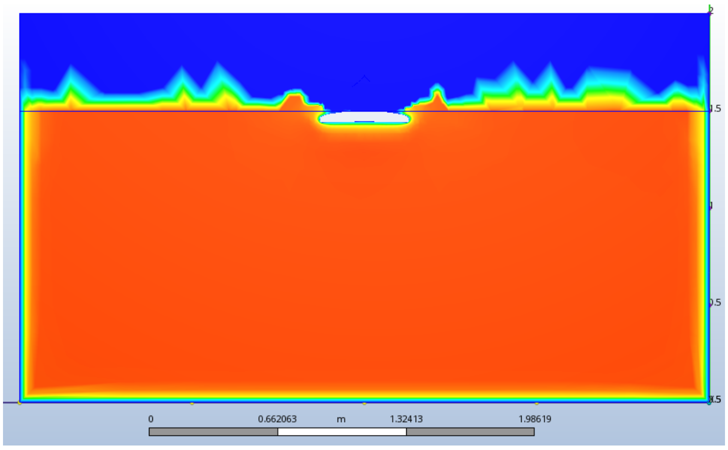


Figure 12: Superfluous domain space for domain D00.

It was decided to assume that the side walls and bottom of the tow tank were of negligible influence on the to be compared experimental results (drag and wake properties). The side walls are over three board widths away and the water was over eight times deeper than the vertical submergence.
The wave lengths of the wakes created by the board during the experiments were not measured, but it was estimated that the waves could be considered as deep water waves. This allows for measuring the wake properties in a simulation without a bottom.

With this assumption it is easily possible to solve the outlined problems by using a smaller domain and different boundary conditions. The domain denoted with code D20 is still 6m long, but only 2m wide and has a water depth and air box height of both 0.25m. There are now slip boundary conditions applied to the sides, as indicated in Figure 3. The inlet flow now is actually constant over the inlet surface. The boundary conditions are now comparable to the AutoDesk example of a free surface analysis of a boat [14]. The length of the domain remained the same. The length upstream appeared to be sufficient for the flow to become stable, i.e. only further from the inlet surface the incoming flow appears to be as expected. The length behind the board was able to capture most of the wake generated by the board and appears to be sufficient for extracting the to be measured wake properties.

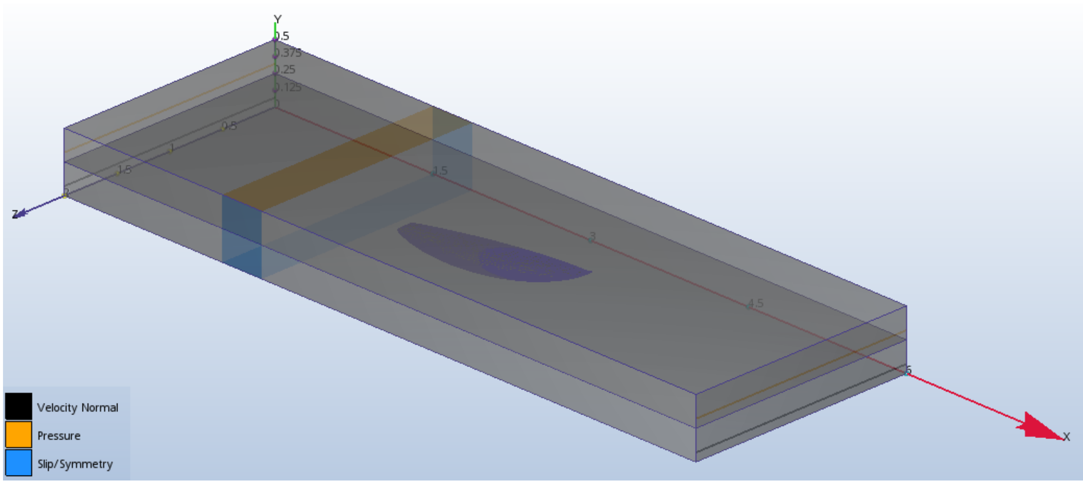


Figure 13: Domain D20 with applied boundary conditions.

In some simulations at the early stage, domain D10 was used which is similar to D20 but has 0.5m water depth and 0.5m high air box.

7.2 Submergence

As reported in paragraph 4.2.2, the board submergence had to be found by measuring distances at pictures. This was considered a precarious method for such an important measure. The influence of the submergence length on the drag force was to be investigated. This could give insight of how close drag could match the experiments, taking into account the submergence length uncertainty. It was expected that drag would increase with submergence.

7.2.1 Method

Setups were made in which the pin tail board submergence length of 767mm was changed by a few centimeters, from -3cm to +3cm in 1cm intervals.
The flow speed was 3.04m/s, domain D10 was used with mesh MB. The experimental drag measured for this case was 48.5N. Other settings were: ADV1, SST k-omega SAS, 0.01s time step with 3 inner iterations, default wall layers. Simulations were run for at least 3000 time steps, the runs turned out to be converged by then. The last five saved time steps were used for extracting drag forces.

7.2.2 Results

In figure 14 and table 2 the resulting drag forces are found. No trend is visible. Besides, for some submergence lengths the variation in drag overlaps the measurements of another submergence length. Runs for -2cm kept on failing, so no results are available for this submergence.

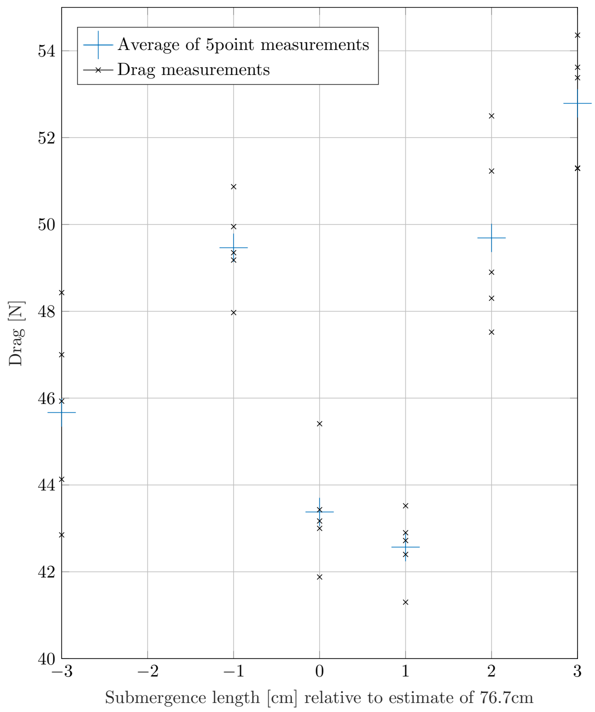


Figure 14: Drag dependency on relative submergence according to simulations.

Relative submergence [cm]	**Drag [N]**	**SSTD [N]**
-3	45.67	2.22
-1	49.46	1.07
0	43.38	1.28
+1	42.57	0.82
+2	49.69	2.09
+3	52.79	1.41

Table 2: Drag measurements for different relative submergence lengths.

7.2.3 Conclusion and discussion

No drag-submergence relationship could be determined from the results of this sub-study. It can be concluded that variation in drag measurements can be as large as 20% for this board-domain-mesh-speed combination, although unsure if this is caused by submergence uncertainty or something else. Let this sub-study be a warning sign that simulation results are not necessarily correct.

When this sub-study was performed, there was still uncertainty about the to be used settings. Above that, the mesh is not proven to be mesh independent. It would be wise to repeat the test when more confidence is achieved for the mesh and settings. Besides, actually tests for every speed and both boards are necessary, although it could possibly be assumed drag-submergence variation is similar for those cases.

7.3 Board Mesh Density

As described in paragraph 6.1.1, the geometry of the boards are represented by 2000 faces. It was desired to know what the influence of this number of faces is on the results.

7.3.1 Method

The square tail board was made with 2000 faces (S00), 4000 faces (S10) and 6000 faces (S20). There was also a board made (S30) for which only the deck mesh had been reduced, the bottom and rail mesh density remained as in the original .STL-file. The S30 board had still about 12000 faces.
Simulations were run with these boards for a speed of 4.55m/s, using domain D20 and mesh MNb. Other settings were: SST k-omega SAS, ADV1, time step 0.005s with 10 inner iterations, default wall layers. 2000 time steps were performed.

7.3.2 Results

The S30 board slowed down the software too much and no results were obtained for this board. Drag measurement for the other board are shown in table 3. It is seen drag forces are almost the same for each board, especially when the sample standard deviation is taken into account. The free surface of the last result sets is displayed in figure 15. It's clear that the wake for S00 looks different than for S10 and S20. Also, S00 does not show the bow wave just in front of the board. The S10 and S20 cases show little difference. For all cases, the water does not flow off the deck.

Board	**Faces**	**Drag [N]**	**SSTD [N]**
S00	2000	64.08	0.94
S10	4000	64.35	0.72
S20	6000	65.09	0.64

Table 3: Drag results for boards with different number of faces. The drag column represents the average of a five point sample, the SSTD column gives the sample standard deviation of this sample.

No great pictures of the wake were taken during the experiment of this case, the only one available is figure 9b. The wakes generated from the rails join at about 30%LOA behind the tail [29]. None of the wakes from the simulations clearly shows this behavior, but it is hard to tell.

It was also investigated if the degrees of freedom converged to the same values for the three cases. As an example, the average over the domain of the velocity in x-direction (Vx) is shown in figure 16. It was seen the averages converged to different values, but within the same order of magnitude. The same was observed for other degrees of freedom (Vy,Vz,Pressure,TKE,TED[1]).

7.3.3 Conclusion and discussion

For drag measurements, it does not seem to matter which board is chosen. Wake patterns look different for higher number of faces and all look different than in the experiment, although little detail is known for the experiments. Board is S10 is preferred for now, showing the same wake as for S20 using less faces.

These results are obtained with mesh MNb, which is not shown to be mesh independent yet. Also, ADV1 was used in combination with SST k-omega, which is not recommended.

It has to be noted that it is still not certain that S10 gives good results. Maybe the board meshing makes it hard to generate a good fluid mesh, or other problems might still be present. It shall be assumed that 4000 faces is also a good amount for the pin tail board.

7.4 Roughness

As was mentioned in 3.2, part of the drag is skin friction drag, increasing for increased surface roughness. In Autodesk CFD many materials do not have a specified surface roughness. In addition, when the board is suppressed, the walls of the remaining gap in the geometry shall have zero surface roughness. It is of interest to see if the drag forces in the simulations can match the experimental results better when surface roughness is assigned to a non-suppressed board. It is expected that increased surface roughness gives an increase in drag.

A surfboard's surface roughness depends on the applied surface finish, on scratches and dents and possibly on some wax (meant to put on the deck for grip) by accident getting stuck on the bottom or rail.

[1]TKE: Turbulent Kinetic Energy, TED: Turbulent Energy Dissipation [18]

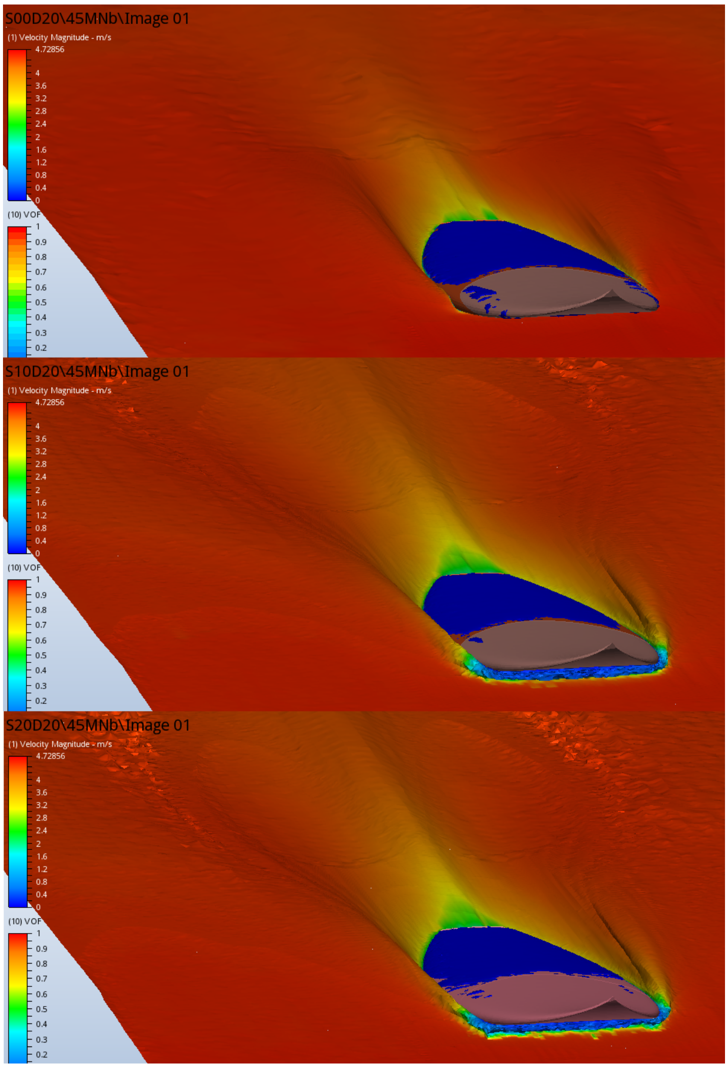


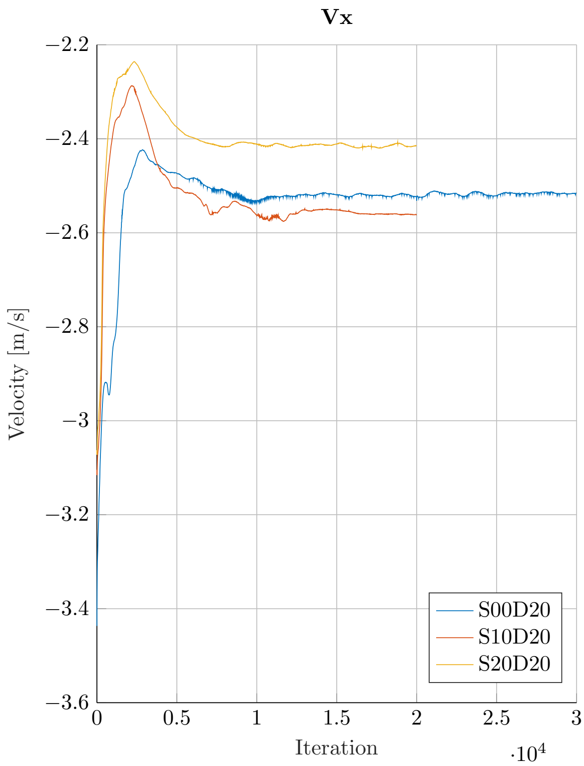
Figure 15: Free surface view of final result sets for the boards S00, S10 and S20 (from top to bottom). Color represents velocity magnitude, color scaling is the same in each image.

7.4.1 Method

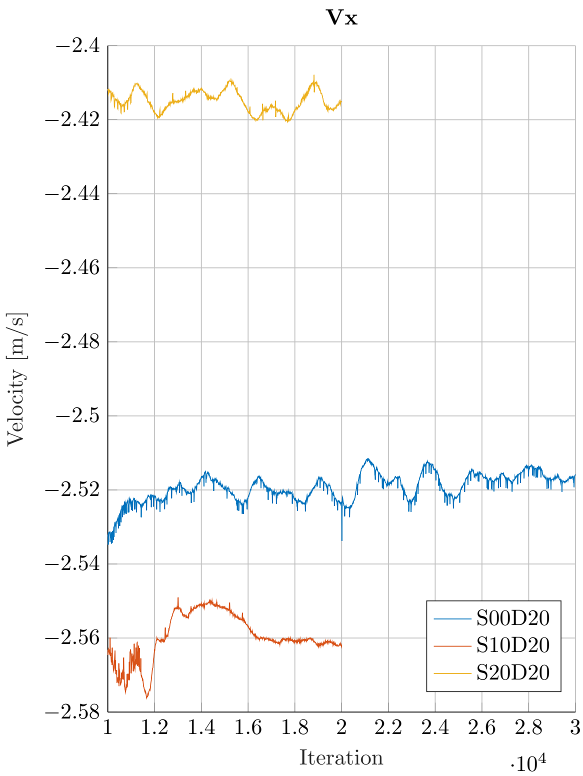
Four different surface roughnesses are assigned to the material of the board, namely 0.1mm, 0.01mm, 0.0015mm and 0.00075mm. The feature 'Intelligent Wall Formulation' (IWF) had to be disabled for these roughness cases [19]. These cases are compared to a board that is suppressed, i.e. has zero surface roughness.
The test case is the 4.55m/s S00 board in domain D20 with mesh MNb, using 10 wall layers, SST k-omega SAS and ADV5. Time step size is 0.005s with 10 inner iterations. Simulations were run until considered converged, which turned out to be after about 2000 time steps.

7.4.2 Results

The results for the 0.00075mm case were inaccessible, the software crashed when opening this scenario. Therefore no results are obtained for this case.
For the other cases the found drag values are visible in table 4. It's easily seen that the results for surface roughness differ substantially from the no roughness base case. It is however hard to see a relation between drag and surface roughness.



(a) All iterations



(b) Iterations from 10000

Figure 16: Convergence of domain averaged Vx for boards S00, S10 and S20. Figure (b) is zoomed in on the 'flattened part of (a). For board S00 more iterations were performed.

Roughness [mm]	**Drag [N]**	**SSTD [N]**
no	42.17	0.48
0.1	28.58	0.78
0.01	27.10	0.61
0.0015	28.41	0.74

Table 4: Drag dependence on board surface roughness according to the simulations of this sub-study.

The free surface wake patterns and other flow characteristics were compared in the Decision Center and the three roughness cases looked very similar to each other and to the no roughness case.

7.4.3 Conclusion and discussion

It can be concluded that applying surface roughness and disabling IWF results in different drag. Taking into account surface roughness is therefore necessary, although it is not sure yet how this can be done properly. No drag-roughness dependence could be estimated.

Quoting the Autodesk Knowledge Network about the SST turbulence models and Intelligent Wall Formulation [19]: *"It reduces the sensitivity of results to the level of mesh refinement along the wall."* This means that disabling it gives increased sensitivity to the mesh refinement along the wall (board). It's therefore a good idea to also study the surface roughness effects in a mesh sensitivity study, especially because the mesh MNb is also not yet shown to give mesh independent results.

7.5 Turbulence and advection model

Regarding the turbulence and advection model to be used, sometimes conflicting advices were found. As an example: for free surface simulations ADV1 is recommended [27], while ADV5 had to be used for the SST k-omega turbulence model [25]. Also, it was advised to first start the analysis with default settings (k-epsilon turbulence model, default wall layers) and go more advanced (SST k-omega SAS,10 wall layers) whenever experimental results had to be matched [25].
It was questioned how big the influence of these settings are on the results and what combination is preferred. It was hoped one combination turned out to match the experimental results closely.

Also, sometimes notes were made about 'increased computational cost' for using certain settings. It was desired to know how much this increase in solving time would approximately be.

7.5.1 Method

For the scenario 4.55m/s S00 with domain D20 and mesh MM, several cases - with different combinations of wall layers, turbulence and advection models - were simulated. These combinations (see table 5) were chosen because different interpretations of the literature could lead to these combinations. Case names referring to advection model, turbulence model and wall layers settings (in that order) were defined to easily separate them. Not always the allowed drag variation of 1N was obtained, but it appeared as if results were not improving anymore after running 2000 time steps (0.005s, 10 iterations). Some combinations just weren't stable enough to converge nicely. The two different meshes (different wall layers) were cloned to make sure all cases with the same wall layer settings had exactly the same mesh.
For clarity, the settings for wall layers as described in paragraph 6.1.6 are repeated here: default (5 layers, layer factor: 0.45, gradation: auto), advanced (10 layers, layer factor: 0.6, gradation: 1.25), both with Wall Layer Blending enabled.
Drag and computation time were extracted as described in paragraph 6.3.1 and 6.3.3. The experimental drag for this scenario was 104.36N.

7.5.2 Results

In table 5 the results for the different cases can be found. It is clear that big differences in drag are possible for different cases. It cannot be stated that a certain advection model always results in lower or higher drag, or takes less or more computation time. Advanced wall layers seem to roughly double computation time and result in lower drag. Based on the drag SSTD, the cases with advanced wall layers seem to converge better, especially the one using ADV5. This case, **5oa**, was also recommended for matching experimental results [25], but unfortunately takes the longest time to simulate.
As a extra test, the cases **1od** and **5od** were run for another 2000 time steps, so that total computation time came closer to the **1oa** and **5oa** cases. Still, **1od** resulted in 70.13 with 2.11N SSTD (extra 25h) and **5od** in 76.24N with 3.79N SSTD (extra 23h). This proves that advanced wall layers (also more elements) result in better convergence per computation time for the SST k-omega SAS model.

Case name	ADV	Turbulence Model	Wall layers	Drag [N]	SSTD [N]	Approx. comp. time
1od	1	SST k-omega SAS	default	71.21	3.34	25h
5od	5	SST k-omega SAS	default	77.37	4.76	18h
1oa	1	SST k-omega SAS	advanced	43.13	1.31	52h
5oa	5	SST k-omega SAS	advanced	43.58	0.78	54h
1ed	1	k-epsilon	default	67.84	2.53	18h
5ed	5	k-epsilon	default	35.31	1.89	23h

Table 5: Drag results and computation times for cases with various wall layers and advection and turbulence models.

Flow visualizations were assessed in the Decision Center. Free surface images can be seen in figure 17. The wakes for **1ed** and **5ed** appear way different than for the other cases. The cases **1od**, **5od** and **5oa** show similar wakes, a bit different to **1oa**.

For the cases **1oa** and **5oa** the simulations turned out to have stored only the final time step (although save settings were correct), for some unknown reason. Therefore these runs had to be restarted from time step 0 before multiple drag measurements could be done. But for the first series of runs (referred to as 'old') it was still possible to view the final time step to capture images. Surprisingly, some really different things are seen in figure 17 when compared to the restarted simulations (referred to as 'restarted'). The most striking difference is that in the old runs the water flows off the deck (as in the experiment), while it remains on the deck - inside the wall layers - for the restarted run (and all other runs). It was checked multiple times if there had been any changes in the setup or settings, but no differences could be found.

7.5.3 Conclusion and discussion

It is obvious that the turbulence and advection model, as well as the wall layer settings, are of huge influence on the results. Advanced wall layers seem to give better converged results for the SST k-omega SAS turbulence model (no other models tested), at the cost of extra computation time. Care must be taken in other sub-studies for drawing conclusions from a certain combination.

Unfortunately, no combination resulted in drag (almost) matching with the experiments. Case **5od** came the closest, but was still 26% off and showed the largest SSTD, which is undesired. When at least one of these combinations is the right one, it can be stated that something else of the setup or settings must be improved for better similarity.

For Part 2, matching experimental results is the main objective of the simulations. This sub-study and the Autodesk Knowledge Network both indicate case **5oa** - showing the best drag convergence - is desired for this purpose. For this reasons the more advanced settings (ADV5, SST k-omega, advanced wall layers) are preferred, at the cost of extra computation time.

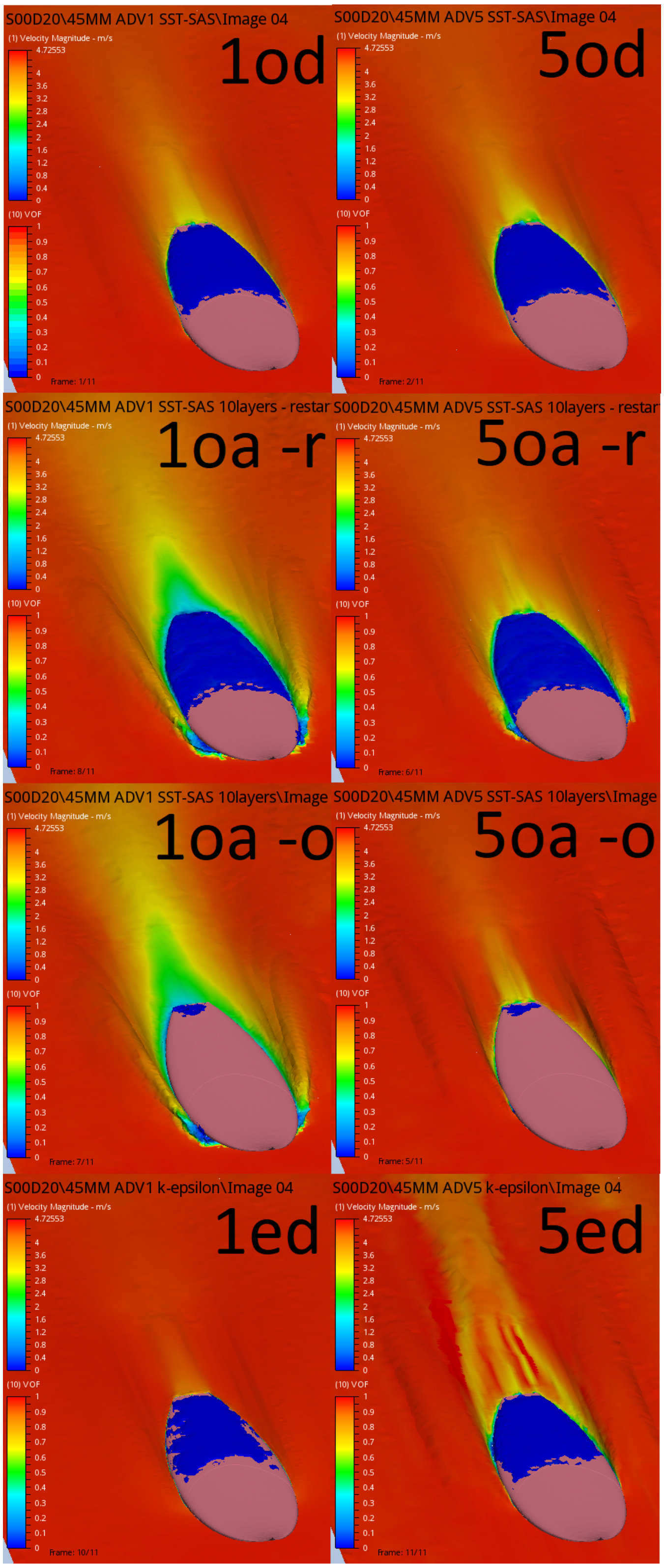


Figure 17: Free surfaces for cases with various wall layers and advection and turbulence models. The left images used ADV1, the right images ADV5. The upper three rows used SST k-omega SAS, the lower row used k-epsilon. The third row (old) used exactly the same setup and settings as the second row (restarted), differences are unexplained.

7.6 Mesh independence

If all other settings and setup are correct, there is still one very important subject to tackle: the mesh. As described in paragraph 6.1.6, a mesh independence study is required.

In the beginning of this project, many different things were tried to improve the mesh. It was especially tried to get better results using different refinement regions. Obviously, nearby the board was a region of importance, but also capturing the free surface was of interest. The problem was though, that it is at first not known where the free surface is going to be behind the board, which gave need for a relatively high (y-direction) refinement region. It was however found that having a fine mesh on the free surface in the entire domain did not give different drag results compared to meshes having only refinement nearby the board. Only for accurately capturing the wake pattern, a finer free surface mesh was desired. For this reason, it was decided to only refine nearby the board. Above that, computation time limitations would not allow for refining the entire domain for the finer used meshes.

7.6.1 Method

Five meshes have been used, each with a different element size nearby the board (refinement region). Mesh MK and ML do not have a refinement region. Some important quantities of these meshes can be seen in table 6. The general element size d_g and the element size in the refinement region d_r (shown in figure 18) were specified. The board's surface was also meshed with d_s, maximally the same element size as nearby the board. Default wall layer settings were used.
All meshes are applied to the 4.55m/s case of the square tail board in domain D20, having a experimental drag of 104.36N. The SST k-omega SAS turbulence model is used with the ADV1 advection scheme. The set time step size was 0.005s with 10 inner iterations. Computations were first ran until the Automatic Convergence Assesment set to 'tight' stopped the simulation. Then, additional time steps were ran until the average convergence plots did not show any improvement anymore. Convergence on each mesh was also monitored by seeing no change anymore in the residual norm over the domain.
Sometimes scenarios were cloned from coarser meshes and results were interpolated onto the finer mesh. But some scenarios were started from the initial conditions, because restarting after cloning sometimes gave errors.

Mesh	d_g **[m]**	d_s **[m]**	d_r **[m]**	**Approx. nr. of elements**
MK	0.1	0.03	no	320k
ML	0.05	0.03	no	750k
MM	0.05	0.025	0.025	1000k
MN	0.05	0.0125	0.0125	3100k
MO	0.05	0.00625	0.00625	16000k

Table 6: Important characteristics of the meshes used in the mesh independence study.

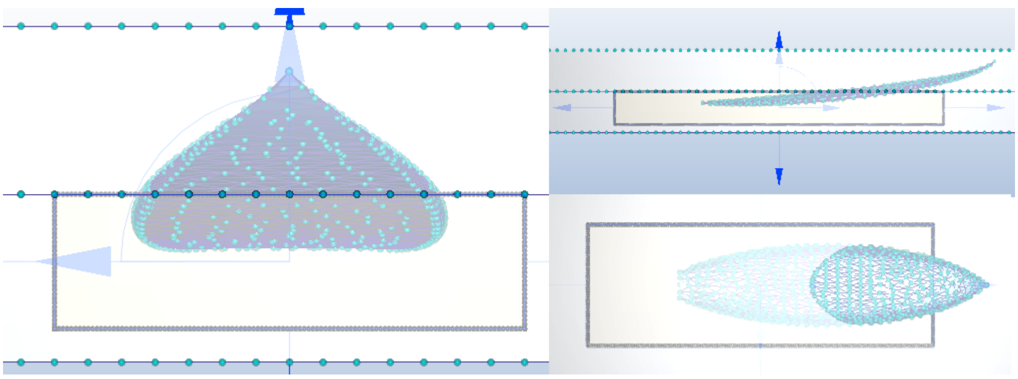


Figure 18: Refinement region around board as applied for meshes MM, MN and MO.

7.6.2 Results

Drag

In figure 19 and table 7 the measured drag forces for every mesh can be examined. Unfortunately the drag force does not look to converge to a certain value for decreasing mesh size yet. The finest mesh, MO, does not seem to be mesh independent. Finer meshes must be used to hopefully see the drag force converge.

One possibility for interpreting the results is that drag force might converge to about 63N, when recognizing the wedge shape between the data point, pointing to the left.This however does not seem likely and can only be tested by running some finer meshes.

In table 7 also the relative errors compared to the experiment are shown. All found drag values are still way too low.

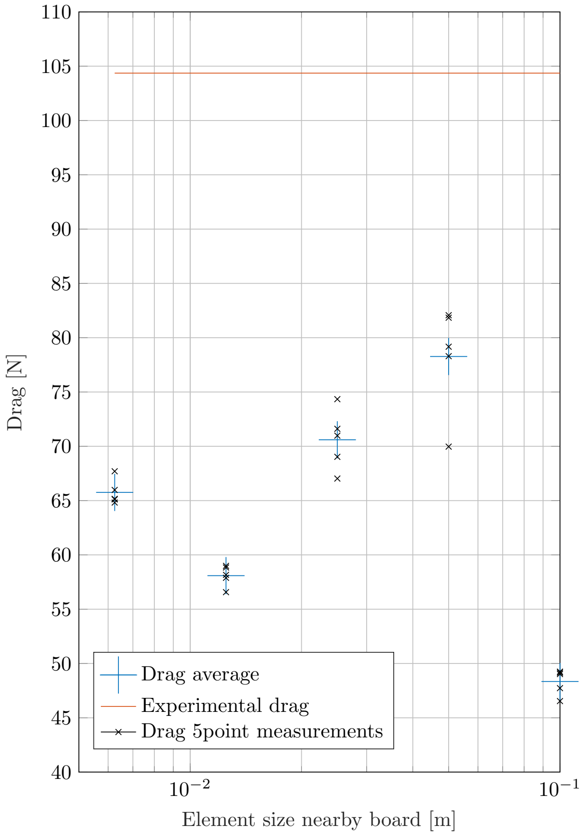


Figure 19: Drag measured in simulations for decreasing element size.

Mesh	El. size [m]	Drag [N]	SSTD [N]	$\frac{D_s-D_e}{D_e}$
MK	0.1	48.34	1.18	-0.54
ML	0.05	78.27	4.92	-0.25
MM	0.025	69.56	2.74	-0.32
MN	0.0125	58.09	0.97	-0.44
MO	0.00625	65.75	1.17	-0.37

Table 7: Drag measurements for different meshes.

Wake

In figure 20 the free surfaces are shown for the simulations of this sub-study. It is nice to see how the wake develops for finer meshes. The free surface of mesh MK does not look like the one of MO at all. The free surface for MM already looks similar to that of MO, but the shape is not that strong yet. The wake starting from the rail seems to bend more inward further downstream for finer meshes. The left and right wake do not join behind the board yet, as they do in the experiments.

The water does not flow off the deck. When a closer look is taken, it was seen that the motionless water remains in the wall layer on the deck.

A bow wave seems to develop for mesh MN. For mesh MO this bow wave is gone, although there is still some fluid attached to the bottom surface of the nose. This looks unnatural. When a different viewpoint is taken, it can actually be seen that there is a very strange valley occurring just in front of the board (figure 21). This is not expected to should happen in reality, so definitely something is wrong here. It is not known if there was a bow wave during the experiments. The simulations were continued for more

time steps and the valley eventually disappeared.
For MO and MN the transition from the refinement region into the rest of the domain is clearly seen as a line in the free surface. Although a coarser appearance of the free surface was to be expected, this division is a numerical phenomenon and should be minimized when comparing wake structure with the experiments. Possibly the free surface downstream of the board could be meshed as fine as the refinement region.
The wake pattern still seems to develop for finer meshes. Therefore it is thought finer meshes than MO are necessary.

Mesh
It is also important to view the mesh and judge if results are looking okay. It is however sometimes hard to view the 3D mesh while looking at 2D pictures. Two pictures are shown were improvement is necessary.
In figure 21a can the mesh nearby the rail can be examined. In the water, the transition from the board to the free stream looks gradual, which is good. In the top of the picture, the contour follows the mesh pattern, which is generally undesired. It is however thought this is in this case only happening due to the free surface present at that position. Another point of remark is that the transition from the free stream mesh to the wall layer does not look smooth, while that is actually needed [10]. Improvement is necessary.
In figure 21c the wall layer (blue) nearby the board (grey) is shown, at the division between the water and air box (initial free surface). Since the water and air box are different volumes, elements must be either only in the water or only in the air box. The wall layer becomes much thinner at this point, which is undesired. A way has to be found to remain the same wall layer thickness. For this specific picture it may not be a big problem, because the water level is below this point. But when this is caused by the mesh, it is a bigger problem.

7.6.3 Conclusion and discussion

Based on both the change of drag and wake for decreasing element size, it is concluded that the finest mesh used, MO, is not mesh independent. Besides, still some strange flow phenomena are occuring, probably because of improper meshing. The mesh therefore needs to be be improved. When also no mesh independence is achieved on this improved mesh, more refinement is necessary.
Unfortunately there was no time and no better computer available during this internship to investigate finer meshes. It already took about a month (many restarts) to obtain the results for the MO mesh.

Actually, ADV5 is advised for the SST k-omega SAS turbulence model, instead of ADV1 used in this sub-study. Also, at least 10 wall layers are recommended for this turbulence model [25]. It would be wise to redo this sub-study with these settings. For now it is assumed that the influence of mesh size is independent of the settings, so that MO is also not mesh independent for the more advanced settings.

It is also admitted that I have not had real hands on experience with meshing before this internship. Although I learned a lot, this remains a complicated and important subject and advise from an expert would be wise to consider to improve the meshing. Documentation about how the mesh can be improved for this specific problem in Autodesk CFD is not found.

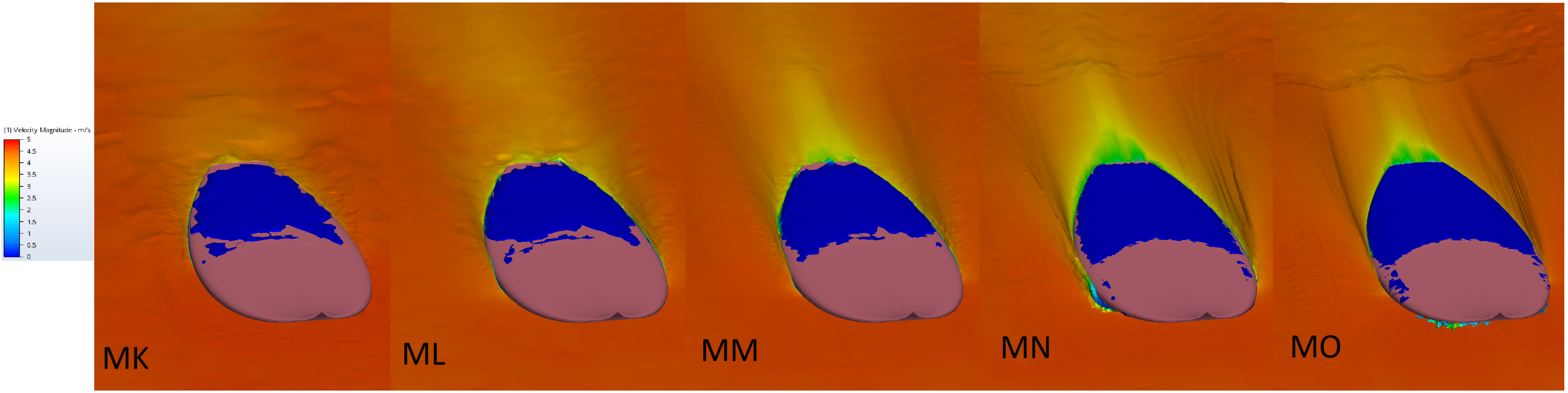
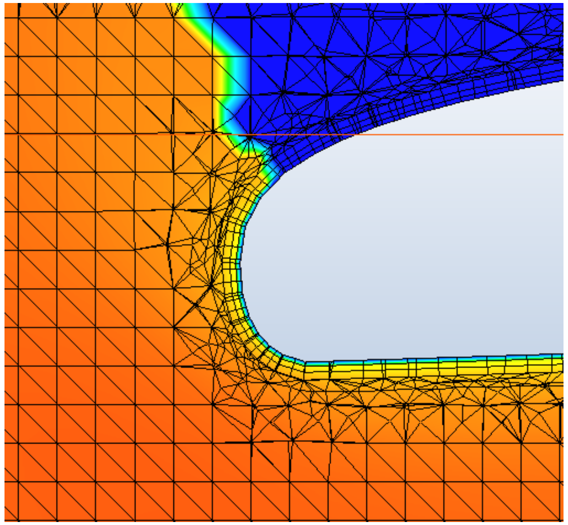
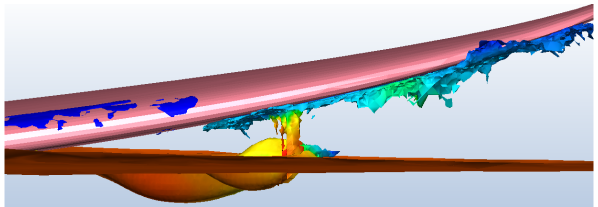


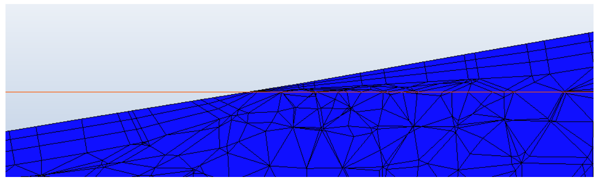
Figure 20: Wake patterns for different meshes.



(a) Rail, colored with velocity magnitude.



(b) Free surface near nose.



(c) Badly meshed point at wall layer-board interface around initial free surface.

Figure 21: Mesh views of the MO mesh.

7.7 Support from forum

At some point closer to the end of the project, it was noticed that results still differed a lot from the experiments. Many ideas of improving the results were already tried, while confidence about the setup and settings increased. It was more and more thought that the mesh was the problem. But, it was desired to know if there were not any other problems still present in the setup or settings. Since supervisor Steven Schmied also had only limited experience with Autodesk CFD, it was decided to ask support from the Autodesk Community through their forum.

7.7.1 Method

A question was posted on the Autodesk Community CFD Forum [2], together with a video in which the setup and settings were described. A few hypotheses were given for why the results differed from the experiments. It was asked if anyone saw any mistakes in the setup or settings, how the model could be improved and if anyone knew why the results were not matching the experimental values.
The post was about the 4.55m/s S00 case with domain D20 and mesh MNb, using ADV5, SST k-omega SAS and advanced wall layer settings. With time steps of 0.005s and 10 inner iterations, the results did not improve anymore after 2000 time steps.
Mesh MNb has a smaller refinement region (less height in y-direction) compared to MN. This saves some elements, without a noticed decrease in accuracy.

7.7.2 Results

A few people responded, under which Autodesk Technical Support specialists David Short and Matt Bemis.
They thought the setup and settings as applied were correct. They reasoned the results might indeed differ due to a too coarse mesh and due to a non-smooth transition from the wall layers to the free stream mesh (as already noticed in figure 21a). Having a more uniform mesh nearby the board could, according to David Short, significantly increase accuracy. The improper meshing is probably a result of the board geometry consisting of many irregularly sized triangular faces, while the wall layer is made up of prismatic elements. Improving the CAD model of the surfboard, preferably out of rectangular faces, may be necessary in order to obtain a good mesh.

7.7.3 Conclusion and discussion

More confidence is gained for the setup and settings. It is now very likely the mesh is still causing problems and that may be the main reason why results differ so much compared to the experiments.

Two to be solved issues with high priority are found.
At first, the CAD model must be improved. The need for fewer faces conflicts with the need to accurately represent the shape. Having a more uniform rectangular shaped board mesh would be preferred. This has been tried a few times without success. Maybe solutions can also be found in having curved edges representing the board shape instead of many straight edges.
Secondly, the mesh in Autodesk CFD needs to be improved for the better CAD model.

7.8 Boundary Layer

It is necessary that the simulations correctly capture the boundary layer to give a correct indication of the skin friction. There is however some indications that the meshing around the boundary layer, so nearby the surfboard surface, is currently of poor quality. Figure 22 shows some prism wall layers as

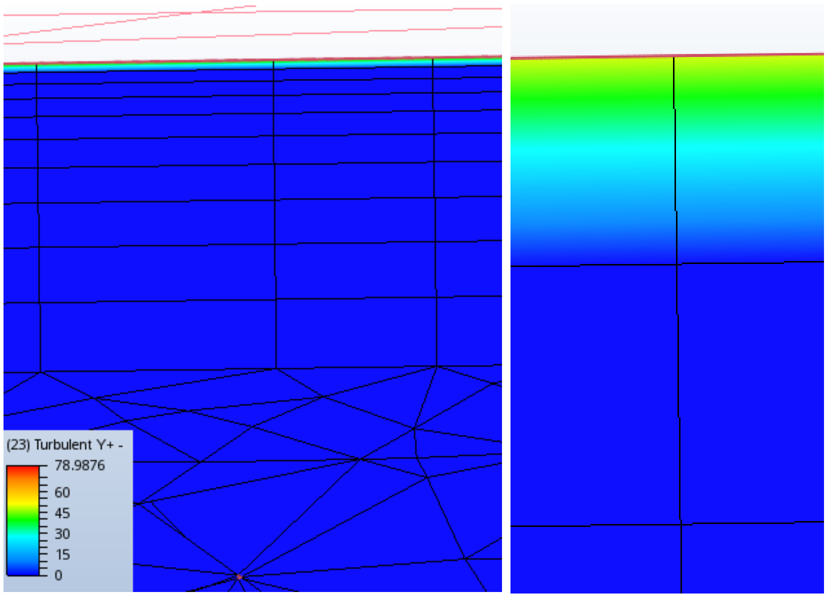
[2] View the post and discussion here: `http://simhub.autodesk.com/discussions/threads/330/post/7677284`.

applied to the surfboard bottom surface in the model of the previous paragraph (7.7). In this paragraph shall be described why the current wall layers are of poor quality and how this should be improved.

7.8.1 Transition from prismatic to tetrahedral elements

At first it is seen in figure 22a that the transition from the prismatic wall layer elements to the tetrahedral elements does not look smooth. The tetrahedral element sizes vary a lot and aspect ratios are sometimes high. In figure 24 can be seen how a desired transition must look like [10].

It is has not been discovered how this transition can be controlled in Autodesk CFD. It seems to be dependent on the element sizes inside and outside of the wall layer. Having this control is necessary and investigating ways to achieve this remains a recommendation. It could also be that another program is needed for manually applying a structured mesh.



(a) All layers. *(b) Layers closest to the wall.*

Figure 22: Wall layers nearby the surfboard (white/gray) as applied to the model of paragraph 7.7, MNb.

7.8.2 Thickness of first wall layer

Figure 22b also shows that y^+ in the inner wall layer is as high as almost 50 and reaches even almost 79 somewhere in the domain, whereas y^+ around 1 was recommended for SST k-omega turbulence models [25, 9, 10]. y^+ is a dimensionless number defined by equation 7.1 and indicates the distance from the wall. It can be used to estimate where in the boundary layer the first node of the wall layers should be placed, i.e. the thickness of the layer closest to the surface.

$$y^+ = \frac{u_* y}{\nu} = \frac{u_* y \rho}{\mu} \quad \rightarrow \quad y = \frac{y^+ \mu}{\rho u_*} \tag{7.1}$$

In equation 7.1 the viscosity μ and density ρ are fluid properties, u_* is the friction velocity and y is the vertical length from the wall. Note the similarity with the Reynolds number (equation 7.5).
The boundary layer consists of several regions [30, 9, 10] that can be seen in figure 23. The viscous sublayer is relatively thin and, in contrast to the larger log-law region, inertial forces are not dominating here. Low Reynolds number turbulence models such as the SST k-omega models are able to capture this viscous sublayer and therefore give better drag prediction, as long as the first wall layer node

is placed in this region [9, 10]. This, when looked at figure 23 explains the recommended y^+ of 1, as well as the indication given by the Autodesk Technical Support Specialists that 2-3 would also be still okay. The viscous sub-layer exhibits laminar flow characteristics and therefore has low turbulent viscosity. The highly turbulent log-law region however contains a maximum turbulent viscosity, usually around the middle of the boundary layer. This can be used to see if enough wall layers are applied. When the maximum occurs further than halfway the observed boundary layer, the wall layers may limit the boundary layer growth. An example of correctly applied wall layers can be seen in figure 24. Unfortunately it was not possible to extract a similar figure from the Autodesk CFD simulations.

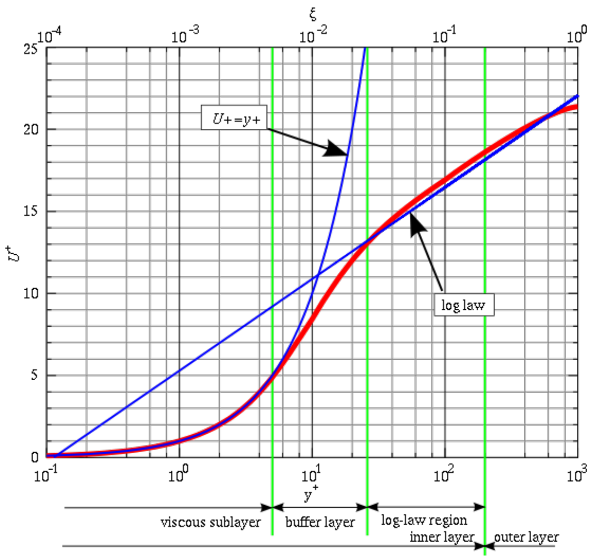


Figure 23: Regions inside the boundary layer. $u^+ = u/u_*$ Image by aokomoriuta - C BY-SA 3.0.

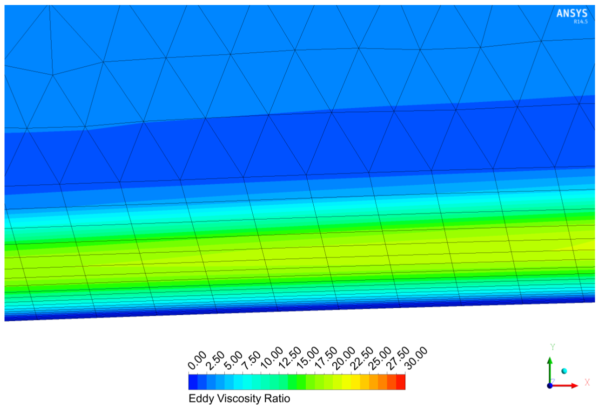


Figure 24: Example of desired transition and wall layers for SST k-omega turbulence model using $y^+ = 1$ [10].

Methods are proposed [8, 5] and used here to estimate the position of the node closest to the wall, in order to obtain the y^+ value that is recommended for the turbulence model. The friction velocity u_* is defined as:

$$u_* = \sqrt{\frac{\tau_w}{\rho}} \tag{7.2}$$

with wall shear stress τ_w. The wall shear stress is dependent on the velocity gradient in direction perpendicular to the wall and can be calculated using the skin friction coefficient C_f according to:

$$\tau_w = \frac{1}{2} C_f \rho u_\infty^2 \tag{7.3}$$

with free stream velocity u_∞. Several approximations for the skin friction coefficient can be used [4], for example one from Schlichting:

$$C_f = 2 \log_{10}(\mathrm{Re}_x) - 0.65)^{-2.3} \quad \text{for} \quad \mathrm{Re}_x < 10^9 \tag{7.4}$$

The Reynolds number needed, also to validate the use of equation 7.4, can be based on the boundary layer length L and is given by equation 7.5

$$\mathrm{Re}_x = \frac{u_\infty L \rho}{\mu} \tag{7.5}$$

It is assumed that the boundary layer length for the surfboard scenario is roughly equal to the submerged length of approximately 0.67m. This is considered reasonable since the Reynolds number for which transition approximately occurs on a flat plate ($\mathrm{Re}_x = 5 * 10^5$) happens when $x = 0.11$ and the surfboard bottom is more or less comparable to a flat plate, although of finite length. When the simulated water properties ($\mu = 1.003 * 10^{-3}$kg/sm, $\rho = 999.875$ kg/m^3) are used for the scenario of $u_\infty = 4.5$m/s, than $\text{Re}_x = 3.4 * 10^6$ and $C_f = 2.9 * 10^{-3}$. Other skin friction coefficient approximations also found very similar values. Substitution in equation 7.2 and subsequently into 7.1, using the desired $y^+ = 1$, results in $y = 5.9 * 10^{-6}$m. This is the desired distance between the wall and the first node away from the wall. In the mesh of figure 22b this distance is about $2.2 * 10^{-4}$m. It is noticed that when this value for y is used in equation 7.1, indeed the same order of the in figure 22b observed y^+ values is found.

So, the first node of the wall layer should be placed much closer to the wall (by a factor of about 40) and the entire wall layer thickness must be decreased. It is currently not known how these thinner wall layers can be applied in Autodesk CFD, since it is not possible to manually set a thickness. This remains an investigation to be done in future research. Maybe another program needs to be used to apply a structured mesh in order to have extra control over the element sizes.

8 Results

In the previous chapter a lot of investigation was done on the setup and settings of the model. Despite all the effort, no successful simulations were performed that came close to the experimental results. Points of improvement - mainly on the mesh and CAD model - had however been found but unfortunately these issues could not be solved anymore during this internship due to limited time.

Although absolute similarity with the experiments could not be found for the drag forces (values still differed substantially), maybe still a similar drag-speed relation could be obtained. In this chapter, all runs done in the experiment shall be mimicked as good as possible. The results are assessed and compared to the experimental ones. Based on these results, the research questions are answered as much as possible.

8.1 Final runs

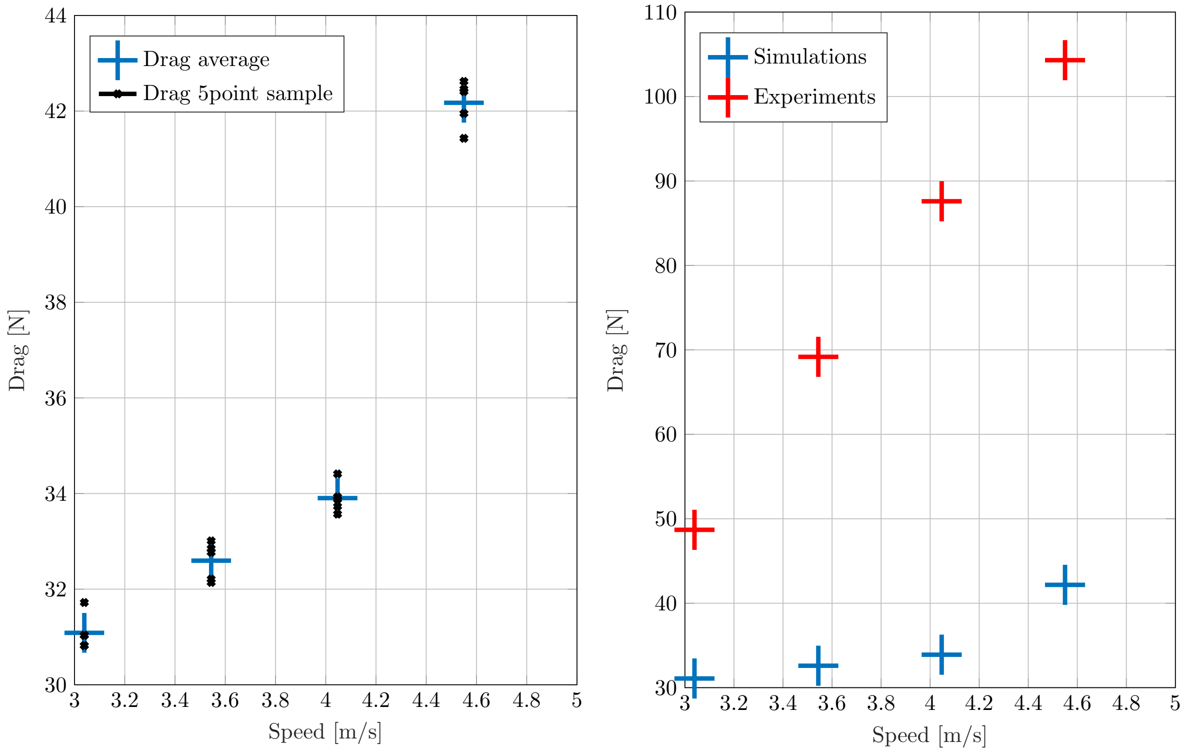
In this paragraph is described how the final runs are simulated. Setup and settings have been underlined.

The final runs are performed using domain D20 and mesh MNb. Although it is known mesh MNb does not give mesh independent results, no better options were available. Computation on mesh MO simply takes too long and restarting several times for all runs is too unpractical.
The S00 and P00 boards were used. At the time of starting the final runs, the results of the board sub-study described in paragraph 7.3 were not obtained yet. Luckily a conclusion of this sub-study was that similar drag forces were found for S00 and S10.
The board was suppressed from the simulation. There was no confidence yet about how surface roughness had to be applied in combination with the Intelligent Wall Formulation, although it was shown to affect the results in the sub-study of paragraph 7.4.
The SST k-omega SAS turbulence model in combination with ADV5 and advanced wall layer settings is used. This was recommended for matching experimental results [25] and was shown to give best convergence per computation time in the sub-study of paragraph 7.5.
A time step size of 0.005s with 10 inner iterations had proved to be running without problems and nice convergence within 2000 time steps, so these settings were used now too.

8.2 Drag

For the pin tail board no results were obtained. For some unknown reason, these simulations took much longer. These simulations are - at time of writing - still around time step 300 and had to be restarted several times, while the simulation for the square tail board only had to be restarted once to complete the 2000 time steps. It has been thoroughly investigated if something in the settings or setup was different, but nothing was found. One of the only possible explanations is that the flow is more complex for the pin tail board and therefore takes longer to solve.
Drag measurements were performed for the square tail board and the results can be viewed and compared

to the experiments in figure 25. The SSTD for the drag measurements was always less than 1N, i.e. the black crosses in figure 25a are close to each other for each run. It is seen that drag increases with speed, but not in the same way as for the experiments. The experimental dependency looks almost linear, whereas it is hard to say what the relation is for the simulations. More speeds had to be measured before formulating a conclusion on this can be tried. The slope dD/dS in the simulations (minimal 2.6Ns/m and maximal 16.5Ns/m) is less than the experimental slope (in between 33 and 41Ns/m). Due to this different slope, the difference between the experiments and simulations is larger for higher speeds.



(a) Drag measurements

(b) Simulated compared to experimental results.

Figure 25: Drag-speed dependence for square tail board, according to simulations and experiments.

8.3 Wake

In figure 26 the free surface wakes of the final runs can be examined. It is seen the wake structure does not change much when the velocity is increased. The wakes escaping from the rails become sharper and the tops stretch out further downstream. But for none of the scenarios the wakes join behind the board, as they do for the experiments in figure 9b.
Figure 27 zooms in on the bow wave of the 4.5m/s case. It is seen that the free surface looks very edgy due to the mesh, which is unrealistic. Still, the build up of water in front of the board is to be expected. The rugged free surface just in front of the board is also strange and does not occur for the other cases. The scenario was run for more time steps and the rugged free surface eventually disappeared.

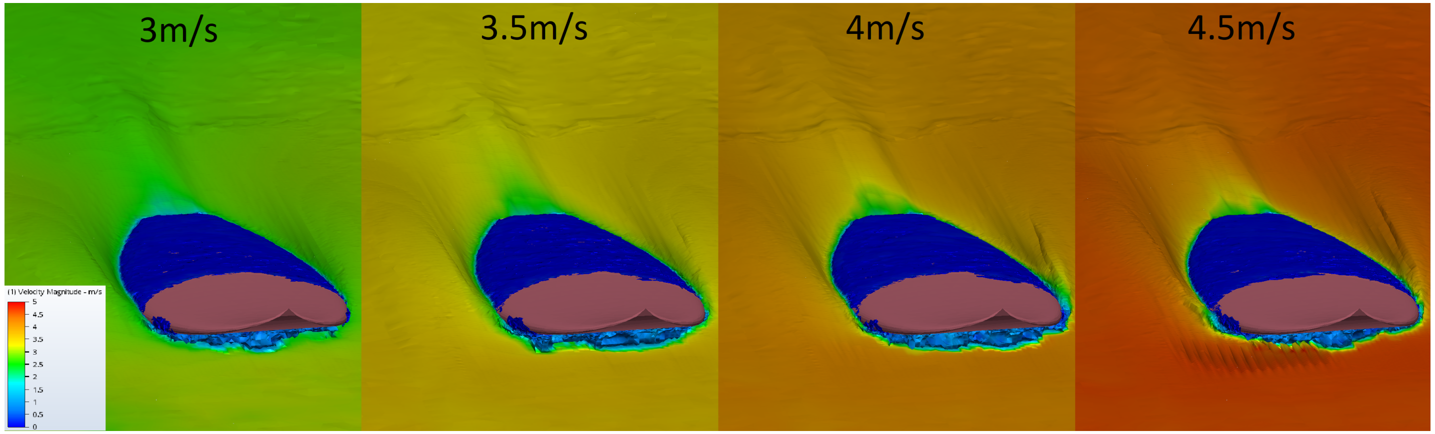


Figure 26: Wake patterns for different velocities of final runs.

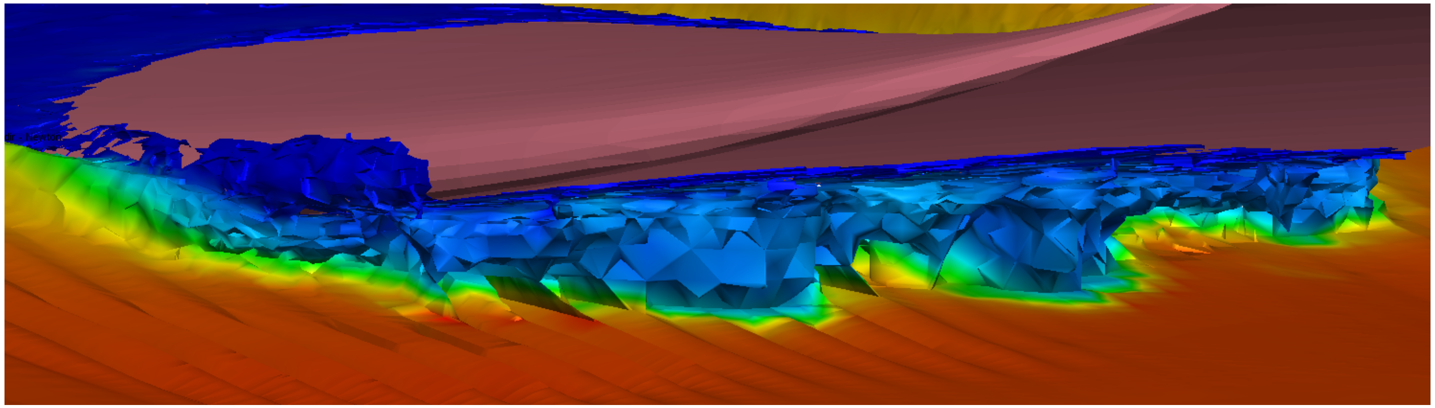


Figure 27: Rugged bow wave and free surface in front of the 4.5m/s scenario of the final runs.

8.4 Answering research questions

- *How much do the simulated drag forces differ from the experimental drag forces?*
  For the final runs, drag forces are in the simulations 17 to 62N less than in the experiments. Simulated drag forces only reach 40 to 64% of the experimental drag.
- *What is the relation between drag and speed for both tail shapes, and how does this differ from the experiments?*
  For the simulations drag increases with speed, but with a much lower slope and not in the almost linear manner as in the experiments.
- *What wake patterns appear during the simulations and what are the differences with the experiments?*
  The simulated wake patterns also show wakes escaping from the rails, but they do not join behind the board as they did in the experiments.
- *How must the CFD model be setup in future surfboard analyses to find realistic results?*
  Unfortunately no conclusion can be drawn for this question. It it thought the setup and settings as for the final runs (paragraph 8.1) are in the good direction, but improvements have to be made mainly on the mesh and CAD geometry.
- *In case drag and wake measurements for the model would match the experiments within acceptable tolerance, what limitations of the model - besides limitations imposed by assumptions - need to be circumvented before the model is usable for the parameter study?*
  The model would still be far from a realistic surfing scenario, mainly due to it's fixed position and orientation and flat water surface. When approaching the question from the other side - i.e. *possible improvements* making the model suitable for a parameter study - more useful advise can be given. This is done in the recommendations (chapter 11).
- *What assumptions were made for setting up the model?*
  Throughout the study many assumptions were made. Assumptions related to CFD in general are not mentioned.
  - Water can be considered as incompressible.
  - Differences in water properties between the experiments and simulations are of negligible influence.
  - The .BRD-files that formed the base for creating the CAD models accurately represent the geometries of the surfboards.
  - Modeling the air as some kind of vacuum - as is the only way of doing it in Autodesk CFD - and therefore neglecting any aerodynamic drag, does not give noticeable different results than the experiments.
  - Surface tension and sprays - not possible to model in Autodesk CFD - have not had significant effect on the experimental results.
  - In the experiments, the side walls and bottom of the towing tank - that were not modeled in domain D20 - have had only negligible influence on the results. Therefore the wakes could be considered as deep water waves.

9 Conclusion

Unfortunately, it has to be concluded that the drag forces obtained from the final model did not come close to the experimentally measured values. Even if one would be satisfied with only obtaining relative similarity instead of absolutely matching drag forces, the current models are not good enough for this. Also the wake structures looked different. It is considered unlikely that these differences are caused by the made assumptions. Throughout the performed sub-studies results are obtained that strongly point in the following direction as being the most likely cause, with possible sub-causes:

- Not capturing flow nearby the board accurately, due to:
  - An unsuitable mesh. The improper mesh is caused by:
    - Too coarse elements. Refinement was not possible due to limited time and computer power.
    - Non-smooth transition from board to free stream mesh. The software was unable to improve the mesh due to high order of non-uniformity in the CAD geometry.
    - Too thick wall layers. It has been calculated how thin the first layer should be, but there was no time left to correctly apply and test this.
  - Not being able to correctly model surface roughness of the board

These possible causes were not confirmed as necessarily being the reasons for non-matching results, since limited knowledge and limited time (also being one of the reasons for not obtaining the required knowledge) did not allow to investigate this any further, and these problems could therefore also not be resolved during this internship. Other possible causes are mentioned in the Discussion (chapter 10). Potential improvements are mentioned in the Recommendations (chapter 11).

The goal 'to deliver a validated CFD model that could be used for the parametric analysis' is not entirely achieved. However, still a lot of progress in the right direction is made. Besides, it can be concluded that adjusting the goal of this internship from a parameter study towards CFD model validation, has been a wise decision. In case the parameter study would have been performed with the models that were used at the beginning of this internship, worthless results would have been obtained, or even worse: incorrect conclusions would have been drawn from those results. The work done during this internship emphasizes the need to be cautious with drawing conclusions from CFD simulations.

Although results did not match the experiments, still a lot of insight is gained in how to model surfboard hydrodynamics in (Autodesk) CFD. This knowledge is well documented and stored for future use in the Aku Shaper project.

10 Discussion

A lot of sub-studies were conducted using unsuitable settings and improper meshes. Remarks about these cases had already been made in the corresponding sections. Also, the most likely causes for not obtaining results matching with the experiments, were mentioned in the Conclusion. Besides that, some other points need to be taken into account regarding the simulations performed.

The submergence of the experimentally used board was estimated by measuring distances on pictures. This is an imprecise method for obtaining a quantity with probably great influence on the drag force.

The .BRD-files were assumed to accurately represent the shape of the boards used in the towing tests. Differences from these files are inevitable by two ways.
Firstly, the actual physical board shall have been different from the digital design due to production processes, such as cutting errors and surface finishing.
Secondly, the digital designs were converted to different file types and mesh reduction was performed, both inducing changes to the geometry.

Furthermore, there is still the possibility that experimental results were (also) incorrect. It has to be emphasized that this is considered unlikely, because of the simple configuration and physically realistic looking results of these experiments and other experiments conducted in that some project. But, some details were not clearly explained in the corresponding documentation. There remains a chance that this could have led to incorrect interpretation of the experimental setup.

11 Recommendations

Based on the results and experiences during this internship, some recommendations can be made for future investigations as part of the Aku Shaper project. Because of restricted time for this internship, it was not possible for me to continue applying these recommendations myself. The recommendations are prioritized in order to show which issues are - according to me - most urgent.

High priority:

- **Improve mesh**. Conduct the mesh independence study with more computing power and an improved mesh. Maybe investigate the use of a different program to create a (structured) mesh.
- **Apply surface roughness**. Investigate how settings must be applied such that effects on the flow by surface roughness of the surfboard can be correctly modeled.
- **Wall layers: make y^+ 1**. It was seen that y+ values near the board are much higher than 1, whereas around 1 is recommended for SST k-omega SAS. Decreasing y+ might improve the way turbulence is captured in the model. This could be achieved by having thinner wall layer elements near the board [8].
- **Conduct Motion Study**. Make the model motion driven, so that the board is actually floating on the water due to it's buoyancy and hydrodynamic lift. Applying forces on the board could then imitate the forces normally applied by a surfer's feet. Board orientation and position are then dependent on the interaction between board, flow and applied forces.
- **Improve CAD model**. High order of non-uniformity of the triangular faces making up the current CAD surfboard model is likely to cause difficulties in mesh generation. A way must be found to have more uniformity and preferably rectangular faces, while still maintaining the shape accurately and not loading the software too much.

Medium priority:

- **Try different software**. Autodesk CFD works relatively intuitively and has great visualization options, but from a scientific point of view it is a pity that it lacks options to extract values. It is thought that other CFD programs are often able to export list results (raw data). Values are probably preferred when designs are to be compared quantitatively and when empirical relations need to be derived in the parameter study. Constructing visuals from numbers is relatively easy using other software (Paraview, Matlab), whereas obtaining numbers from visuals is much harder. It is thought that a lot of the knowledge obtained during this internship can still be applied in a different CFD package.
- **Use symmetry**. Save a lot on computation time by using the board and domain symmetry (use half the domain). The symmetry of the experiment was not used in the simulations, although I really believe it would not be a problem. Drag could simply be doubled for comparison with the experiments. When the half model is validated, one run could be done to validate the half model against a full model. Than the full model is ready for the (often non-symmetrical) parameter study.
- **Better free surface capture**. It should be investigated if meshing the free surface with a finer mesh could result in better capturing wave making drag. A better computer is needed for this, since the total number of elements shall greatly increase.
- **Implement fins**. Fins are an essential part of the surfboard. Implementation of fins in the design is definitely desired for the parameter study.
- **Model waves**. The current model is a board traveling over flat water and is far from real life surf situation. A way should be invented to model (surfable) waves in the CFD simulations.

Low priority:

- **Change water properties**. See how results change when water properties are changed. In the simulations they were probably not exactly the same as in the experiments.

- **Make board symmetric**. Use a symmetrical board (left side same as right side) and see if symmetrical results are obtained. Now, due to the remeshing, the board was not exactly symmetric, whereas it should idealy be (for most designs).

Besides these subject-related recommendations, I've also got an idea to improve working for this project from a practical point of view. For certain CFD issues it is really an outcome when it is possible to discuss with others and show them things on the computer. During this internship, communication with supervisor Steven Schmied was possible over a distance through telephone and e-mail. It helped a lot, but showing and discussing things in reality definitely circumvents making small mistakes and helps to define possible solutions. Having a colleague around with some CFD knowledge would help, whenever it would be possible.

12 Acknowledgements

These three months have been an amazing time for me. There is some people I would really like to thank for supporting me with the project, as well as making sure that I was having a good time at the factory and in Australia.

First of all, Steven, the biggest thanks are for you. Your fast replies on my questions and your frequent suggestion 'dumps' - always covered in a sauce of enthusiasm and unmistakable love for surfing - were helping me to resolve or circumvent the never ending stream of bigger and smaller troubles occurring and kept me motivated even after obtaining disappointing results. Together with Kolona you seem to have the ability - or at least try - to turn dreams into action. Although there might still be some serious obstacles on the way, you both appear to don't waste time and just start. I appreciate that mindset.
Kolona, when we went surfing the first day together with Steven, I directly felt very welcome. It was great that you were around to always make sure I was doing fine and that I was able to borrow your bicycle, giving me the necessary Dutchie flexibility. Also, I'm sure I'll never forget you and Steven when I'm riding a wave on a beautiful board.
Mike, thanks a lot for hosting me at your factory. You are a really great guy. Luke, I liked to arrive in the morning and hear the music was already playing loud. Together with your ever smiling grin it always made a good start of the day. Alex, cool to have you here just before I left. Tom, you were fantastic company upstairs. I liked our interesting discussions and ideas about life.
And last but not least, professor Hoeijmakers, thanks a lot for bringing me in contact with Steven and making this internship possible. Without your effort I would not have had such a great experience. I would also like to thank you for taking the time to assess this report.

Hope to see all of you again one day!

Jason

References

[1] AMC towing tank. http://www.amc.edu.au/facilities/towing-tank. Accessed: January 3, 2018.

[2] Autodesk CFD, error: "Analysis has stopped because the solver has exited unexpectedly" in Autodesk CFD. https://knowledge.autodesk.com/support/cfd/troubleshooting/caas/sfdcarticles/sfdcarticles/Error-Message-Analysis-has-stopped-because-the-Solver-has-exited-Unexpectedly.html. Accessed: November 20, 2017.

[3] Boundary layer theory - flow past a flat plate. https://ecourses.ou.edu/cgi-bin/eBook.cgi?doc=&topic=fl&chap_sec=09.3&page=theory. Accessed: January 30, 2018.

[4] CFD-online: Skin friction coefficient. https://www.cfd-online.com/Wiki/Skin_friction_coefficient. Accessed: January 24, 2018.

[5] CFD-online: Y plus wall distance estimation. https://www.cfd-online.com/Wiki/Y_plus_wall_distance_estimation. Accessed: January 24, 2018.

[6] Wolframalpha gravitational acceleration, Launceston Australia. http://www.wolframalpha.com/widget/widgetPopup.jsp?p=v&id=d34e8683df527e3555153d979bcda9cf&title=Gravitational%20Fields&theme=red&i0=Launceston%2C%20Australia&podSelect=&includepodid=Input&includepodid=GravitationalFieldStrength&includepodid=LocationOfTheGeographicalItemOfInterest. Accessed: January 30, 2018.

[7] leap Australia CFD, tips & tricks: Convergence and mesh independence study. https://www.computationalfluiddynamics.com.au/convergence-and-mesh-independent-study/, January 2012. Accessed: January 9, 2018.

[8] leap Australia CFD, tips & tricks: Estimating the first cell height for correct y+. https://www.computationalfluiddynamics.com.au/tips-tricks-cfd-estimate-first-cell-height/, July 2013. Accessed: January 24, 2018.

[9] leap Australia CFD, turbulence part 3 - selection of wall functions and y+ to best capture the turbulent boundary layer. https://www.computationalfluiddynamics.com.au/turbulence-part-3-selection-of-wall-functions-and-y-to-best-capture-the-turbulent-boundary-laye April 2013. Accessed: January 24, 2018.

[10] leap Australia CFD, turbulence part 4 - reviewing how well you have resolved the boundary layer. https://www.computationalfluiddynamics.com.au/tips-tricks-turbulence-part-4-reviewing-how-well-you-have-resolved-the-boundary-layer/, May 2013. Accessed: January 24, 2018.

[11] Autodesk CFD: Advection schemes. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/SimCFD-UsersGuide/files/GUID-F691B334-CCE2-47E9-B6C4-21666712C163-htm.html, February 2015. Accessed: January 10, 2018.

[12] Autodesk CFD: Diagnostics. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/SimCFD-UsersGuide/files/GUID-C0D12CCF-704A-4402-88E3-66DD86E274B9-htm.html, February 2015. Accessed: January 9, 2018.

[13] Autodesk CFD: External incompressible flow. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2014/ENU/SimCFD/files/GUID-4EED9E6E-A694-4505-9502-8D9CC42A5EC2-htm.html, December 2015. Accessed: January 8, 2018.

[14] Autodesk CFD: How to set up a free surface analysis of a boat. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/sfdcarticles/sfdcarticles/

How-to-set-up-a-free-surface-analysis-of-a-boat.html, September 2015. Accessed: January 9, 2018.

[15] Autodesk CFD: Manual mesh sizing. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/SimCFD-UsersGuide/files/GUID-DC8D7EDA-5E18-4088-93AB-3BB517D6469C-htm.html, February 2015. Accessed: January 9, 2018.

[16] Autodesk CFD: Transient flows. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2014/ENU/SimCFD/files/GUID-384EC529-309C-426C-B036-93ADDB842F0E-htm.html, December 2015. Accessed: January 10, 2018.

[17] Autodesk CFD: Transient parameters. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2014/ENU/SimCFD/files/GUID-7BE54C33-D7CE-4F95-81B0-FD3B5517E9B3-htm.html, December 2015. Accessed: January 10, 2018.

[18] Autodesk CFD: Two equation turbulence models (tke & ted). https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2016/ENU/SimCFD-Learning/files/GUID-61C4EB55-362C-48A0-8B22-20F9148D190D-htm.html, August 2015. Accessed: January 19, 2018.

[19] Autodesk CFD: Turbulence. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2016/ENU/SimCFD-UsersGuide/files/GUID-E9E8ACA1-8D49-4A49-8A35-52DB1A2C3E5F-htm.html, March 2016. Accessed: January 10, 2018.

[20] Autodesk CFD: Automatic convergence assessment. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/SimCFD-UsersGuide/files/GUID-477E1D94-2D6B-4C25-A5C6-6842532AF911-htm.html, February 2017. Accessed: January 10, 2018.

[21] Autodesk CFD: Convergence plot. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/SimCFD-Learning/files/GUID-9169AD51-4916-4C43-80D0-C78213DCEC41-htm.html, February 2017. Accessed: January 10, 2018.

[22] Autodesk CFD verification: Drag force on a cylinder. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/SimCFD-Learning/files/GUID-C8165A4A-97E0-43BC-90BC-A179DDE9FB6A-htm.html, February 2017. Accessed: January 3, 2018.

[23] Autodesk CFD: Wall layers. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/SimCFD-UsersGuide/files/GUID-F9C4DDB4-8111-4F25-8EDE-D7C38B3BAD99-htm.html, February 2017. Accessed: January 9, 2018.

[24] Autodesk CFD: What happens with solid volumes in an external flow simulation? https://knowledge.autodesk.com/support/cfd/learn-explore/caas/discussion/t5/CFD-Forum/What-happens-with-solid-volumes-in-an-external-flow-simulatiobn-/td-p/7335562.html, August 2017. Accessed: January 8, 2018.

[25] Royce Abel. Autodesk CFD video: Understanding the turbulence models available in Autodesk Simulation CFD. https://www.youtube.com/watch?v=Yf2iVABc8cg, June 2014. Accessed: January 9, 2018.

[26] Terry Hendricks. Surfboard hydrodynamics, part 1: Drag. 1969.

[27] Matt Bemis Marwan Azzam. Autodesk CFD video: Free surface best practices. https://knowledge.autodesk.com/support/cfd/learn-explore/caas/video/youtube/watch-v-rNTcjBrmwVY.html, February 2017. Accessed: January 9, 2018.

[28] Michael Paine. Hydrodynamics of surfboards. 1974.

[29] Sam Schumacher. Empirical and experimental investigation of surfboard hydrodynamics. Technical report, Sydney University. School of Aerospace, Mechanical and Mechatronic Engineering, 2011.

[30] Thomas A. Shapiro. The effect of surface roughness on hydrodynamic drag and turbulence. May 2004.

Appendices

A Project Evaluation

Time has been flying for me the past three months. I think the project is much further now than when I started. I learned a lot, not only skills, but also lessons about what I like. Here I would like to discuss these experiences.

A.1 The employer

The project is carried out for Steven Schmied and Kolona Mohenao for Peau Pty Ltd (the employer) in order to eventually develop a Surfboard Performance Analyser, possibly implementable into the AKU shaper software. The activities were conducted at the AKU shaper factory in Tweed Heads. There were four colleagues, of which two were building surfboard shaper machines in the workshop and two were writing surfboard design software. I was sitting upstairs - in a quite messy but cool place with surfboards standing everywhere - with the software guys that were about my age. The employers visited me when I arrived. Steven (the external supervisor) then helped me to get started and gave support from a distance during the rest of the time. Kolona dropped in every now and then to make sure everything was going alright.

A.2 Working at the factory

I had great flexibility in working time. This allowed me to start and leave early, or the other way around. Also, I came in on a weekend day a few times to compensate for the occasional afternoon that I quit earlier when the surf was good. It was great to be trusted that I would manage my time by myself, be independent and make sure work was progressing.
The factory was situated one kilometer from the beach, which gave me the opportunity to enjoy the sea and surf after work, or as a lunch break. I was even able to borrow a few surfboards.
No dress codes applied. I found it comfortable to be able to wear thongs (flip flops), board shorts and t-shirts to work.
My colleagues were great company, it was always nice to have a chat with all of them.
I also liked it to see the machines be built and tested.

A.3 Experiences during the project

I think the process of this internship has been fairly characteristic for three month internships, as far a I've heard from fellow students. During the first three weeks or so, I was busy getting properly started. Defining the problem and activities, gaining background knowledge and getting used to all the new things (software). In the middle of the internship, a lot of progress was made. Many simulations were run. In the last few weeks, simulations had already to be finalized and it was noticed that still a lot of results needed to be properly analyzed and documented. The report had to be written. Also, everything was to be left behind properly so that a successor could start where I finished. Time went fast and contributing significantly to the employer's needs only happens during the middle, as well as through the report.

I really like to have a well defined plan before getting into action. I don't necessarily have to follow this plan, but it gives me something to hold onto. For this internship, I didn't find the assignment too specified. Therefore I needed to define what was to be done by myself. At first I tried to stick to the original plan of performing a parameter study, but I soon ran into problems of which I realized they had to be resolved first. This took longer than expected and therefore the plan had to be changed several times. Following a prescribed planning and to-do-list would have been much easier, although much less challenging. Eventually, I'm happy with the followed path and am sure it forms a good step in the direction of the final goal of the project, the AKU Shaper Performance Analyser.

I had a large amount of independence. Although suggestions of things to do were made by my supervisor, I had still had to decide a lot of things myself. I noticed that I sometimes found it hard to make decisions on my own when I'm unable to talk to someone. I rather discuss things with others first to structure my thoughts.

Every week I had to hand in a progress report in the form of a powerpoint presentation, mentioning my goals of that week, the conducted activities and the aims for the next week. This was a good tool to evaluate my progress and efficiency, as well as to communicate with Steven.
But supervision from a distance was not ideal. Although having frequent contact over phone and e-mail, sometimes it's just better to talk and show things in real life. Especially when the small language barrier makes communication over distance a tiny bit more difficult and when questions are about subtle or very specific cases.

I experienced that doing CFD analyses is complicated, especially for the first time. I did not have anyone experienced with CFD at the factory. Steven is experienced, but not that much with the Autodesk CFD software. I learned a great lot and am sure that in the future I won't make the same mistakes again.

Sometimes the results were disappointing. No trends - that obviously had to be there - were visible and it just looked as if results were just random. I found it hard to report on these trial-and-error activities that cost a lot of time, but were not generating much visible progress.

A.4 Unexpected delays

I am usually fairly positive, but maybe a bit naive, when it comes to estimating times for a planning. Often, things were more complicated or took more time than expected, at least during this project. Characteristic are my thoughts I had the first days.

The first day, Steven showed me his AutoDesk CFD model of a partially submerged hull shape linearly moving through water. On the second day, while being on my own, I thought it would be relatively easy to make a similar model in which the hull was replaced with the surfboard. Even though I knew modifications had to be made (domain, orientation, mesh, other settings), I thought I would be able to find out how within a few days. That should give us drag values differing maybe 10-20 percent from the experiments within a week, so with a few more adjustments we would reach a level in which the error could solely be explained by the fundamental differences between numerical solutions and real life. That would allow a lot of time to conduct interesting surfboard parametric sensitivity analyses. Unfortunately, I couldnt have been more wrong. Already that second day I noticed that AutoDesk CFD simply not accepted the file type of the board design. On the way, a lot more problems revealed themselves. Almost everything turned out to be more complicated and cost more time than originally presumed.

Problems that took unexpectedly much time to resolve but of which you dont see much result:

- Reduce the mesh density of the board.
- Get the board into AutoDesk CFD without the software getting to run too slowly.
- Getting AutoDesk CFD installed with proper licensing.
- Find out how to circumvent the Blue Screen of Death error, causing the pc to crash. Error could not be solved, but solution was to not solve on the pc, but in the Autodesk cloud only. This was unfortunately slower.
- Finding the reason for the simulations to stop with the error 'Simulation has stopped because the solver excited unexpectedly'. It was probably due to a combination of wall layer settings, turbulence model and boundary conditions.
- Discover that the boundary conditions as first suggested by the supervisor were wrong, having caused many simulation failures in the first few weeks.
- Trying to let the convergence plots flatten, but finding out that they do not.

- Trying to obtain mesh independent results, but finally realizing computer power was insufficient to run finer meshes (although other mesh settings were also still incorrect).
- Performing the final runs for the pin tail board that took much longer than the square tail board. Eventually, they did not even finish and were not useful for the report.

A.5 Personal

Besides the skills that I gained and improved during this internship (CFD, meshing, hydrodynamics, boundary layer, structured planning, etc.), I also learned a few lessons about what I like and want to do, and what not.

Earlier, during my bachelor - while doing a lot of assignments in groups of about eight people - I was looking forward to do a project on my own and see what I could achieve. Now that I've done that, I realize I actually prefer to work in a small group (2-4), just like I did for most of my courses in the master. This gives the opportunity to discuss things, which results in making better decisions and avoiding simple mistakes. Also, it is just more fun to do something together.

With CFD simulations, as well as for documenting, a lot of time is spend on the computer, which makes me feel mushy after a while. This is probably characteristic for many academic engineering jobs nowadays. But I might actually prefer to do something in which I can vary the computer time with something else.

The AKU shaper factory was a small place with only a few colleagues. I liked the personal feel of it and felt I was part of the team, although doing things independently. It's good to remember this preference for smaller companies, although I'd like to try and work for a bigger company one day also, maybe for my graduation project.

B Run Sheet

The next five pages contain a for this report modified version of the run sheet. The first documented simulation was done end of November and the last end of January. The runs are not listed in chronological order. Performed dates were not always recorded, so omitted.
For some runs five drag measurements were done. These average numerical drag forces are underlined. If the text is green, it means the sample standard deviation is less than 1, otherwise the text is red.
When a cell contains 'off 500' or a similar expression, this means Automatic Convergence Assessment was disabled and 500 time steps were run.

This is a modified version of the original Run Sheet. The original version contained more information that is now ommitted for clearity.

g [m/s^2] = 9.804

	SETUP GENERAL		SIMULATION INFO		SETUP CFD				SOLVER SETTINGS						RESULTS EXPERIMENTAL	RESULTS NUMERICAL							DRAG MEASUREMENTS					
Run	Board	speed	in file	Continued from	domain	mesh	Notes		timestep size	iterations	ADV	TURB	Convergence assessment	Wall layer	Drag	Drag	converged at time step	Notes			Computation time		1	2	3	4	5	SSTD
	-	m/s	<filename>.cfdst		-	-			s	#	-	-	-	def/adv	N	N	#				h							
PART 1																												
Many more runs were performed for Part 1, but these were not worth to keep track of, since it was really done on a trial and error basis and are not refered to in the report.																												
V1_30P00D00M00_1	P00	3.04	V1.cfdst		D00	M00			0.01/0.05	10	ADV5	k-omega	almost tigh	default	48.43176		not converged yet	Had to run 0.01 first, then could do 0.05. Mesh views showed where meshing was effective, so possible to update refinement regions										
V1_45P00D00M00_1	P00	4.506	V1.cfdst		D00	M00			0.01/0.05	10	ADV5	k-omega	default	default														
Preliminary Mesh study1																												
V1_30P00D00M34_1	P00	3.04	Analysis1.cfdst		D00	M34			0.01	3	ADV1	SST k-omeg	default	default	48.43176			Wave formation at the back of the domain for all mesh study1 cases										
V1_30P00D00M34b_1	P00	3.04	Analysis1.cfdst		D00	M34b			0.01	3	ADV1	SST k-omeg	default	default	48.43176													
V1_30P00D00M33_1	P00	3.04	Analysis1.cfdst		D00	M33			0.01	3	ADV1	SST k-omeg	default	default	48.43176													
V1_30P00D00M3b_1	P00	3.04	Analysis1.cfdst		D00	M33b			0.01	3	ADV1	SST k-omeg	default	default	48.43176	48.5		only one sample										
V1_30P00D00M3b_2	P00	3.04	Analysis1.cfdst	V1_30P00D00M3b_1	D00	M33b			0.01	3	ADV1	SST k-omeg	tight	default	48.43176													
V1_30P00D00M32_1	P00	3.04	Analysis1.cfdst		D00	M32			0.01	3	ADV1	SST k-omeg	default	default	48.43176													
V1_30P00D00M31_1	P00	3.04	Analysis1.cfdst		D00	M31			0.01	3	ADV1	SST k-omeg	default	default	48.43176													
V1_30P00D00M31b_1	P00	3.04	Analysis1.cfdst		D00	M31b			0.01	3	ADV1	SST k-omeg	default	default	48.43176													
V1_30P00D00M33ba_1	P00	3.04	Analysis1.cfdst		D00	M33ba			0.01	3	ADV1	SST k-omeg	default	default	48.43176													
V1_30P00D10M43_1	P00	3.04	D10.cfdst		D10	M43	Domain D10 is smaller and uses slip conditions		0.01	3	ADV1	SST k-omeg	default	default	48.43176		50 time steps, not converged yet											
V1_30P00D10M43_2	P00	3.04	D10.cfdst	V1_30P00D10M43_1	D10	M43			0.01	3	ADV1	SST k-omeg	default	default	48.43176		400 time steps, not converged yet											
Preliminary mesh investigation																												
V1_30P00D10MA_1	P00	3.04	D10 2.cfdst		D10	MA			0.01	3	ADV1	SST k-omeg	default	default	48.43176	28.67	298											
V1_30P00D10MA_2	P00	3.04	D10 2.cfdst	V1_30P00D10MA_1	D10	MA			0.05	3	ADV1	SST k-omeg	off	default	48.43176	30.3	off 2298	Only vz (vy slightly) changing, rest converged. Some noflow water on top surface. Channels in free surface. Blocky wake. Mesh needs to be improved	animation shows bow wave breaking and building. Probably reason for non converged vz vy.		about 3							
V1_30P00D10MA_3	P00	3.04	D10 2.cfdst	V1_30P00D10MA_2	D10	MA	Still called run2 in file	Measuring run	0.05	3	ADV1	SST k-omeg	off	default	48.43176	30.394	off +10						31.06	30.53	30.3	29.66	30.42	0.502573
V1_30P00D10MB_1	P00	3.04	D10 2.cfdst	V1_30P00D10MA_1	D10	MB	New mesh		0.01	3	ADV1	SST k-omeg	off	default	48.43176		off 600	Not really converged	Channels in free surface are larger. Wake less blocky.		about 4.5							
V1_30P00D10MB_2	P00	3.04	D10 2.cfdst	V1_30P00D10MB_1	D10	MB	Still called 30MB_1 in file		0.05	3	ADV1	SST k-omeg	off	default	48.43176		off 200	CFD crashes when trying to see results										
V1_35P00D10MA_1	P00	3.543	D10 2.cfdst	V1_30P00D10MA_1	D10	MA	Increased velocity		0.01	3	ADV1	SST k-omeg	off	default	69.667224	37.77	off 898	same remarks as for V1_30P00D10MA_2	higher bow wave									
V1_35P00D10MA_2	P00	3.543	D10 2.cfdst	V1_35P00D10MA_1	D10	MA	Measurement run		0.05	3	ADV1	SST k-omeg	off	default	69.667224	37.42	Off +100	same remarks as for V1_30P00D10MA_2	higher bow wave				38.02	36.7	37.21	37.4	37.77	0.511224
V1_35P00D10MB_1	P00	3.543	D10 2.cfdst	V1_30P00D10MA_1	D10	MB	New mesh		0.01	3	ADV1	SST k-omeg	off	default	69.667224	48.85	off 600	not really converged yet.	Channels, some weird peaks.	Blocky bow wave, about as high as for MA								
V1_35P00D10MB_2	P00	3.543	D10 2.cfdst	V1_30P00D10MA_1	D10	MB	Still called 35MB_1 in file		0.05	3	ADV1	SST k-omeg	off	default	69.667224	49.34	off +400	maybe not really converged, but drag values do not change much anymore										
V1_40P00D10MA_1	P00	4.044	D10 2.cfdst	V1_30P00D10MA_1	D10	MA			0.01	3	ADV1	SST k-omeg	off	default	87.64776	45.7	off 500	maybe not really converged, but drag values do not change much anymore										
V1_35P00D10MC_1	P00	3.543	D10 2.cfdst	V1_35P00D10MB_1	D10	MC	New mesh		0.005	3	ADV1	SST k-omeg	off	default	69.667224	37.41	off 20	not converged	still very blocky		about 6h							
V1_35P00D10MC_2	P00	3.543	D10 2.cfdst	V1_35P00D10MC_1	D10	MC			0.01	3	ADV1	SST k-omeg	off	default	69.667224		off 600	failed										
V1_35P00D10MC_3	P00	3.543	D10 2.cfdst	V1_35P00D10MB_1	D10	MC	continue from t=20		0.005	3	ADV1	SST k-omeg	off	default	69.667224		off 600	failed										
V1_45S00D20MK_1	S00	4.55	Mesh S00D20 2.cfdst		D20	ML			0.01	10	ADV1	SST k-omeg	tight	default	104.2988736	49.4	665	almost no wake	no saved time steps...	run for measurin g								
V1_45S00D20MK_2	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MK_1	D20	ML	measure		0.01	10	ADV1	SST k-omeg	off 500	default	104.2988736	48.34	off 500						49.25	47.73	49.15	49.03	46.54	1.180297
V1_45S00D20ML_1	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MK_2	D20	ML			0.01	10	ADV1	SST k-omeg	off 200	default	104.2988736	85.25	off 200	need to converge more										
V1_45S00D20ML_2	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20ML_1	D20	ML			0.01	10	ADV1	SST k-omeg	tight	default	104.2988736	95.43	51	good enough for cloning	do more iterations for measuring									
V1_45S00D20ML_3	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20ML_2	D20	ML			0.01	10	ADV1	SST k-omeg	off 500	default	104.2988736	76.77	off 500	save settings incorrect so no intermediate time steps	nice 'converged'. Run more for measureing		about 3,5h							

V1_45S00D20ML_4	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20ML_3	D20	ML	measure		0.01	10	ADV1	SST k-ome	off 100	default	104.2988736	80.138	off 100	only vz and vy bit variation				851timeste	79.79	88.04	75.92	82.67	74.27	5.504514
V1_45S00D20ML_5	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20ML_4	D20	ML	measure	let converge more	0.05	10	ADV1	SST k-ome	off 1000	default	104.2988736	78.27	off 1000						81.84	69.97	82.07	79.17	78.3	4.921529
V1_45S00D20MM_1	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20ML_1	D20	MM	new mesh		0.01	10	ADV1	SST k-ome	off 500	default	104.2988736	65.24	off 500	good enough for cloning	no strange looking things	save settings were incorrect, so only last time step	about 5h							
V1_45S00D20MM_2	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MM_1	D20	MM	measure		0.01	10	ADV1	SST k-ome	off 100	default	104.2988736		weird unphysical result. Have to redo											
V1_45S00D20MM_3	S00	4.55	Mesh S00D20 2.cfdst		D20	MM	start again		0.01	10	ADV1	SST k-ome	off 1000	default	104.2988736	70.598	off 1000						74.34	69.03	67.03	71.61	70.98	2.755081
V1_45S00D20MM_4	S00	4.55	Mesh S00D20 2.cfdst		D20	MM	continue		0.01	10	ADV1	SST k-ome	off 500	default	104.2988736	69.564	off 500	graphs show only small variations					69.02	69.17	65.96	70.06	73.61	2.741173
V1_45S00D20MN_1	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MM_1	D20	MN	new mesh		0.01	10	ADV1	SST k-ome	tight	default	104.2988736		failed											
V1_45S00D20MN_2	S00	4.55	Mesh S00D20 2.cfdst		D20	MN	continue from t251		0.005	10	ADV1	SST k-ome	tight	default	104.2988736		failed											
V1_45S00D20MN_3	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20ML_4	D20	MN	new mesh	cloned from ML_4	0.005	10	ADV1	SST k-ome	tight	default	104.2988736													
V1_45S00D20MN_4	S00	4.55	Mesh S00D20 2.cfdst		D20	MN	start from 0		0.001	10	ADV1	SST k-ome	tight	default	104.2988736	100.35	362	free surface is not stable yet. Wave coming in			about 22h							
V1_45S00D20MN_5	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MN_4	D20	MN	continue		0.005	10	ADV1	SST k-ome	off 1000	default	104.2988736	58.09	off 1000	mainly variation in TED			about 31h		58.99	58.84	58.16	56.57	57.89	0.965376
V1_45S00D20MO_1	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MM_3	D20	MO	MO from MM		0.005	10	ADV1	SST k-ome	off 1000	default	104.2988736		off 1000	failed										
V1_45S00D20MO_2	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MN_4	D20	MO	MO from MN		0.001	10	ADV1	SST k-ome	tight	default	104.2988736	48.63	398	free surface is not stable yet. Wave coming in			about 73h							
V1_45S00D20MO_3	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MO_2	D20	MO	continue		0.005	10	ADV1	SST k-ome	off 1000	default	104.2988736	60.94667	total 812	saved time steps too far apart, also some in non-converged part. Use only last three saved time steps					64.5	61.91	56.43			4.120344
V1_45S00D20MO_7	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MO_3	D20	MO	continue	bigger time step, see if works	0.05	10	ADV1	SST k-ome	off 500	default	104.2988736		total 1235	job exceeded time limit	graphs look okay, but not long enough to be certain	results not accessabl e								
V1_45S00D20MO_9	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MO_7	D20	MO	can only continue from t812	smaller timestep for saving	0.005	10	ADV1	SST k-ome	off 320	default	104.2988736	65.754	total 1452	not converged					67.7	65.16	64.82	65.11	65.98	1.170248
V1_45S00D20MO_4	S00	4.55	Mesh S00D20 2.cfdst		D20	MO	start clean		0.005	10	ADV1	SST k-ome	tight	default	104.2988736	47.39	418	free surface is not stable yet. Wave coming in			about 73h							
V1_45S00D20MO_5	S00	4.55	Mesh S00D20 2.cfdst	V1_45S00D20MO_4	D20	MO	continue		0.005	10	ADV1	SST k-ome	off 1000	default	104.2988736	54.94	total 853	vz varying	job exceeded time limit	saved time steps too far apart, also some in non-converged part			63.49	56	57.34	49.89	47.98	6.203431
V1_45S00D20MO_6	S00	4.55	Mesh S00D20 2.cfdst	55V1_45S00D20MO_5	D20	MO	continue	bigger time step, see if works	0.05	10	ADV1	SST k-ome	off 500	default	104.2988736	66.84	total 1274	job exceeded time limit	graphs look okay, but not long enough to be certain	not enough time steps saved	about 73h		67.22	66.46				0.537401
V1_45S00D20MO_8	S00	4.55	Mesh S00D20 2.cfdst	5V1_45S00D20MO_6	D20	MO	continue	smaller timestep for saving	0.005	10	ADV1	SST k-ome	off 320	default	104.2988736	69.066	off 320		begins to converge				69.12	70.4	70.09	67.94	67.78	1.199241
V1_45S00D20MO_10	S00	4.55	Mesh S00D20 2.cfdst	5V1_45S00D20MO_8	D20	MO	continue		0.005	10	ADV1	SST k-ome	off 320	default	104.2988736	68.346	off 320						70.53	65.9	72.55	63.63	69.12	3.580018
D40																												
MNc_1	S00	4.55	D40 meshes.cfdst		D40	MNc			0.005	10	ADV5	SST k-ome	off 2000	advanced	104.2988736	85.83	stopped at 1081	job exceeded time limit	variation in vz and TKE				87.24	84.6	85.75	85.82	85.74	0.937497
MNc_1	S00	4.55	D40 meshes.cfdst		D40	MNc	continue		0.005	10	ADV5	SST k-ome	off 919	advanced	104.2988736	85.742	total 2000	average values have flattened, except vz	water remains in wall layer on deck of board				85.81	85.15	86.62	85.37	85.76	0.56229
MOc_1																	stopped at t46	job exceeded time limit	far from converged	26M elements								
Suppression																												
V1_30P00D10MB_1polsup	P00	3.04	Suppression.cfdst		D10	MB	polystyreen suppressed		0.01	3	ADV1	SST k-ome	off	default	48.43176		failed, had to restart											
V1_30P00D10MB_2polsup	P00	3.04	Suppression.cfdst	V1_30P00D10MB_1pols	D10	MB	polystyreen suppressed	restart	0.01	3	ADV1	SST k-ome	off	default	48.43176	44.39	off 1500						41.81	41.72	43.31	45.32	49.79	3.35323
V1_30P00D10MB_3polsup	P00	3.04	Suppression.cfdst	V1_30P00D10MB_2pols	D10	MB	polystyreen suppressed	continue	0.01	3	ADV1	SST k-ome	off	default	48.43176	49.79	off 1000											
V1_30P00D10MB_4polsup	P00	3.04	Suppression.cfdst	V1_30P00D10MB_3pols	D10	MB	polystyreen suppressed	measurement run for use in submergence	0.01	3	ADV1	SST k-ome	off	default	48.43176	43.378	off 1000						45.41	41.88	43.17	43.43	43	1.280808
V1_30P00D10MB_1polun	P00	3.04	Suppression.cfdst		D10	MB	polystyreen unsuppressed		0.01	3	ADV1	SST k-ome	off	default	48.43176	41.95	off 1500	almost converged, do extra run for checking/measuring	area of board is higher, why?? Solved									
V1_30P00D10MB_2polun	P00	3.04	Suppression.cfdst	V1_30P00D10MB_1polu	D10	MB	polystyreen unsuppressed	mearure/check	0.01	3	ADV1	SST k-ome	off	default	48.43176		off 1200											
V1_30P00D10MB_3polun	P00	3.04	Suppression.cfdst	V1_30P00D10MB_2polu	D10	MB	polystyreen unsuppressed	mearure/check	0.01	3	ADV1	SST k-ome	off	default	48.43176	42.586	off 1000						41.58	41.06	43.02	43.06	44.21	1.264033

V1_30P00D10MB_1stesup	P00	3.04	Suppression.cfdst		D10	MB	steel suppressed		0.01	3	ADV1	SST k-ome	off	default	48.43176	44.55	off 1500	almost converged, extra run for measuring/check										
V1_30P00D10MB_2stesup	P00	3.04	Suppression.cfdst	V1_30P00D10MB_1stes	D10	MB	steel suppressed	measure/check	0.01	3	ADV1	SST k-ome	off	default	48.43176	43.304	off 200						39.47	43.55	44.16	44.99	44.35	2.203946
V1_30P00D10MB_3stesup	P00	3.04	Suppression.cfdst	V1_30P00D10MB_2stes	D10	MB	steel suppressed	measure/check	0.01	3	ADV1	SST k-ome	off	default	48.43176	44.126	off 1000						45.77	40.93	43.59	43.94	46.4	2.145351
V1_30P00D10MB_1steun	P00	3.04	Suppression.cfdst		D10	MB	steel unsuppressed		0.01	3	ADV1	SST k-ome	off	default	48.43176	36.09	off 1500	almost converged, extra run for measuring/check										
V1_30P00D10MB_2steun	P00	3.04	Suppression.cfdst	V1_30P00D10MB_1steu	D10	MB	steel unsuppressed	measure/check	0.01	3	ADV1	SST k-ome	off	default	48.43176	36.152	off 200						36.07	36.13	36.02	35.1	37.44	0.835207
V1_30P00D10MB_3steun	P00	3.04	Suppression.cfdst	V1_30P00D10MB_2steu	D10	MB	steel unsuppressed	measure/check	0.01	3	ADV1	SST k-ome	off	default	48.43176		off 1000	not happy with graph yet	try 10 iterations per time step									
V1_30P00D10MB_4steun	P00	3.04	Suppression.cfdst	V1_30P00D10MB_3steu	D10	MB	steel unsuppressed	measure/check	0.01	10	ADV1	SST k-ome	off	default	48.43176	34.064	off 100						33.96	34.47	34.69	32.87	34.33	0.718317
Submergence																												
V1_30P00D10MB_+3_1	P00	3.04	D10 submergence.cfdst		D10	MB	+3cm	measuring	0.01	3	ADV1	SST k-ome	off	default		52.79	off 3000	looks relatively well converged					53.62	51.29	53.38	51.3	54.36	1.411737
V1_30P00D10MB_+2_1	P00	3.04	D10 submergence.cfdst		D10	MB	+2cm		0.01	3	ADV1	SST k-ome	off	default		44.59	off 1500	same MB problems for all these MB runs										
V1_30P00D10MB_+2_2	P00	3.04	D10 submergence.cfdst		D10	MB	+2cm	measuring	0.01	3	ADV1	SST k-ome	off	default		48.326	off 200						48.36	44.48	54.58	49.27	44.94	4.070059
V1_30P00D10MB_+2_3	P00	3.04	D10 submergence.cfdst		D10	MB	+2cm	measuring	0.01	3	ADV1	SST k-ome	off	default		49.25	off 1000	not happy with graph yet	try 10 iterations per time step									
V1_30P00D10MB_+2_4	P00	3.04	D10 submergence.cfdst		D10	MB	+2cm	measuring	0.01	10	ADV1	SST k-ome	off	default		49.69	off 400	large TED variation	total 13100 iterations				48.9	51.23	47.52	52.5	48.3	2.093609
V1_30P00D10MB_+1_1	P00	3.04	D10 submergence.cfdst		D10	MB	+1cm		0.01	3	ADV1	SST k-ome	off	default		38.5	off 1500											
V1_30P00D10MB_+1_2	P00	3.04	D10 submergence.cfds	V1_30P00D10MB_+1	D10	MB	+1cm	measuring	0.01	3	ADV1	SST k-ome	off	default			off 200	Not possible to asses result and restart…	Had to be started from beginning				21.44					
V1_30P00D10MB_+1_3	P00	3.04	D10 submergence.cfds	start again	D10	MB	+1cm	measuring	0.01	3	ADV1	SST k-ome	off	default		42.568	off 3000						41.3	42.4	42.9	42.72	43.52	0.817875
V1_30P00D10MB_-1_1	P00	3.04	D10 submergence.cfdst		D10	MB	-1cm		0.01	3	ADV1	SST k-ome	off	default			off 1500	failed. Also failed with dt=0.005 and dt=0.001										
V1_30P00D10MB_-1_2	P00	3.04	D10 submergence.cfdst		D10	MB	-1cm	removed small edges/surfaces	0.001	3	ADV1	SST k-ome	off	default			off 20	worked to start without failure										
V1_30P00D10MB_-1_3	P00	3.04	D10 submergence.cfds	V1_30P00D10MB_-1_2	D10	MB	-1cm		0.01	3	ADV1	SST k-ome	off	default		49.464	off 3000						49.35	49.95	47.97	50.87	49.18	1.065214
V1_30P00D10MB_-2_2	P00	3.04	D10 submergence.cfdst		D10	MB	-2cm		0.01	3	ADV1	SST k-ome	off	default			off 1500	failed. Also failed with dt=0.005 and dt=0.001										
V1_30P00D10MB_-2_2	P00	3.04	D10 submergence.cfdst		D10	MB	-2cm		0.001	3	ADV1	SST k-ome	off	default			off 20	failed										
V1_30P00D10MB_-3_1	P00	3.04	D10 submergence.cfdst		D10	MB	-3cm	measure	0.01	10	ADV1	SST k-ome	off	default		45.668	total 3200						47	45.93	48.43	44.13	42.85	2.223628
TurbADV																												
All the runs in this test series use 45S00D20MM																												
ADV1 SST-SAS	S00	4.55	TurbAdv.cfdst		D20	MM			0.005	10	ADV1	SST-k-ome	off 2000	default	104.2988736	71.218	off 2000	wake pattern barely changing	low frequency vairation of Vz and Vy, TKE increasing	drag value is comparab le to run V1_45S00 D20MM_ 3	25h		76.68	71.16	68.7	71.26	68.29	3.34485
ADV1 SST-SAS_2	S00	4.55	TurbAdv.cfdst		D20	MM	continue, to see if better converge for more steps		0.005	10	ADV1	SST-k-ome	off 2000	default	104.2988736	70.128	off 2000	wake pattern barely changing	low frequency vairation of Vz and Vy, TKE increasing	drag value is comparab le to run V1_45S00 D20MM_ 3	25h		71.41	68.68	69.38	73.15	68.02	2.113994
ADV5 SST-SAS	S00	4.55	TurbAdv.cfdst		D20	MM			0.005	10	ADV5	SST-k-ome	off 2000	default	104.2988736	77.366	off 2000	wake pattern barely changing	low frequency vairation of Vz and Vy, TKE increasing		18h		76.52	82.23	82.34	73.61	72.13	4.760203
ADV5 SST-SAS_2	S00	4.55	TurbAdv.cfdst		D20	MM	continue, to see if better converge for more steps		0.005	10	ADV5	SST-k-ome	off 2000	default	104.2988736	76.24	off 2000				23h		78.07	74.22	80.9	70.97	77.04	3.793343
ADV1 Lam	S00	4.55	TurbAdv.cfdst		D20	MM			0.005	10	ADV1	Laminar	off 2000	default	104.2988736	22.43	off 2000				13h		23.14	21.44	22.29	22.14	23.14	0.723187
ADV5 Lam	S00	4.55	TurbAdv.cfdst		D20	MM			0.005	10	ADV5	Laminar	off 2000	default	104.2988736	23.822	off 2000				18h		24.56	23.31	24.84	23.04	23.36	0.816713
ADV5 SST-SAS 10layers	S00	4.55	TurbAdv.cfdst		D20	MM			0.005	10	ADV5	SST-k-ome	off 2000	advanced	104.2988736	38.93	off 2000	constant variation in Pres, Vy, Vz	only last time step accesible		51h							
ADV5 SST-SAS 10layers	S00	4.55	TurbAdv.cfdst		D20	MM	continue		0.005	10	ADV5	SST-k-ome	off 100	advanced	104.2988736	48.25	off 100	constant variation in Pres, Vy, Vz	only last time step accesible	water flows off deck nicely	51h							
ADV5 SST-SAS 10layers	S00	4.55	TurbAdv.cfdst		D20	MM	restart from 0		0.005	10	ADV5	SST-k-ome	off 2000	advanced	104.2988736	43.576					54h		43.21	44.78	43.12	43.93	42.84	0.783983
ADV1 SST-SAS 10layers	S00	4.55	TurbAdv.cfdst		D20	MM			0.005	10	ADV1	SST-k-ome	off 2000	advanced	104.2988736	44.5	off 2000	large high frequency variation in Vz	only last time step accesible		49h							
ADV1 SST-SAS 10layers	S00	4.55	TurbAdv.cfdst		D20	MM	restart from 0		0.005	10	ADV1	SST-k-ome	off 2000	advanced	104.2988736	43.134	off 2000				52h		43.02	45.2	42.72	41.59	43.14	1.307356
ADV1 k-epsilon	S00	4.55	TurbAdv.cfdst		D20	MM			0.005	10	ADV1	k-epsilon	off 2000	default	104.2988736	67.84	off 2000	two values of +125N were disregarded, so bit unfair			18h		68.65	68.93	66.37	70.9	64.35	2.527291
ADV5 k-epsilon	S00	4.55	TurbAdv.cfdst		D20	MM			0.005	10	ADV5	k-epsilon	off 2000	default	104.2988736	35.306	off 2000				23h		37.72	36.08	33.66	35.95	33.12	1.892691
V1_30P00D00M3b_1	P00	3.04	Analysis1.cfdst		D00	M33b			0.01	3	ADV1	k-omega S	default	default	48.43176													
Domain and Board																												
All the runs in this test series use 45MNb																												
S00D20_1	S00	4.55	DomBoard.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 1000	default	104.2988736	68.83	off 1000	only last time step accessible, wrong save settings										
S00D20_2	S00	4.55	DomBoard.cfdst	S00D20_1	D20	MNb			0.005	10	ADV1	SST k-ome	off 100	default	104.2988736	66.17	off 100	still variation in vz	wake barely changes		4h		66.06	65.69	65.21	64.76	69.13	1.725819

S00D20_3	S00	4.55	DomBoard.cfdst	S00D20_2	D20	MNb	interesting to see if results are the same as other run that ran 2000 time steps		0.005	10	ADV1	SST k-ome	off 900	default	104.2988736	63.786	off 900				24h		63.61	63.4	62.28	64.29	65.35	1.134473
S00D20_4	S00	4.55	DomBoard.cfdst	S00D20_3	D20	MNb	see if smaller dt works		0.05	10	ADV1	SST k-ome	off 900	default	104.2988736	64.084	off 1000	big vz variance	larger dt seems to work				63.05	65.59	64.06	64.11	63.61	0.943811
S00D30_1	S00	4.55	DomBoard.cfdst		D30	MNb			0.005	10	ADV1	SST k-ome	off 1000	default	104.2988736		stopped at 349	job exceeded timelimit.	18M elements, too much	only last time step accessible , wrong save settings								
S00D30_2	S00	4.55	DomBoard.cfdst	S00D30_1	D30	MNb	continue from t349		0.005	10	ADV1	SST k-ome	off 700	default	104.2988736	52.55	stopped at 667	job exceeded time limit.	wake has not reached end of domain yet									
S00D30_3	S00	4.55	DomBoard.cfdst	S00D30_2	D30	MNb	continue from t667		0.005	10	ADV1	SST k-ome	off 400	default	104.2988736	30.06	stopped at 980	job exceeded time limit.	wake has not reached end of domain yet	average plotsbegin to flatten, except vz and TKE			26.21	23.15	27.07	36.43	37.44	6.452751
S00D30_4	S00	4.55	DomBoard.cfdst	S00D30_2	D30	MNb	continue		0.005	10	ADV1	SST k-ome	off 400	default	104.2988736	68.212	stopped at 1282	job exceeded time limit.	wake has not reached end of domain yet				103.82	107.06	45.33	38.36	46.49	34.14553
S10D20_1	S10	4.55	DomBoard.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 1000	default	104.2988736		off 1000	only last time step accessible, wrong save settings. Group not defined										
S10D20_2	S10	4.55	DomBoard.cfdst	S10D20_1	D20	MNb			0.005	10	ADV1	SST k-ome	off 100	default	104.2988736	83.348	off 100	still a lot of variation in graphs					75.09	76.46	73.65	96.57	94.97	11.39718
S10D20_3	S10	4.55	DomBoard.cfdst	S10D20_2	D20	MNb			0.005	10	ADV1	SST k-ome	off 900	default	104.2988736	64.348	off 900	very nicely converged, only minor variation					64.26	63.86	63.66	64.46	65.5	0.71744
S20D20_1	S20	4.55	DomBoard.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 1000	default	104.2988736		off 1000	only last time step accessible, wrong save settings. Group not defined	constant variation in TED	4.4M elements								
S20D20_2	S20	4.55	DomBoard.cfdst	S20D20_1	D20	MNb			0.005	10	ADV1	SST k-ome	off 100	default	104.2988736	65.572	off 100	constant variation remains			7.5h		65.82	65.88	65.78	64.78	65.6	0.454885
S20D20_3	S20	4.55	DomBoard.cfdst	S20D20_2	D20	MNb	interesting to see if converges to S00 case		0.005	10	ADV1	SST k-ome	off 900	default	104.2988736	65.09	off 900						65.42	64.09	65.59	64.82	65.53	0.637456
Roughness																											
All runs in this test 45S00D20MNb																											
R1_1	S00	4.55	Roughness.cfdst		D20	MNb	roughness1	0.00075mm	0.005	10	ADV1	SST k-ome	off 900	default	104.2988736		results inaccessable (program crashes when opening scenario)											
R2_1	S00	4.55	Roughness.cfdst		D20	MNb	roughness2	0.0015mm	0.005	10	ADV1	SST k-ome	off 900	default	104.2988736	44.344	stopped at t1479	job exceeded time limit	looks converged, constant variation		73h		44.1	45.5	44.72	44.07	43.33	0.812484
R2_1	S00	4.55	Roughness.cfdst		D20	MNb	roughness2, Intelligent Wall Formulation disabled	0.0015mm	0.005	10	ADV1	SST k-ome	off 500	default	104.2988736	28.412	off 500						29.46	27.9	28.4	28.75	27.55	0.744627
R3_1	S00	4.55	Roughness.cfdst		D20	MNb	roughness3	0.01mm	0.005	10	ADV1	SST k-ome	off 900	default	104.2988736	41.076	stopped at 1436	job exceeded time limit	mainly vz variation				41.6	42.58	41.54	40.08	39.58	1.222898
R3_2	S00	4.55	Roughness.cfdst		D20	MNb	roughness3	0.01mm	0.005	10	ADV1	SST k-ome	off 500	default	104.2988736	40.292	off 500	Turned out Intelligent Wall Formulation was not disabled, really have to have selected scenario active					40.18	39.48	39.77	41.12	40.91	0.709204
R3_3	S00	4.55	Roughness.cfdst		D20	MNb	roughness3, Intelligent Wall Formulation disabled	0.01mm	0.005	10	ADV1	SST k-ome	off 500	default	104.2988736	27.104	off 500	Constant variation in TED, Vy and Vz					28.11	27.18	26.88	26.85	26.5	0.611825
R4_1	S00	4.55	Roughness.cfdst		D20	MNb	roughness4	0.1mm	0.005	10	ADV1	SST k-ome	off 900	default	104.2988736	44.382	stopped at t1375	job exceeded time limit	looks converged, constant variation				44.73	44.45	44.44	44.6	43.69	0.404809
R4_2	S00	4.55	Roughness.cfdst		D20	MNb	roughness4	0.1mm	0.005	10	ADV1	SST k-ome	off 500	default	104.2988736	42.998	off 500	Turned out Intelligent Wall Formulation was not disabled, really have to have selected scenario active					42.87	42.86	43.74	42.52	43	0.451243
R4_3	S00	4.55	Roughness.cfdst		D20	MNb	roughness4, Intelligent Wall Formulation disabled	0.1mm	0.005	10	ADV1	SST k-ome	off 500	default	104.2988736	28.584	off 500	Constant variation in TED, Vy and Vz					29.52	29.27	28.21	28.26	27.66	0.781876
R0_1	S00	4.55	S00D20 ALL.cfdst	45S00D20MNb ADV5 1(	D20	MNb	board suppressed, Intelligent Wall Formulation disabled	0mm	0.005	10	ADV5	SST k-ome	off 500	advanced	104.2988736	32.826	off 600						33.74	33.72	33.26	32.15	31.26	1.087695
R0_2	S00	4.55	S00D20 ALL.cfdst	45S00D20MNb ADV1	D20	MNb	board suppressed, Intelligent Wall Formulation disabled	0mm	0.005	10	ADV1	SST k-ome	off 500	advanced	104.2988736		off 600											
Part 2																											
P00D20MNb ADV1																											
30_1	P00	3.04	P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default			failed											
30_2	P00	3.04	P00D20 ALL.cfdst	30_1, from t=0.4	D20	MNb			0.0025	10	ADV1	SST k-ome	off 2000	default		37.9	stopped at t190											

30_3	P00	3.04	P00D20 ALL.cfdst	30_2, from t190	D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default			failed											
30_4	P00	3.04	P00D20 ALL.cfdst	30_2, from t190	D20	MNb			0.0025	10	ADV1	SST k-ome	off 2000	default		36.38	stopped at t282	job exceeded time limit										
35_1	P00		P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default			stopped at step115											
35_2	P00		P00D20 ALL.cfdst	35_1, from t115	D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default			failed 'solver excited unexpectedly'											
35_3	P00		P00D20 ALL.cfdst	35_2, from t115	D20	MNb			0.0025	10	ADV1	SST k-ome	off 2000	default		45.97	stopped at 221	results somehow not saved										
35_4	P00		P00D20 ALL.cfdst	35_3, from 221	D20	MNb			0.0025	10	ADV1	SST k-ome	off 2000	default		24.26	stopped at t308	results somehow not saved										
40_1	P00		P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default			failed											
40_2	P00		P00D20 ALL.cfdst	40_1, from t=0.2	D20	MNb			0.0025	10	ADV1	SST k-ome	off 2000	default		56.46	stopped at 157	job exceeded time limit										
40_3	P00		P00D20 ALL.cfdst	40_2, from t157	D20	MNb			0.005	10	ADV1	SST k-ome	off 1843	default			failed											
40_4	P00		P00D20 ALL.cfdst	40_2, from t157	D20	MNb			0.0025	10	ADV1	SST k-ome	off 1843	default		26.79	stopped at t228											
45_1	P00		P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default			stopped at 175											
45_2	P00		P00D20 ALL.cfdst	45_1, from t175	D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default		26.69	stopped at t280	job exceeded time limit	results somehow not saved									
45_3	P00		P00D20 ALL.cfdst	45_2, from t175	D20	MNb			0.005	10	ADV1	SST k-ome	off 1720	default		55.98333	stopped at t331	job exceeded time limit					62.12	57.37	48.46			6.93477
P00D20MNb ADV5 10 layers																												
30_1	P00	3.04	P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced			stopped at t138											
30_2	P00	3.04	P00D20 ALL.cfdst	30_1, from t138	D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced			stopped at 283	job exceeded time limit			73h							
30_3	P00	3.04	P00D20 ALL.cfdst	30_2, from t283	D20	MNb	try with less iterations		0.005	3	ADV5	SST k-ome	off 1777	advanced		20.655	stopped at t425				73h		19.2	19.53	20.93	22.96		1.709903
35_1	P00		P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced			stopped at t107											
35_2	P00		P00D20 ALL.cfdst	35_1, from t107	D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced			stopped at 203	job exceeded time limit			73h							
35_3	P00		P00D20 ALL.cfdst	35_2, from t203	D20	MNb	try with less iterations		0.005	3	ADV5	SST k-ome	off1797	advanced		24.9675	stopped at t312	not converged yet			73h		24.85	24.55	25.61	24.86		0.451839
40_1	P00		P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced			stopped at t114											
40_2	P00		P00D20 ALL.cfdst	40_1, from t114	D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced		40.94	stopped at t205	job exceeded time limit			73h							
40_3	P00		P00D20 ALL.cfdst	40_2, from t205	D20	MNb	try with less iterations		0.005	3	ADV5	SST k-ome	off 1795	advanced		35.7125	stopped at t269	job exceeded time limit			73h		30.85	30.26	40.94	40.8		5.960511
45_1	P00		P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced			stopped at 104											
45_2	P00		P00D20 ALL.cfdst	45_1, from t104	D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced		38.34	stopped at t177	job exceeded time limit			73h							
45_3	P00		P00D20 ALL.cfdst	45_2, from t177	D20	MNb	try with less iterations		0.005	3	ADV5	SST k-ome	off 1823	advanced		39.154	stopped at t261				73h		34.35	33.56	35.88	38.34	53.64	8.300559
S00D20MNb ADV1																												
30_1	S00	3.039	P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default	48.6847032	47.19	off 2000	variation constant					48.06	46.72	47.93	47.82	45.42	1.124189
35_1	S00	3.544	P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default	69.1652592	46.912	stopped at 1873	variation constant. Seems is not improving anymore					46.23	45.24	46.38	48.28	48.43	1.389198
40_1	S00	4.047	P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default	87.5879556	50.034	stopped at 1674	variation constant. Seems is not improving anymore					52.3	48.61	50.18	49.95	49.13	1.414931
45_1	S00	4.55	P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV1	SST k-ome	off 2000	default	104.2988736	70	off 2000	compare to Domboard same run, drag there 63.8					70.94	66.81	71.54	71.74	68.97	2.092833
S00D20MNb ADV5 10 layers																												
30_1	S00	3.039	P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced	48.6847032	31.086	stopped at 1518	only small variation					30.82	30.82	31.03	31.04	31.72	0.370378
35_1	S00	3.544	P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced	69.1652592	32.596	stopped at 1506	variation constant. Seems is not improving anymore					32.21	32.77	32.85	33.01	32.14	0.39469
40_1	S00	4.047	P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced	87.5879556	33.652	stopped at 1331	vz variation still slightly too large					34.03	33.14	32.7	34.11	34.28	0.692004
40_2	S00	4.047	P00D20 ALL.cfdst	40_1	D20	MNb			0.005	10	ADV5	SST k-ome	off 700	advanced	87.5879556	33.904	off 700	vz variation still bit too large, but does not seem to improve					33.93	34.41	33.73	33.57	33.88	0.31588
45_1	S00	4.55	P00D20 ALL.cfdst		D20	MNb			0.005	10	ADV5	SST k-ome	off 2000	advanced	104.2988736	43.262	stopped at 1498	still some variation in vz and vy					40.9	42.77	43.83	44.02	44.79	1.504417
45_2	S00	4.55	P00D20 ALL.cfdst	45_1	D20	MNb			0.005	10	ADV5	SST k-ome	off 502	advanced	104.2988736	43.832	off 502	vz variation still bit too large, but does not seem to improve					43.01	43.35	44.89	44	43.91	0.717928
45_3	S00	4.55	P00D20 ALL.cfdst	45_2	D20	MNb	see if less iterations works now also		0.01	3	ADV5	SST k-ome	off 500	advanced	104.2988736	42.174	off 500						42.41	42.62	42.46	41.43	41.95	0.484799

C Mesh Codes

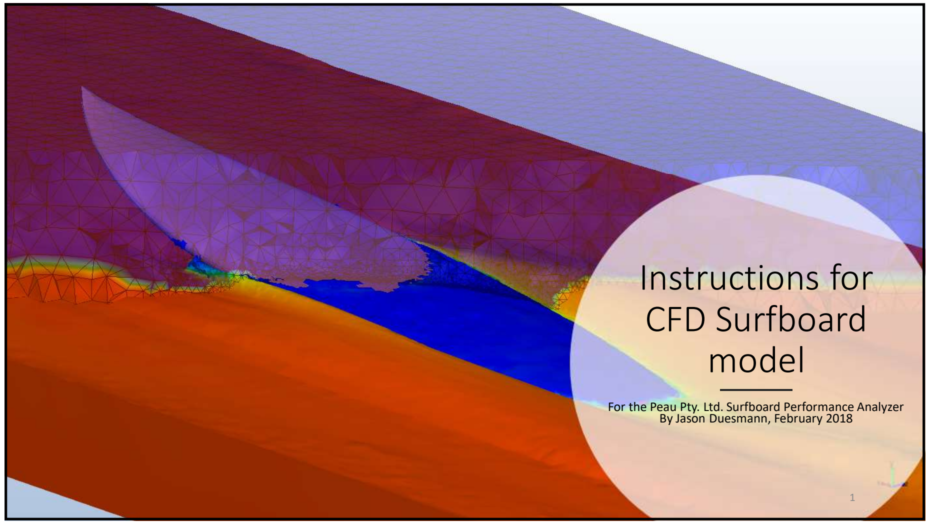
Mesh	Domain	Element size			Refinement region dimensions					
		general	refinement	surface	x offset	y offset	z offset	x length	y length	z length
M34	D00	0.5	0.05	0.005	0	-0.1	0	6	0.7	3.55
M34	D00	0.5	0.05	0.04	0	-0.1	0	6	0.7	3.55
M33	D00	0.5	0.1	0.01	0	-0.1	0	6	0.7	3.55
M33b	D00	0.5	0.1	0.04	0	-0.1	0	6	0.7	3.55
M32	D00	0.5	0.2	0.01	0	-0.1	0	6	0.7	3.55
M31	D00	0.5	0.4	0.04	0	-0.1	0	6	0.7	3.55
M31b	D00	0.5	0.4	0.01	0	-0.1	0	6	0.7	3.55
MB	D10	0.1/0.2[a]	0.05	0.04	0	0	0	6	0.3/0.5[b]	2
MK	D20	0.1	-	0.03	-0.3	-0.1	0	2	0.2	0.7
ML	D20	0.05	-	0.03	-0.3	-0.1	0	2	0.2	0.7
MM	D20	0.05	0.025	0.025	-0.3	-0.1	0	2	0.2	0.7
MN	D20	0.05	0.0125	0.0125	-0.3	-0.1	0	2	0.2	0.7
MNb	D20	0.05	0.0125	0.0125	-0.3	-0.1	0	2	0.2	0.7
MO	D20	0.05	0.00625	0.00625	-0.3	-0.1	0	2	0.2	0.7

Table 8: Mesh setups of most meshes in the the Run Sheet. 'Surface' refers to the element size of the surface of the board, not to the board mesh density. All distances in meters.

[a] Water box 0.1, Air box 0.2.
[b] front section of 3.5m x length has 0.3 y height, back 2.5m has 0.5.

D Instructions for Aku Shaper project

The next few pages contain instructions in a PowerPoint format that could be used for a future student or researcher working on the same project. Screencast videos are recorded, for these being an efficient way of communicating the workflow in the used software programs.
To ensure it is easy to continue were I ended, the instructions are accompanied by model files and additional literature. These are not shown in this report.



Preface

Hello,

These instructions are meant for someone working in the Peau Pty Ltd Surfboard Performance Analyser Project, who is ready to continue from where I left.
During my internship, I tried to validate a CFD surfboard model to experimental results. Although getting quite far, matching results were not obtained. This was most likely caused by poor meshing, but I was unable to improve this due to limited computing power.
In these instructions I give an overview of how I recommend the CFD model to be set up in Autodesk CFD, using screencast videos to show you what to do. I also provide a lot of references which are definitely (!) worth reading in order to understand everything. Besides, I made a few comment on things to be improved. These improvements are necessary when the model has to be used for a parametric analysis.
Furthermore, I could advice you to have a look at my report. It makes clear why the model is setup this way. Maybe you notice mistakes or believe things could be done better or differently.
Of course, more knowledge about CFD, hydrodynamics and surfing is necessary, but this information is not provided in these instructions.
Some of the references may however prove useful. In a nearby folder also a manual of Autodesk CFD 2018 is available.
It may also be good to know that I saved the important design study files (.cfdst) in a nearby folder, so you could have a look at these.
These instructions are to be used for a model in Autodesk CFD, but I believe they are still fairly useful when another program becomes the first choice.

Good luck!

Jason
February 2, 2018

2

Experiments for validation

Experiments were performed by Sam Schumacher in 2011 at the Australian Maritime College in Launceston, Tasmania. A surfboard was towed through a towing tank in a straight line at four different velocities, in a fixed orientation.

This situation is a great simplification compared to real life surfing, but in a later model extra complexities can be added.

3

Content 1

I split up the model making and simulations in four parts. First, the digital board and domain geometry have to be prepared, so that they can be imported into Autodesk CFD. Secondly, I explain how to do the setup in Autodesk CFD. Subsequently, I show how to solve the simulation. Lastly, I make clear how to view and analyze results.
Every video is preceded with a short description about the content. A few notes and useful references are given in the slides following the video.

1. Prepare board
    1. From BRD to STL
    2. Reduce number of faces
    3. Align board, make domain box and combine
2. CFD Setup
    1. Material, BC (boundary conditions), IC (Initial conditions)
    2. Mesh Sizing
3. CFD Solver
    1. Settings
    2. Simulation Job Manager
4. CFD Results
    1. Convergence Assessment
    2. Viewing Mesh
    3. Visualizations
    4. Cloning, comparing and sharing

4

1.1 | Prepare Board: From BRD to STL

This video ' From BRD to STL' is the first of three videos in the series 'Prepare Board'. In this series CAD files are made so that your surfboard is importable into Autodesk CFD and that awesome flow simulations can be performed.

Start: a surfboard design .brd-file in AKUshaper

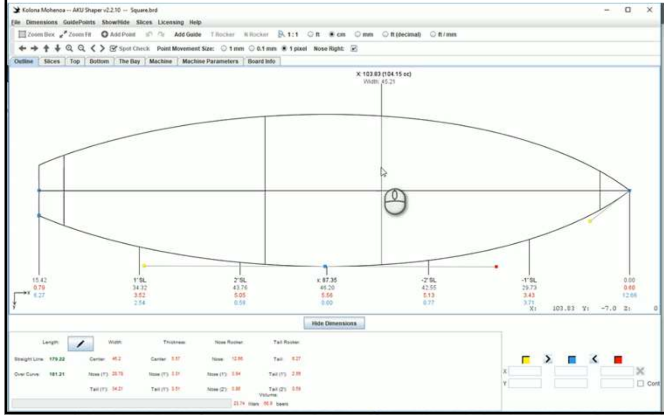
In this video:
-Open .brd-file in AKUshaper and export as .stl
- Upload .stl to Fusion 360 and open it
- Close gaps in geometry by exporting to Brep and stitching in the Patch Workspace
- Scale, rotate and align the body as desired
- Save as .stl

End: a surfboard .stl ready for further adjustments

In the next video, the mesh of the closed .stl-file is going to be reduced, converted to a body and exported as .igs.

5

1.1 | Prepare Board: From BRD to STL



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2mYH8CY

1.1 | Prepare Board: From BRD to STL

Notes:

- Often, the shape of the stl is already closed, the stitching is not needed then.
- Not all board types could be generated in Aku Shaper. You could maybe also design your board in a CAD program and export it as an stl, or export it as a body directly.

7

1.2 | Prepare Board: Reduce Number of Faces

This video 'Reduce Number of Faces'' is the second of three videos in the series 'Prepare Board'. In this series CAD files are made so that your surfboard is importable into Autodesk CFD and that awesome flow simulations can be performed.

Start: .stil-file with closed mesh body, with large number of faces

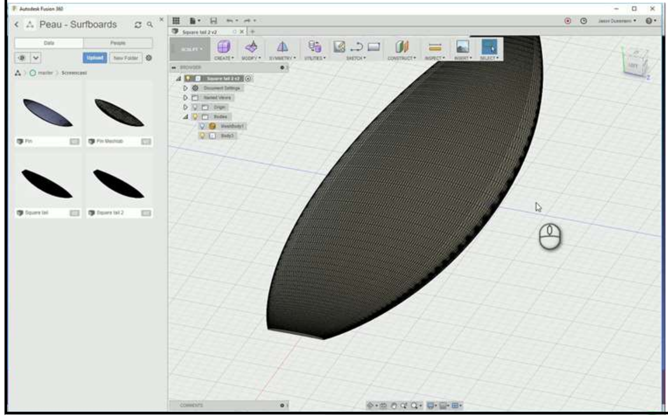
In this video:
- Open .stl in MeshLab
- Reduce number of faces in MeshLab
- Export reduced geometry as .stl
- Upload and open in Fusion 360
- Convert mesh body to Brep

End: body in Fusion 360 with lower number of faces

In the next video the board shall be aligned as desired and the domain box shall be made. In SimStudio Tools both objects shall be combined into one .igs-file that can be opened in Autodesk CFD.

8

1.2 | Prepare Board: Reduce Number of Faces



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2mXP5IE

1.2 | Prepare Board: Reduce Number of Faces

Notes:

- Please try if your computer is able to handle the unreduced mesh in Autodesk CFD, or that you also need a reduction.
- The meshed board is slightly unsymmetric. Maybe a symmetric board is preferred.
- As discussed in the report, the board mesh becomes quite irregular and this may be a problem for generating a nice fluid mesh. Look for ways to create more uniform board mesh, preferable even rectangular shaped faces or curved edges.

10

1.3 | Prepare Board: Align board, Make Domain Box and Combine

This video 'Align board, Make Domain Box and Combine' is the third of three videos in the series 'Prepare Board'. In this series CAD files are made so that your surfboard is importable into Autodesk CFD and that awesome flow simulations can be performed.

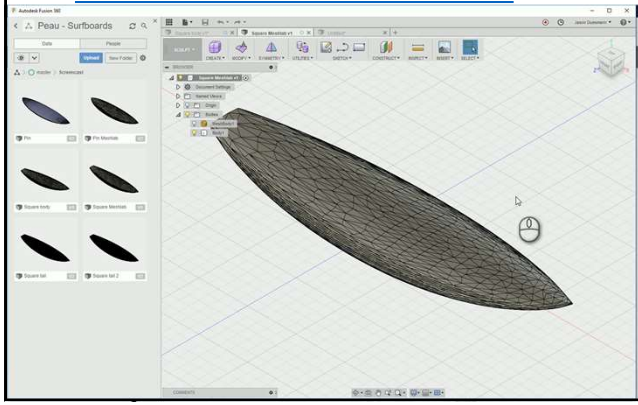
Start: surfboard body in Fusion 360

In this video:
Align Board
- Scale and align board in origin
- Make active component and sketch alignment point
- Tilt and move alignment point to origin
- Export as .igs
Make Domain Box
- Sketch rectangles, set width and depths
- Mirror to create other half of domain
- Extrude both boxes seperately, two sided
- Export as .igs
Combine
- Insert both board and box in SimStudio Tools
- Export as .igs

End: .igs-file with board and box, ready to import in Autodesk CFD

11

1.3 | Prepare Board: Align board, Make Domain Box and Combine



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2F0Xv9x

1.3 | Prepare Board: Align board, Make Domain Box and Combine

Notes:

- Obviously, different alignment/orientation might be desired.
- The domain box length is recommended to be even longer, but this increases computation time really a lot. I think this is a nice balance, but please also investigate if good results can also be obtained when only the region very nearby the submerged part of the board is used.
- Exporting .step files is also possible, these can also be imported into Autodesk CFD.
- Use symmetry of design to half the domain size to save computation time, if the supervisor agrees.

1.3 | Prepare Board: Align board, Make Domain Box and Combine

References:

- Autodesk CFD: External incompressible flow. About length of domain.
- Autodesk CFD: How to set up a free surface analysis of a boat. Reason for making the boxes.

2.1 | CFD Setup: Material, BC, IC

This is the first of 2 videos in the series ' CFD Setup' in which I'll show you how to setup your CFD model of the surfboard in Autodesk CFD.

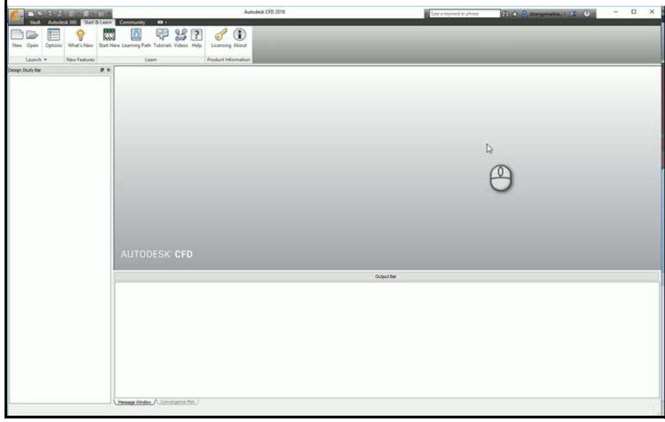
Start: .igs CAD model of surfboard and domain box

In this video:
- Set length units
- Change view to transparant
- Rename designs and scenarios
- Apply materials
- Edit materials
- Suppress bodies
- Make groups
- Apply velocity, pressure and slip/symmetry boundary conditions
- Set height of fluid and velocity initial conditions
- Check your setup in the design study bar

End: setup done, except the mesh sizing.

In the next video I'll show how to apply a mesh.

2.1 | CFD Setup: Material, BC, IC



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2DB0ycu

2.1 | CFD Setup: Material, BC, IC

Notes:

- Please investigate the use of surface roughness in combination with the turbulence model and Intelligent Wall Formulation
- Look if results change when using different water properties
- Of course, board suppression must not be done once motion studies are performed.

17

2.1 | CFD Setup: Material, BC, IC

References:

- Autodesk CFD: How to set up a free surface analysis of a boat. BC en IC
- Autodesk CFD: What happens with solid volumes in an external ow simulation? About suppression
- Thomas A. Shapiro. The effect of surface roughness on hydrodynamic drag and turbulence.

18

2.2 | CFD Setup: Mesh Sizing

This is the second of 2 videos in the series ' CFD Setup' in which I'll show you how to setup your CFD model of the surfboard in Autodesk CFD.

Start: setup done, except the mesh sizing

In this video:
- Assign element sizes to volumes
- Mesh a surface
- Define refinement region
- Apply wall layers
- Use the diagnostics
- Ways to generate your mesh

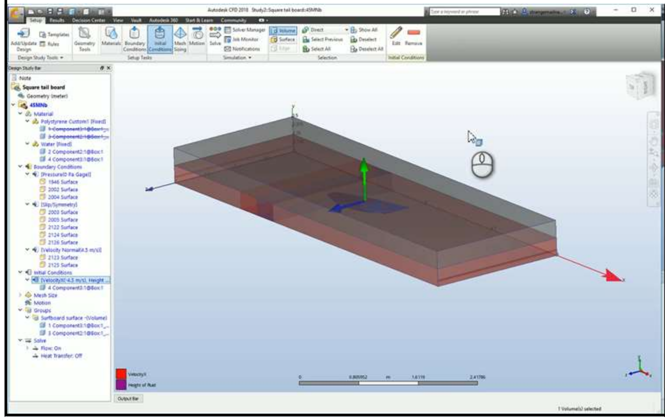
End: setup all done, ready for solving

Note: the assigned mesh of this video does result in a mesh that is not mesh independent and wall layers leading to too high y values. Please improve the mesh.

The next series of videos will show you how you can correctly apply the solver settings.

19

2.2 | CFD Setup: Mesh Sizing



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2nabL8y

20

2.2 | CFD Setup: Mesh Sizing

Notes:

- The applied mesh does definitely not lead to good results. Some bad things are shown in video 4.2. Investigate how it could be improved.
- The wall layers are important for the boundary layer. Look into the report or the references for estimating the required thinness of the first layer.
- Apply a mesh sensitivity study.

21

2.2 | CFD Setup: Mesh Sizing

References:

- Autodesk CFD: Wall layers. Wall layer settings and their effects.
- Autodesk CFD: Manual mesh sizing.
- Autodesk CFD: Diagnostics.
- leap Australia CFD, tips & tricks: Convergence and mesh independence study.
- leap Australia CFD, tips & tricks: Estimating the first cell height for correct y+.
- leap Australia CFD, turbulence part 3 - selection of wall functions and y+ to best capture the turbulent boundary layer.
- CFD-online: Y plus wall distance estimation.
- leap Australia CFD, turbulence part 4 - reviewing how well you have resolved the boundary layer.

22

3.1 | CFD Solver: Settings

This is the first of 2 videos in the series ' CFD Solver'. In this series I explain how to apply correct solver settings for you surfboard simulation and how to use the Simulation Job Manager.

Start: completed setup of your surfboard model in Autodesk CFD

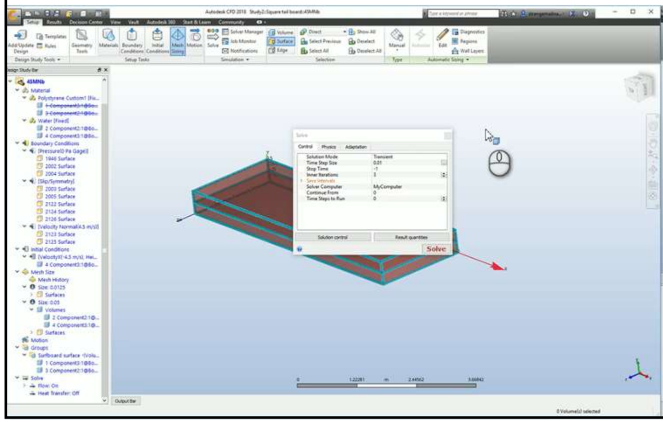
In this video:
- Physics: free surface
- Hydrostatic pressure and adaptation not possible
- Physics: Turbulence model, SST k-omega SAS
- Solution Controls: Advection scheme 5
- Solution Controls: Enable Intelligent Solution Control
- Solution Controls: Show Automatic Convergence Assessment
- Set time step size, stop time and inner iterations
- Save intermediate results
- Set solver computer
- Set result quantities
- Submit job

End: submitted a job to the CLOUD that is being solved

In the next video I'll show how to use the Simulation Job Manager

23

3.1 | CFD Solver: Settings



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2DH4KXc

3.1 | CFD Solver: Settings

Notes:

- Take care of the limitations/assumption involving a free surface analysis in Autodesk CFD.
- It is also possible to right-click on a non-active scenario and solve it. I however noticed that sometimes not all adjustments to the solver settings are actually saved when doing it this was. I experienced this with enabling/disabling Intelligent Wall Formulation in the advanced turbulence settings.
- Maybe play around with the time step size and iterations. For finer meshes step size might have to be smaller, whereas for coarser meshes step size could maybe be larger.
- See if you can use a better computer or cluster for solving.
- You can use the flag manager to decrease download sizes if desired.
- Maybe use a different gravity vector when simulating surfing on an inclined surface.

25

3.1 | CFD Solver: Settings

References:

- Autodesk CFD, error: "Analysis has stopped because the solver has exited unexpectedly" A few possible solutions to an error that might occur.
- Autodesk CFD: Advection schemes.
- Autodesk CFD: Transient flows.
- Autodesk CFD: Transient parameters.
- Autodesk CFD: Turbulence.
- Royce Abel. Autodesk CFD video: Understanding the turbulence models available in Autodesk Simulation CFD.
- Matt Bemis Marwan Azzam. Autodesk CFD video: Free surface best practices.

26

3.2 | CFD Solver: Simulation Job Manager

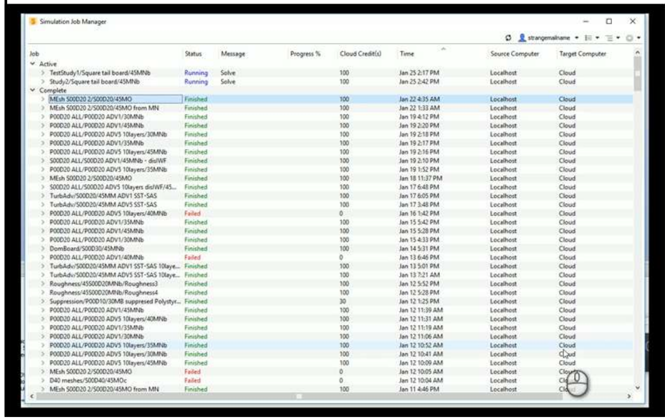
This is the second of 2 videos in the series ' CFD Solver'. In this series I explain how to apply correct solver settings for you surfboard simulation and how to use the Simulation Job Manager.

In this video:
- Use Simulation Job Manager when solving in the CLOUD
- See active and completed jobs
- See Output in Autodesk CFD
- View details and data transfer
- Set preferences for temporary storage folder
- Don't forget to empty your bin every now and then to avoid your disk running out of space

In the next series of videos I'll show how to view the results of your surfboard simulation in Autodesk CFD.

27

3.2 | CFD Solver: Simulation Job Manager



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2n8peOr

3.2 | CFD Solver: Simulation Job Manager

Notes:

- I forgot to mention that the CLOUD computer has a job time limit of 3 days. When the job has not reached the set number of time steps after 72hours, it will stop the simulation and download the results. You can then restart it from the last saved time step. But it is really annoying when you want to run longer simulations. It's much better if a more powerfull computer without this limitation could be found.

29

4.1 | CFD Results: Convergence Assessment

This is the first videos of the CFD Results series, in which I show how to analyze the surfboard simulation results in Autodesk CFD. This first video is about convergence assessment.

Start: A finished simulation

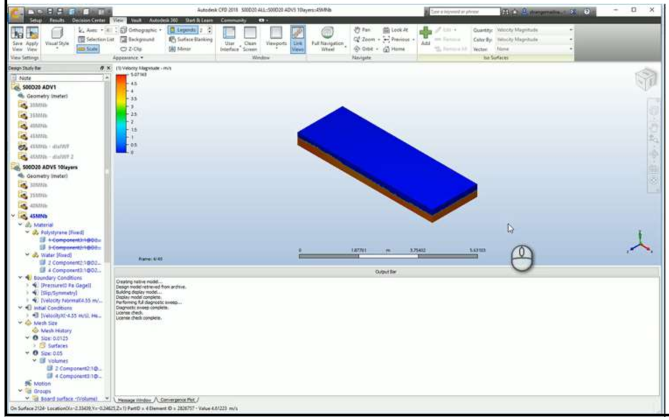
In this video:
- Loading results
- Three ways to assess convergence:
- Convergence plots
- Animating flow visualizations
- Quantity monitoring

End: results considered to be converged, ready for analysis.

In the next video I'll show you how to assess the mesh.

30

4.1 | CFD Results: Convergence Assessment



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2DS6ihV

4.1 | CFD Results: Convergence Assessment

Notes:

- Residual monitoring is a often used way to check convergence. In the convergence plots, you are able to view 'Residual Out', but I did not completely understand what this meant.
- Exporting the values is also a good way to see the magnitude of the remaining variations of a variable.
- The animation that was done for the free surface, could of course also be done for other visualizations.

32

4.1 | CFD Results: Convergence Assessment

References:

- Autodesk CFD: Convergence plot.
- Autodesk CFD: Automatic convergence assessment.

4.2 | CFD Results: Viewing Mesh

This is the first videos of the CFD Results series, in which I show how to analyze the surfboard simulation results in Autodesk CFD. In this video I'll tell you how to view your mesh and make a few comments on poor meshing seen.

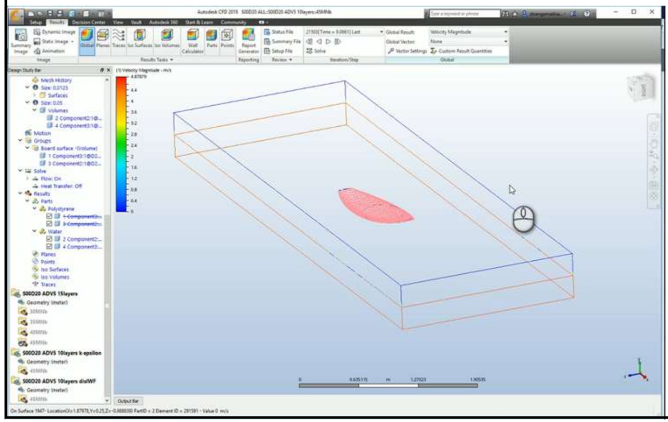
Start: generated mesh

In this video:
- Make planes to view mesh
- View wall layers
- Transition from prismatic wall layer to tetrahedral elements
- Wall layer thickness and y
- Badly meshed point at initial water interface
- Advice to perform mesh independence study

End: you have assessed your mesh and know if it needs improvement

The next video shows some flow visualization options.

4.2 | CFD Results: Viewing Mesh



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2DXfbpV

4.2 | CFD Results: Viewing Mesh

Notes:

- Please understand that the shown mesh needs improvement.
- Transition from wall layers to rest of domain
- Thickness of wall layers
- Decrease element size
- Perform mesh sensitivity study
- In the next video, I discovered the Z-clip viewing option to see the elements in 3D.

4.2 | CFD Results: Viewing Mesh

References:

- Autodesk CFD, error: "Analysis has stopped because the solver has exited unexpectedly" A few possible solutions to an error that might occur.
- Autodesk CFD: Advection schemes.
- Autodesk CFD: Transient flows.
- Autodesk CFD: Transient parameters.
- Autodesk CFD: Turbulence.
- Royce Abel. Autodesk CFD video: Understanding the turbulence models available in Autodesk Simulation CFD.
- Matt Bemis Marwan Azzam. Autodesk CFD video: Free surface best practices.

37

4.3 | CFD Results: Visualizations

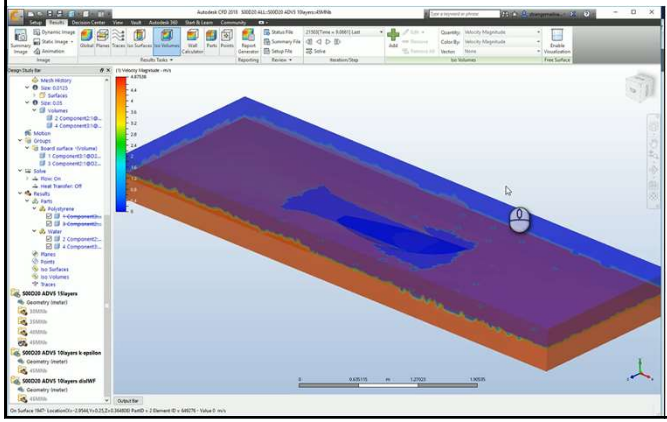
This is the first videos of the CFD Results series, in which I show how to analyze the surfboard simulation results in Autodesk CFD. In this video I show a few ways how to make the flow or other quantities visible.

In this video:
- Iso-surface
- Iso-volume
- Traces
- Z-clip
- Saving and applying views

The next video will show you how to clone scenarios or designs and use the decision center.

38

4.3 | CFD Results: Visualizations



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2FuJJMC

4.3 | CFD Results: Visualizations

Notes:

- You can also export dynamic images and view these in Autodesk CFD Viewer (free).

40

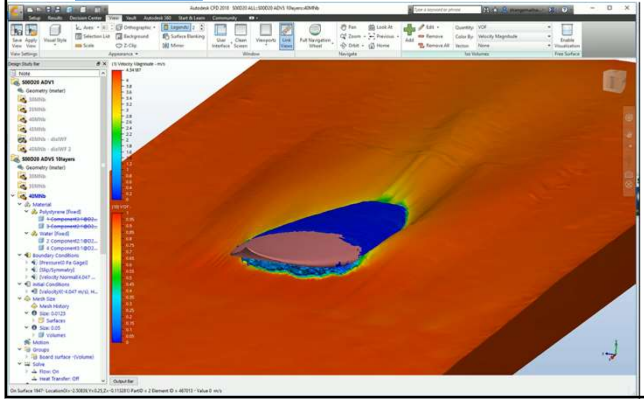
[4.4 | CFD Results: Cloning, Comparing and Sharing](http://autode.sk/2nwvhwa)

This is the fourth (and probably last) video of the CFD Results series, in which I show how to analyze the surfboard simulation results in Autodesk CFD. This video is about cloning, comparing in the Decision Center and sharing files containing less information.

In this video:
- Cloning scenarios, with or without mesh and results
- Cloning designs and updating designs if desired
- Making summary images
- Comparing in the Decision Center
- Exporting smaller files for sharing purposes

41

4.4 | CFD Results: Cloning, Comparing and Sharing



You can also find the video file in a nearby folder or view the online version (fullscreen): http://autode.sk/2nwvhwa

Recommended settings

Here's a list of my recommendations for the setup and settings to be used. Please read the recommendations to see what needs to be improved, with suggestions how to do this. The underlined ones have high priority.

- Board: S10, 4000 faces. <u>Try to improve CAD model.</u>
- Domain: D20. Investigate use of smaller domain box.
- Material: water and suppressed board. See what changes for different water properties. <u>Apply surface roughness and see what Intelligent Wall Formulation does for taking into account the roughness in the turbulence model.</u>
- BC and IC as shown in video.
- Mesh: start with MNb <u>and improve this. Refine.</u>
- Wall layers: use the advanced settings (10 layers, layer Factor: 0.6, gradation: 1.25). <u>Improve transition to tetrahedral elements. Make thinner. Circumvent badly meshed point near initial free surface.</u>
- Solver: ADV5, SST k-omega SAS, 0.005s, 10 inner iterations. <u>Investigate Intelligent Wall Formulation influence on turbulence model.</u>
- Convergence: run for at least 2000 time steps. Monitor drag forces over time.

43

The best model, using the setting recommended in these instructions, can be found in "S00D20 ALL.cfdst". Look at the design ADV5 10 layers.

Please read the conclusion, discussion and recommendations of my report to know how useful the described model is.

End

44