Master's thesis

Marine technology

2021

Teppo Aro

CFD -SIMULATIONS OF SHIP PERFORMANCE IN WAVES

TURKU AMK
TURKU UNIVERSITY OF
APPLIED SCIENCES

MASTER'S THESIS | ABSTRACT

TURKU UNIVERSITY OF APPLIED SCIENCES

Marine Technology

2021 | number of pages 41, number of pages in appendices 1

Teppo Aro

CFD-SIMULATIONS OF SHIP PERFORMANCE IN WAVES

Vessel performance predictions are done during the initial stages of design. CFD simulations are getting more commonly used, but only calm water simulations are currently done. Often the hull's performance and behavior in waves is evaluated at model test which is done at the tail end of design. This means there isn't always much that can be done without wasting concurrently done design time.

In this thesis methods for simulating ship performance and behavior in waves have been considered. In the first phase a static wave breaker was simulated, the simplified approach meant most of the effort could be concentrated on the wave simulations. Comparative measured data was not available for these simulations, but the results seemed to be at least on the right magnitude.

In the second phase a 170 m ferry was simulated in both regular and irregular waves. Several different methods were compared in regular waves, against each other and model test results. The results were in good alignment with model test results, with the best method being the self-propulsion CFD case.

Two different sea states were represented with irregular wave simulations and compared against model test results. In the smaller wave height of Adriatic sea the simulation results were very close to the ones found in the model test report, but in the more demanding case of S.North sea the results were diverging. Several different factors could be contributing to this and future research should be done to find out the effect of each separate factor. As a proof-of-concept this thesis provided additional credibility for the methods and trust that in the future these methods could provide reliable insight into vessel performance in waves.

KEYWORDS:

Fluid dynamics, Hydrodynamics, Ship, Vessel, Performance, Wave

OPINNÄYTETYÖ YAMK | TIIVISTELMÄ

TURUN AMMATTIKORKEAKOULU

Meritekniikka

2021 | 40 sivua, 1 liitesivu

Teppo Aro

ALUKSEN SUORITUSKYVYN ARVIOINTI AALLOKOSSA CFD:N AVULLA

Alusten suunnittelun aikana niiden suorituskykyä yritetään arvioida mahdollisimman tarkkaan jo suunnitteluvaiheessa. Virtausmekaaniset simulaatiot ovat yleistyneet tässä käytettävänä työkaluna, mutta ne toteutetaan tällä hetkellä yleensä tyynen veden tilanteina. Aaltokäyttäymistä ja suorituskykyä arvioidaan yleensä vasta mallikokeissa, jotka suoritetaan suunnittelun loppuvaiheissa, joten tilaa ei muutoksille ole paljoa.

Tässä työssä tutkittiin menetelmiä simuloida aluksen käyttäytymistä aallokossa ja niiden suorituskykyä. Ensimmäisessä vaiheessa tutkittiin kiinteää aallonmurtajaa säännöllisessä aallokossa. Tämä osoittautui yksinkertaisemman tapauksen tutkia. Vertailudataa tästä tapauksesta oli heikosti, mutta tulokset olivat ainakin oikeaa suruusluokkaa.

Toisessa vaiheessa simulaatioita tehtiin ~170 m alukselle sekä säännöllisessä, että epäsäännöllisessä aallokossa. Useita menetelmiä verrattiin toisiinsa ja mallikokeeseen säännöllisessä aallokossa. Liikkeitä vertaillessa ne olivat hyvin linjassa mallikokeen kanssa.

Epäsäännöllisessä aallokossa simuloitiin kahta eri tilannetta, kuvastamaan kahta eri merialuetta. Pienemmässä aallokossa tulokset olivat linjassa mallikokeen kanssa, mutta vaativammassa meritilanteessa tuloksissa oli merkittävästi eroa.Menetelmässä on monta suuntaa joihin pitää saada lisää varmistusta ja kehitystä, mutta alustavast tulokset näyttävät lupaavalta.

ASIASANAT:

Virtauslaskenta, Aallot, Laiva, Alus, Suorituskyky, Fluididynamiikka, Hydrodynamiikka

CONTENT

LIST OF ABBREVIATIONS AND SYMBOLS 7

1 INTRODUCTION 6

2 WAVE THEORY 7
2.1 Stokes’s wave theory 7
2.2 Kelvin wake 7
2.3 Wave force from drag coefficient 7
2.4 Irregular waves 8
2.5 Response Amplitude Operators (RAOs) 8
2.6 Wave scatterplots for different areas 9

3 CFD-THEORY 10
3.1 Best practises from literature 10
3.2 Meshing 10
3.3 Overset mesh 11
3.4 Temporal discretization and timestep length 12
3.5 Turbulence modeling 13
3.6 Wall functions and Y+ 14

4 SIMULATIONS 15

5 WAVE BREAKER SIMULATIONS 16
5.1 Problem setup 16
5.2 Simulation setup 17
5.3 Results 18

6 SEA FARING VESSEL 22
6.1 Problem setup 23
6.2 Simulation setup 25
6.3 Results 27

7 SEA FARING VESSEL IN IRREGULAR WAVES 29
7.1 Problem setup 29
7.2 Simulation setup 30

7.3 Results 32

8 FUTURE RESEARCH 37

9 CONCLUSIONS 38

10 REFERENCES 39

FIGURES

Figure 1 Wave scatterplot example [6] 9
Figure 2 Best practices from documentation [7] 10
Figure 3 Overset topology. Top: Background mesh, Middle: Overset mesh, Bottom: Combined final mesh 12
Figure 4 Project visualization 15
Figure 5 Wave breaker drawing 16
Figure 6 Wave breaker simulation boundaries 17
Figure 7 Wave breaker meshing 18
Figure 8 Wave breaker snapshot 19
Figure 9 Deep water wave breaker simulation 20
Figure 10 Wave height at different cross sections 21
Figure 11 Superfast hull lines and charasteristic dimensions 22
Figure 12 Technical drawing of Superfast's appendices [11] 23
Figure 13 Boundary conditions for regular wave simulation 25
Figure 14 Wave damping and forcing areas 26
Figure 15 StarCCM+ and Aqwa side view visualization 27
Figure 16 Heave RAO comparison for fore waves [11] 27
Figure 17 Pitch RAO comparison for fore waves [11] 28
Figure 18 Regular wave visualization from StarCCM+ 28
Figure 19 Comparison of used spectra 30
Figure 20 Adaptive mesh refinement example 31
Figure 21 Irregular wave simulation visualization from StarCCM+ 32
Figure 22 Thrust in calm water [11] 33
Figure 23 Additional thrust required due to waves in Adriatic sea [11] 34
Figure 24 Additional thrust required due to waves in S.North sea [11] 35

EQUATIONS

Equation 1 Force from drag coefficient 8
Equation 2 Wall y+ [10] 14

TABLES

Table 1 Wave breaker dimensions 16
Table 2 Wave parameters for wave breaker simulation 17
Table 3 Drag forces of wave breaker simulation 19
Table 4 Drag forces for deep water wave breaker simulation 20
Table 5 Regular wave cases 23
Table 6 Irregular wave cases 29

LIST OF ABBREVIATIONS AND SYMBOLS

6-DOF	six degrees of freedom
Aqwa	Ansys Aqwa
Cell	One element of mesh
CFD	Computational fluid dynamics
k-ε	Turbulence model
k-ω	Turbulence model
Mesh, Meshing	Spatial discretization
RAO	Response Amplitude Operator
RANS	Reynolds averaged Navier-Stokes
StarCCM+	Siemens Simcenter StarCCM+

1 INTRODUCTION

Estimating hull performance is an important part of a design process, for any vessel type. It has an effect on plethora of other design disciplines, so figuring out the main hull shape early in design is crucial. Ship design is highly interconnected so changes in space reservations for some ship functions can cause changes to the hull, which in turn affects the power requirements for example.

Currently the ship hull form is usually optimized with CFD in calm water simulations. Meaning that several hull shapes performance are evaluated during the design process. These results are then factored into the other requirements and a suitable balance between them is found.

Once the hull is mostly finished and other design work has been started, a model test is performed for the hull. Usually these contain all of the necessary tests to fully evaluate the hull. Hull performance is tested in calm water as well as some sea states and stability checks. Based on these results some minor adjustments can be done to finalize the hull, but often this is done so late in the design process that the improvements available are limited.

The purpose of this thesis was to study the simulations of ship performance in waves. If found feasible these simulations could be used to support model testing or replace them entirely. With enough accuracy and cost-effectiveness this could provide great insight into the ships performance, way earlier in the design process, which in turn could lead to much more optimized hull shapes.

2 WAVE THEORY

Waves appear in many different areas of physics and mathematics, but in this thesis they are only considered from hydrodynamics and ship building point of view. This means that when referring to waves, they are considered as surface waves on inviscid fluid in normal gravity and piezometric pressure.

2.1 Stokes’s wave theory

For practical use of waves in deep water Stokes’s wave theory is commonly used in designing coastal and offshore structures. Stokes’s wave is a periodic wave, much like a common sine wave, but includes sharper peaks and a flatter through. In this thesis the regular waves used are the fifth order expansion on Stokes’s theory by Fenton [1]. This expansion provides a more accurate representation of waves.

2.2 Kelvin wake

A vessel moving across a surface on water produces a wake pattern known as Kelvin wake. In slow moving vessels this forms a pattern of two lines at approximately 19.47° from each other, forming a V -shape. Inside this area is a wave pattern that can be described by Stokes’s wave theory. This shape can be masked by propulsor wake and in high speeds the pattern changes, but it generally holds true in most cases. [2]

2.3 Wave force from drag coefficient

Drag coefficients are often used in fluid dynamics to roughly scale forces on a body to different velocities. This generally works also with wave induced force on a stationary body, but it is not as accurate. With the wave breaker simulations no measurement data was available, so using estimated drag coefficients was the only way to have any comparisons, even if it is used only to check that the same order of magnitude is achieved. Drag force from a coefficient is defined in Equation 1.

$$F_d = \frac{1}{2}\rho\mu^2 c_d A$$

Equation 1 Force from drag coefficient

F_d = Drag force

ρ = Density of fluid

μ = Characteristic velocity

C_d = Drag force coefficient

A = Projected sectional area from the flow direction

All of the values needed in Equation 1 are known, exept for the drag force coefficient, which can be estimated based on the nature of the used geometry.

2.4 Irregular waves

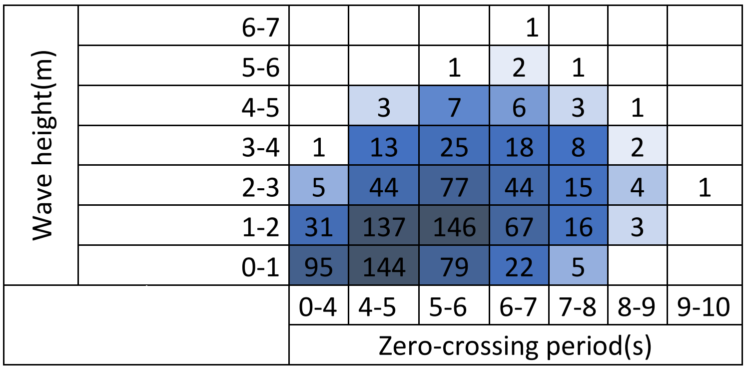
Ocean waves are produced by the wind and as such they are highly irregular. To represent this, there are statistical models that are used. JONSWAP is the most commonly used one these days, building on the older Pierson-Moskowitz spectra. Pierson-Moskowitz assumes a fully developed sea state, a sea in which the winds blew steadily over hundreds of miles for several days, but in reality this rarely occurs. Joint North Sea Wave Observation Project JONSWAP added artificial factor to the Pierson-Moskowitz to improve the fit to their measurements. [3] [4]

2.5 Response Amplitude Operators (RAOs)

Response amplitude operators are a transfer function commonly used in ship design, describing the response of the hull for the specified wave frequency and direction. Usually these are shown in separate graphs for each wave direction, which have the wave frequency in the X-axis and the RAO at Y-axis. [5]

2.6 Wave scatterplots for different areas

Wave scatterplots can be used to represent sea area’s wave height and length prevalence. Figure 1 has a scatterplot for a slightly sheltered sea area, which means it has a bit different waves more common than the larger seas. These types of plots can be used to determine the most common sea states a vessel will encounter and possible reasonable worst case situations.



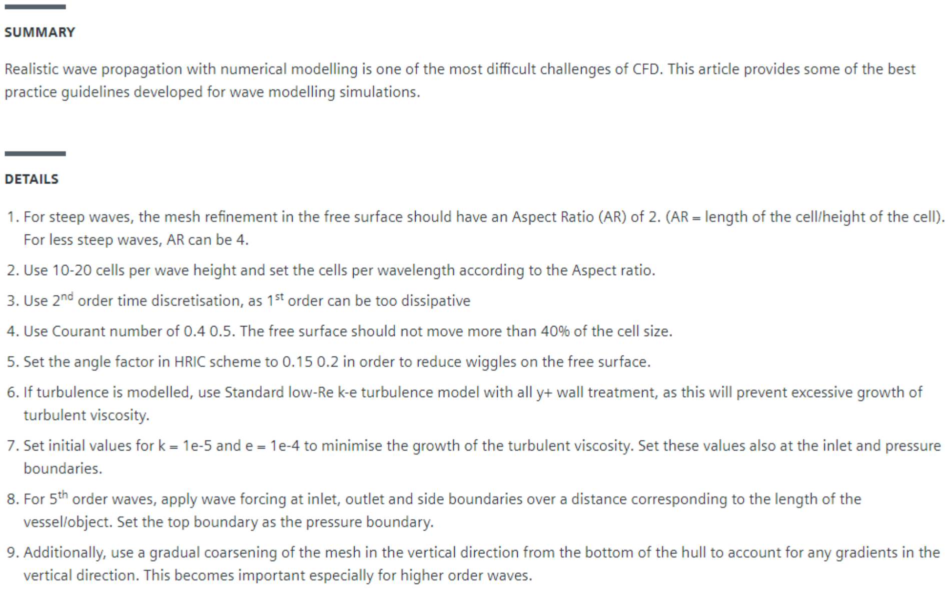
Wave height(m)	6-7				1			
	5-6			1	2	1		
	4-5		3	7	6	3	1	
	3-4	1	13	25	18	8	2	
	2-3	5	44	77	44	15	4	1
	1-2	31	137	146	67	16	3	
	0-1	95	144	79	22	5		
		0-4	4-5	5-6	6-7	7-8	8-9	9-10
		Zero-crossing period(s)						

Figure 1 Wave scatterplot example [6]

3 CFD-THEORY

3.1 Best practises from literature

Best practices for 6-dof wave simulations can be found in the documentation for StarCCM+, but it has been experienced that these are not infallible and sometimes conflicting information can be found in different articles. In calm sea state simulations the documented best practices do not always produce good enough solutions, so the accuracy of these simulations using their best practice setup is questionable, but they do provide a good starting point.



SUMMARY

Realistic wave propagation with numerical modelling is one of the most difficult challenges of CFD. This article provides some of the best practice guidelines developed for wave modelling simulations.

DETAILS

1. For steep waves, the mesh refinement in the free surface should have an Aspect Ratio (AR) of 2. (AR = length of the cell/height of the cell). For less steep waves, AR can be 4.
2. Use 10-20 cells per wave height and set the cells per wavelength according to the Aspect ratio.
3. Use 2nd order time discretisation, as 1st order can be too dissipative
4. Use Courant number of 0.4 0.5. The free surface should not move more than 40% of the cell size.
5. Set the angle factor in HRIC scheme to 0.15 0.2 in order to reduce wiggles on the free surface.
6. If turbulence is modelled, use Standard low-Re k-e turbulence model with all y+ wall treatment, as this will prevent excessive growth of turbulent viscosity.
7. Set initial values for k = 1e-5 and e = 1e-4 to minimise the growth of the turbulent viscosity. Set these values also at the inlet and pressure boundaries.
8. For 5th order waves, apply wave forcing at inlet, outlet and side boundaries over a distance corresponding to the length of the vessel/object. Set the top boundary as the pressure boundary.
9. Additionally, use a gradual coarsening of the mesh in the vertical direction from the bottom of the hull to account for any gradients in the vertical direction. This becomes important especially for higher order waves.

Figure 2 Best practices from documentation [7]

3.2 Meshing

Meshing is undoubtetly a crucial part of any CFD simulation. Even the best setup will fail if the spatial discretization has too many problems. There are a plethora of different types of mesh problems that can be interfering with the simulation, but to a point the solver might be able to cope with these. A coarse mesh is a common cause for inaccuracy in solution, but can also lead to simulation failure. Negative volume for a few cells sometimes occurs when meshing. This often leads to either simulation failure or weird flow behavior. Cell aspect ratios rarely cause failures, but can often lead to unrealistic flow patterns.

These wave simulations have similar requirements as on the calm sea state simulations, so a reasonable starting point for meshing is a standard calm water simulation mesh settings which is then fine tuned for wave simulations.

The main domain is meshed using trimmed cell mesh (also known as cutcell mesh), while the area near the vessel has a separate domain which can also be trimmed mesh, or it can be a polyhedral type mesh. The motion is done using a 6-dof (degrees of freedom) DFBI (dynamic fluid-body interaction) solver, while the interaction between the two domains is interpolated using the overset method(also known as chimera mesh).

Trimmed mesh is used in the main domain because it is fast to mesh, the mesh is aligned with coordinate system and denser area can be easily defined for the required details. Being aligned with the coordinate system provides lesser diffusivity due to mesh in comparison to polyhedral in cases where the main direction of the flow is well known.

3.3 Overset mesh

Overset mesh (also known as Chimera mesh) is a spatial discretization method, where one can have several meshes in the same volume. These meshes are then cut from background mesh and stitched into it with interpolation methods to generate a single domain. This method is mostly used in cases with large motions, e.g. several overlapping gears or in-cylinder simulations. Its benefit is also that one can have different types of meshes which are then stitched together. [8]

In calm water simulations mesh morphing is usually used for the motion, but when expecting larger motions for the vessel overset type mesh is used. For example a well designed cruise vessel will have very minor changes in sink and trim, so the morphing can handle it easily, but with a fast planing vessel overset is required.

In the top image of Figure 3 the background mesh can be seen, with its active and inactive cells in separate colors. The inactive cells are in essence “cut” out of the spatial discretization, with them being replaced by the cells in the overset domain. The blue colored cells in both top and middle images are the acceptor/donor cells, where the interpolation between the cells is done.

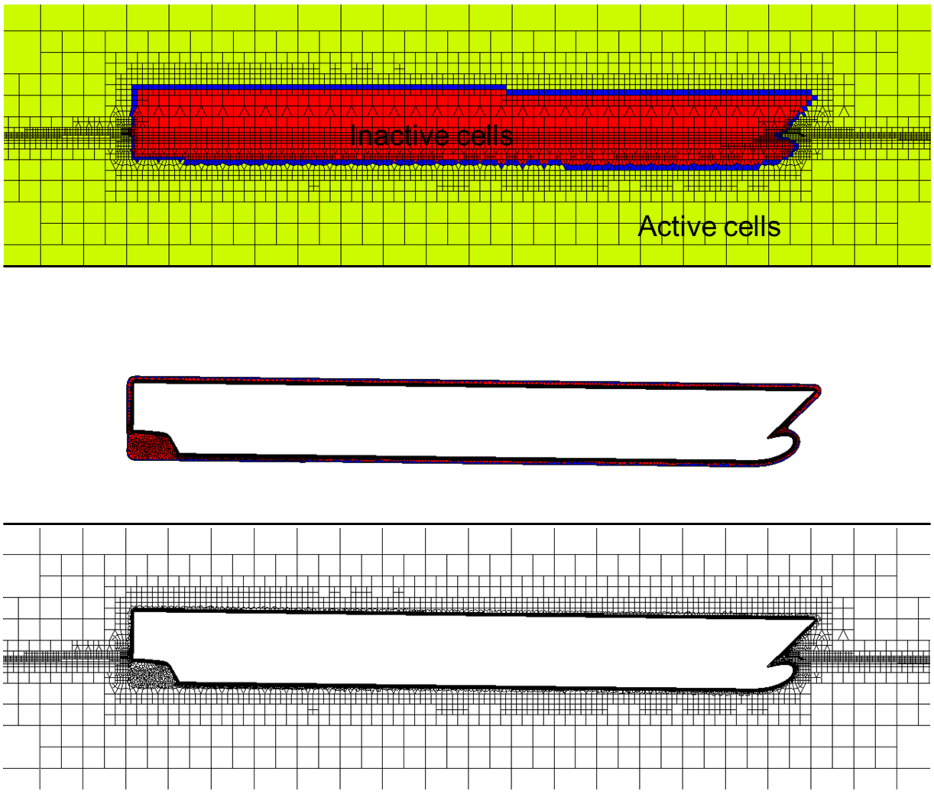


Figure 3 Overset topology. Top: Background mesh, Middle: Overset mesh, Bottom: Combined final mesh

StarCCM+ has several different interpolation options for the method used in overset mesh, but since linear interpolation was recommended in best practises [7] it was used in all cases. This is the most common method used generally, since it is a good combination of robustness and speed with enough accuracy.

3.4 Temporal discretization and timestep length

Second order temporal discretization is used in all of the cases as per the best practices documents [7] recommendations. Not much thought was spared for that from the beginning, since it should be useful with reducing dissipativeness in the domain with only minor impact on CPU time. In the future additional cases could be run with comparisons between first and second order, to see the true impact on results and solution time.

Timestep is one of the most important factors in unsteady simulations, with it having a large impact on pretty much everything going on in the simulation. As a baseline the timestep listed in best practices was used, but was quickly found to be too large in the cases run. Reducing the timestep to around wavelength/1000 provided more stable simulation. Reducing the timestep further was tested, but the turbulence modeling started having trouble and the CPU time became unfeasibly large.

3.5 Turbulence modeling

Modeling turbulence is important in some of the CFD cases, but for estimating hull performance it is crucial. The turbulence provides important parts of the hull resistance. Without it the performance predictions are based on extrapolation from the wetted surface area, which doesn’t fully capture the phenomena. Only in recent years has it been possible to simulate larger hulls in full-scale with turbulence modeling.

As in the best practices [7], Low-Re k-ε turbulence model with all y+ wall treatment was used. This provides a more robust turbulence model for these initial simulations, for increased accuracy Realizable k-ε or SST k-ω should be tested. Especially SST k-ω has proven to be accurate and robust in calm water simulations and is mostly used in those types of simulations.

Low-Re k-ε is the most commonly used RANS turbulence model and has been the industry standard for a while now. It describes the turbulence with two transport equations, the first variable being turbulent kinetic energy (k) and the second is rate of dissipation of turbulent energy (ε). Realizable k-ε adds an additional realizability term to the transport equations, which provides improved prediction for spreading of jets and superior performance in flows involving rotations and circulation.

k-ω is similarly a two-equation turbulence model, with the first term being the same as in k-ε and the second being specific rate of dissipation (ω). The k-ω model itself has limited use as it quickly becomes unstable in free-stream areas of the simulations, but provides increased accuracy near walls compared to k-ε. SST k-ω (Shear Stress Transport) model combines the best of both worlds. It uses the k-ω near the walls for increased accuracy and blends it into k-ε in free streams for its increased stability. [9]

3.6 Wall functions and Y+

Wall functions are used in CFD to model boundary layer flow when the mesh isn't dense enough on the surface of the boundary to fully solve the flow phenomena. y+ is a non-dimensional distance describing the boundary layer meshe's first cell height to the flow charasteristics.

$$y^{+} = \frac{yu_{t}}{v}$$

y = absolute distance from wall

u_t = friction velocity

v = kinematic viscosity

Equation 2 Wall y+ [10]

In calm water full scale hull simulations wall y+ of around 100 is usually aimed at. Aiming for fully solvable (y+ <5) viscous sublayer is infeasible as the meshes become too large to simulate in reasonable time when considering larger (>100 m) ships. StarCCM+'s best practices often say to avoid y+ of 40-60, so aiming for average of 100 gives error margin for differences in the hull. To achieve this, commonly the first cell height is between 5-20 mm for the larger vessels.

In these simulations the boundary layer mesh is done similarly, as with the calm water that provides the y+ of 100, but of course the ships motion in waves changes the y+ from moment to moment.

4 SIMULATIONS

During the project the simulations were split into two phases: Phase 1 simulations were for a simple cylindrical wave breaker, the Phase 2 were simulations for a ~200m seafaring vessel. Most of the simulations were done using regular waves, with a few of the latter ones being in irregular waves.

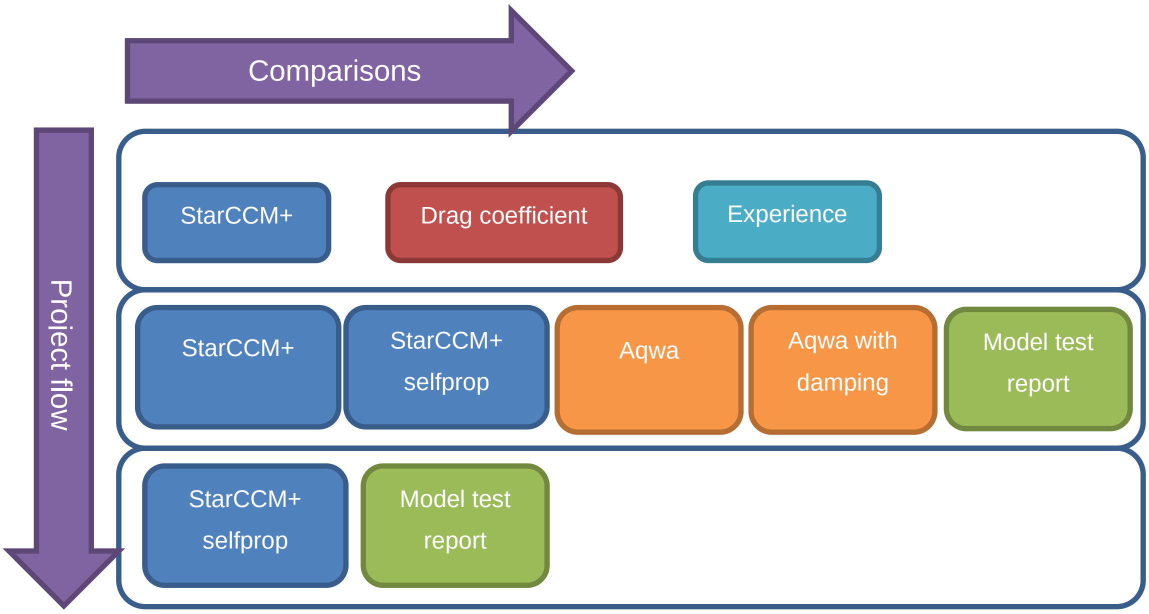


Figure 4 Project visualization

5 WAVE BREAKER SIMULATIONS

Wave breaker was chosen for the initial simulations due to the more simple nature of the required simulation. No motion or interfaces were required, so we could concentrate completely on figuring out the parameters and requirements to best simulate the wave itself. Elomatic has an offshore team, who are experienced in designing these types of installations, so the dimensions and wave parameters were chosen to be similar to a design case they have.

5.1 Problem setup

The initial case was chosen to be of a wave breaker near a beach, with a gradual incline into the land. The best practices mentioned in StarCCM+ documentation were used as a starting point for the solver set-up, which was then iterated over until a successful method was found. Wave breaker drawing which defines the geometric parameters is Figure 5 with the parameter values listed in Table 1.

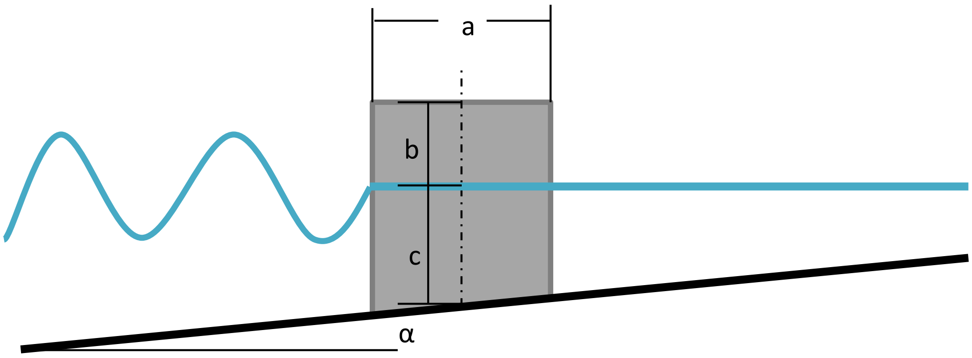


Figure 5 Wave breaker drawing

a	4 m
b	1.5 m
c	2 m
α	4°

Table 1 Wave breaker dimensions

Wave parameters were chosen based on the scatter diagram of the design area, these are listed in Table 2. These values give fairly long waves, this was done to simplify the case by limiting the chance of wave breaking.

Wave height	1 m
Wave length	4 s (~23 m)

Table 2 Wave parameters for wave breaker simulation

5.2 Simulation setup

Simulation boundary conditions and simulation domain size are shown in Figure 6, the symmetry side wall condition was chosen to represent a line of the wave breakers protecting the shore. Velocity inlet with forced zero velocity is used as the inlet condition to force all of the motions to be induced by the wave motion and not by current. The wave itself is generated by the time dependant volume fraction at the velocity inlet, the height of the sea surface changes based on the defined wave.

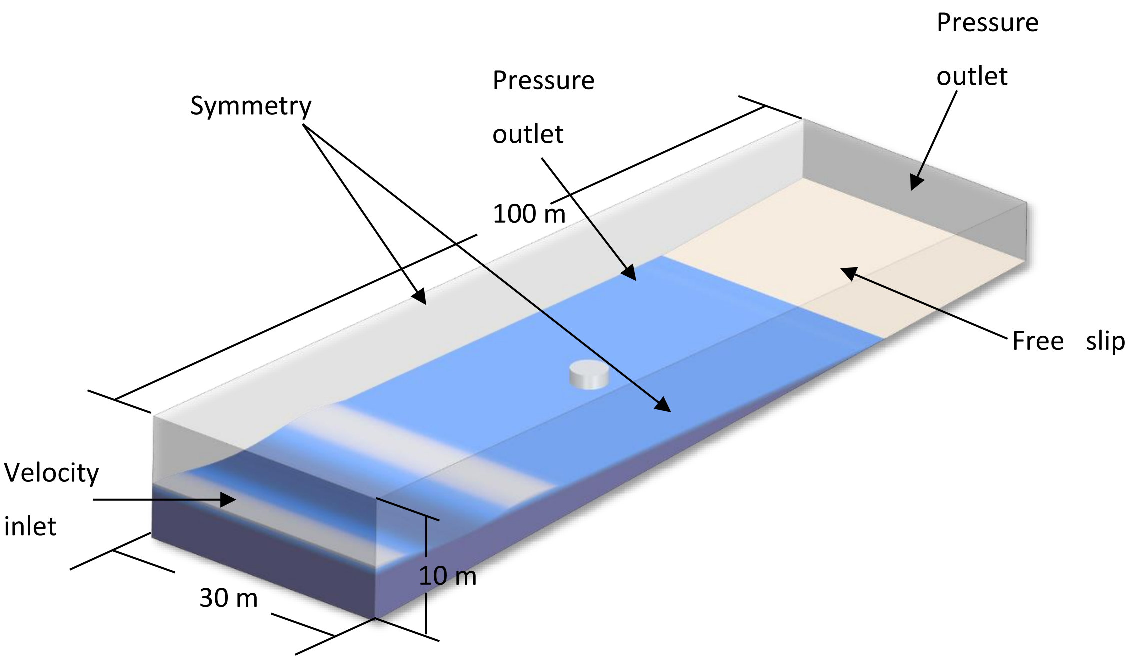
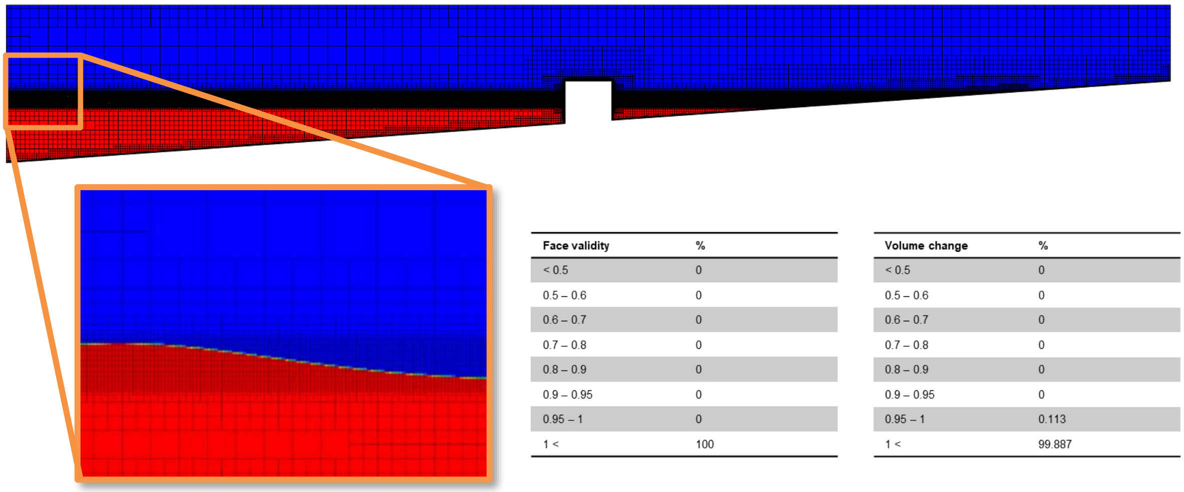


Figure 6 Wave breaker simulation boundaries

In meshing the case special consideration is placed upon providing high quality and dense mesh in the volume around the wave surface. There are 20 elements in the vertical area of the wave with additional 5 to each side. Aspect ratio for the cell in this area is strictly set as 2. These values are directly the same as with the documentation best practices. The mesh has around 9.5 million cells, which gives reasonable simulation time with the available computational capacity, while being dense enough to accurately solve the simulation.

The simple nature of the geometry made it easy to generate high quality mesh for the simulation. In Figure 7 we can see that the face validity is above 1 in all of the cells in the domain and volume change being also almost perfect.



Face validity	%
< 0.5	0
0.5 – 0.6	0
0.6 – 0.7	0
0.7 – 0.8	0
0.8 – 0.9	0
0.9 – 0.95	0
0.95 – 1	0
1 <	100

Volume change	%
< 0.5	0
0.5 – 0.6	0
0.6 – 0.7	0
0.7 – 0.8	0
0.8 – 0.9	0
0.9 – 0.95	0
0.95 – 1	0.113
1 <	99.887

Figure 7 Wave breaker meshing

The case 2 was a similar sized wave breaker simulated in deep water, with a comparable case done without the wave breaker.

5.3 Results

As previously stated there wasn’t any data available, to which a comparison from these results could be done. So the main targets for these initial simulations were to get a simulation working, getting decent looking results and to be on the same order of magnitude as the drag force equation.

Figure 8 has a snapshot image of the simulations and the appendix 1 has complete animations for the entire simulations. These visualizations match well the expectations and provided additional support that the simulation model is working as intended. In the animation some fairly small splashing can be detected, even if the mesh density isn’t enough to capture all of the phenomena.

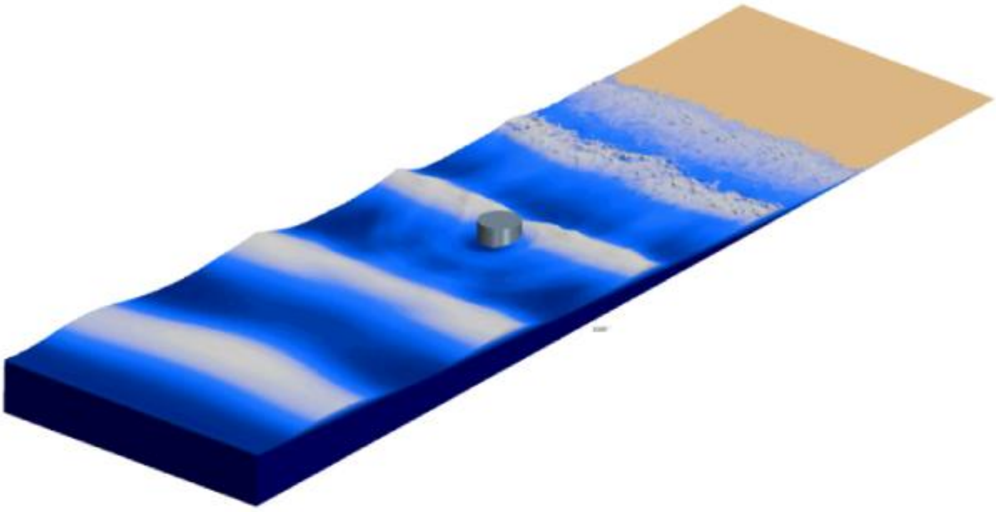


Figure 8 Wave breaker snapshot

The drag force generated by the wave was compared against values gotten using a drag force coefficient, as described in 2.3. These are very rough estimates, but they show that the results are on the ballpark even if much larger than what was to be expected. The Elomatic Offshore team commented that the values shown in Table 3 seemed reasonable in their experience.

	C_D	F(KN)
Sphere	0.47	31
Box	1.05	70
CFD		96

Table 3 Drag forces of wave breaker simulation

The second case for the wave breaker was to simulate it in deep water, with a comparable case done without the wave breaker to see its effect on the wave. In Figure 9 we can see the surface pressure on the wave breaker on the instant that the wave is hitting the surface. The top one being the pressure on the forward side and the lower one being the backside of the model.

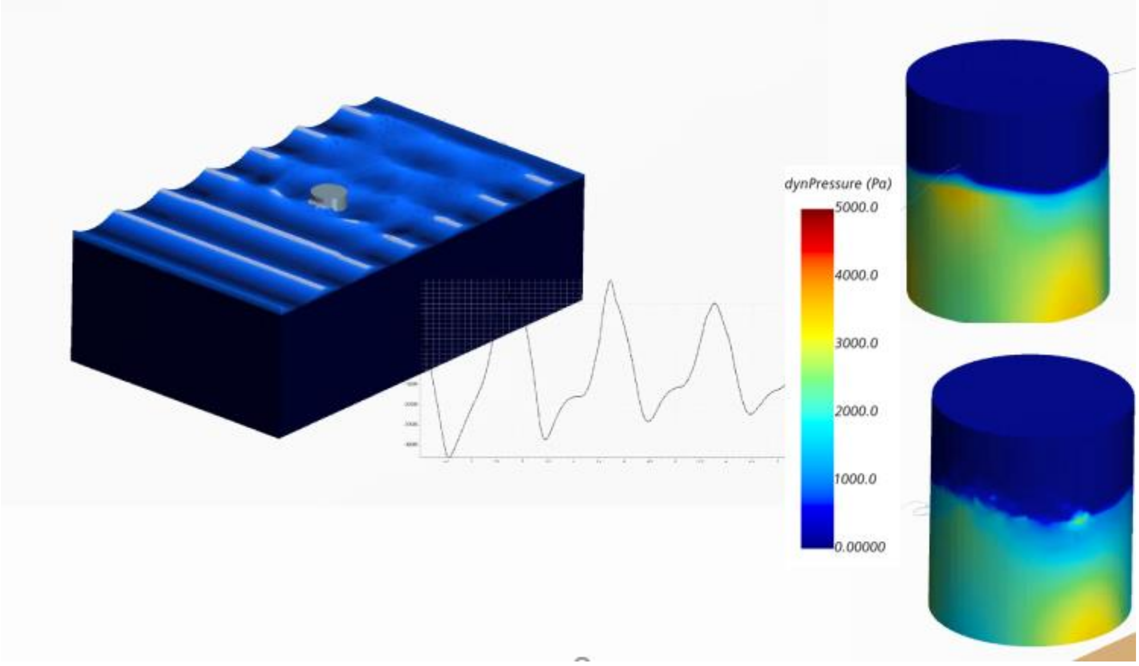


Figure 9 Deep water wave breaker simulation

The force of the wave oscillated between 24 kN and -24 kN, which was still higher than expected, but closer than in the previous case. The oscillation between the same maximum in each direction was to be expected, with similar phenomena having been detected in regular ship design.

	C_D	F(KN)
Sphere	0.47	10
Box	1.05	23
CFD		24

Table 4 Drag forces for deep water wave breaker simulation

Wave breaking capability doesn’t have a set performance index currently, so quantifying and comparing the wave breakers isn’t easy. In the future such index would be useful, but for now settling for visually estimating the performance is a necessity. In Figure 10 one can see that the current wave breaker design reduced the wave height in its wake from 1m to approximately 0.5m.

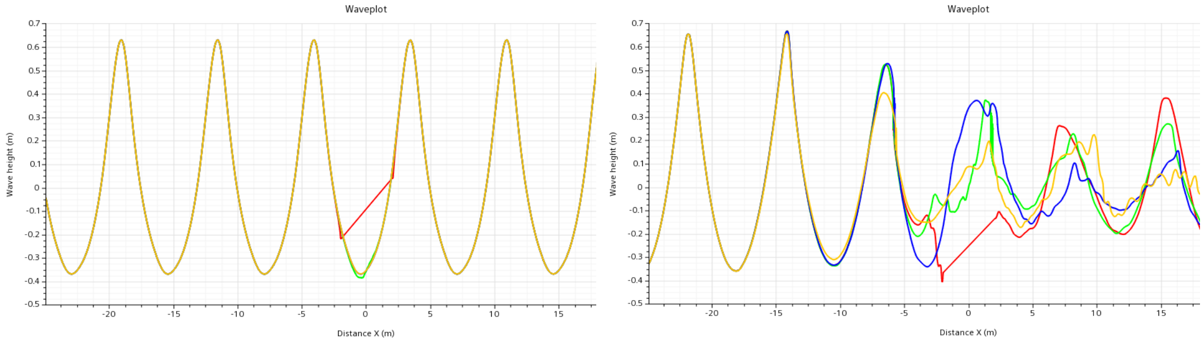


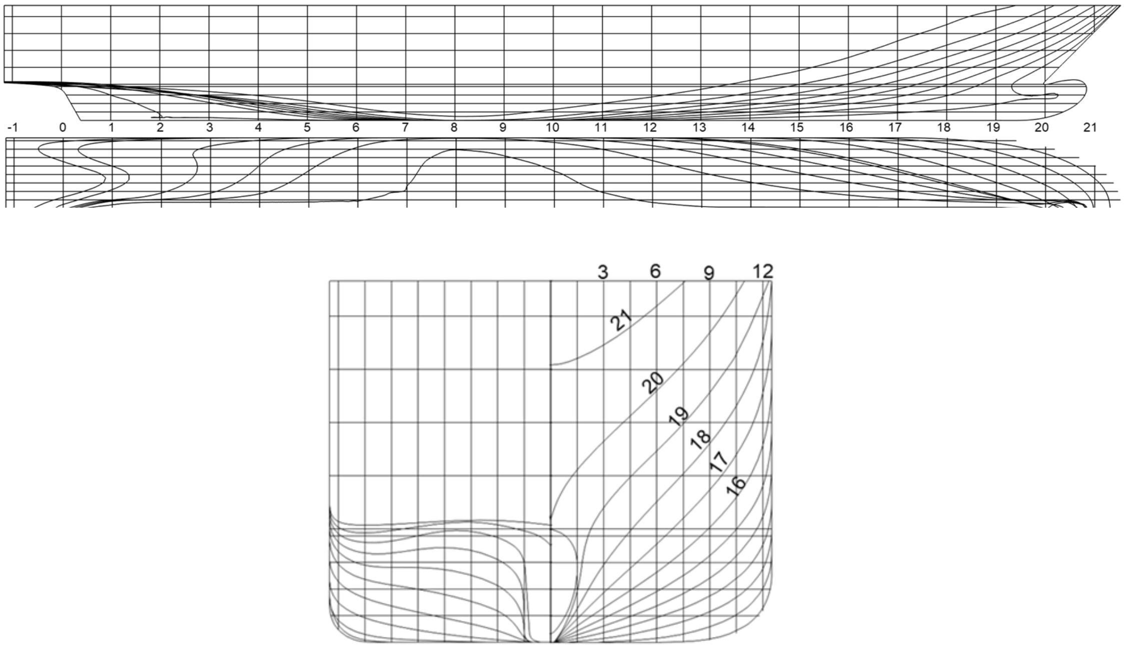
Figure 10 Wave height at different cross sections

Even with such rough estimates and poor comparable data, these initial results gave confidence in the project to move forward to more complex simulations with better comparison data.

6 SEA FARING VESSEL

For the Phase 2 of the simulations were carried out for a 200 meter long sea faring vessel. The vessel chosen is Superfast III (and sister ship Superfast IV) which is a Ro-Pax -type vessel completed in 1998 Turku shipyard. Full hull model and Napa model was available for this use, as well as MARIN's model test reports for maneuvering, seakeeping and calm situations were available to use for comparison. Being an older vessel is also beneficial, because there is more in use experience to reflect upon.

The hull shape can be seen in Figure 11, as well as some basic values for the vessel. The hull shape is mostly fairly common for a faster ferry; slightly up turned medium sized bulbous bow, smaller than average flat bottom, and fairly light S-shape in aft body. The main distinctive charasteristics of the hull is the wave damping afterbody, which can be seen around frame 0 and 1. This provided additional benefits that the model test could not predict and were only discovered during sea trials.



Lpp	174.8	m
Bwl	25.0	m
T	6.4	m
Dp	5.1	m

Figure 11 Superfast hull lines and charasteristic dimensions

6.1 Problem setup

Initial set of simulations were run for a series of regular waves, as listed in Table 5. Ansys Aqwa simulations were computationally inexpensive, so they were run for all of the same situations as the model test. StarCCM+ simulations were only run for forwards directed waves.

HEADING(DEG)	SPEED(KN)	FREQUENCIES(RAD/S)	WAVE HEIGHT(M)
180	24	0.5, 0.55,0.6,0.7,1.0	2
135	24	0.6,0.65,0.7,0.85,1.1	2
65	24	0.75,0.9,1.0,1.05,1.1,1.2	2

Table 5 Regular wave cases

The model test model was fitted with all of the relevant appendixes, with fin stabilizers efficacy being tested on some of the cases. Figure 12 has technical drawings of the rudder, the axle, its struts, bilge keel and fin stabilizers.

To keep the simulation model simplistic these were left out of the geometrical model, but were modeled with coefficients in the relevant cases.

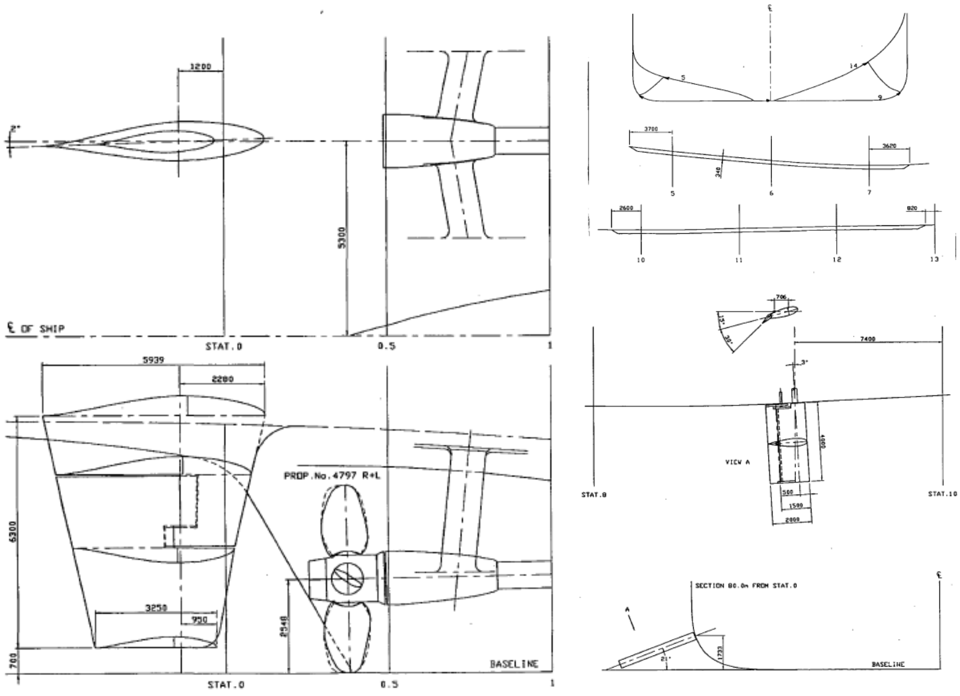


Figure 12 Technical drawing of Superfast's appendices [11]

In the chapter 6.3 comparisons are made between the followingly named data:

1. Modelt test
    - Results taken from the MARIN’s model test report
2. Aqwa
    - Results taken from the Ansys Aqwa simulation result
3. Aqwa with damping
    - Results taken from the Ansys Aqwa simulation result
    - Bilge keel and fin stabilizer modeled using coefficients
4. StarCCM+
    - Results taken from the StarCCM+ simulation result
    - Hull is forced to move at the specified velocity
    - Surge is fixed
5. StarCCM+ selfprop
    - Results taken from the StarCCM+ simulation result
    - Propulsion is modeled using an actuator disc model
    - Propulsor moment is fixed so that the vessel on average moves at the specified velocity

6.2 Simulation setup

The boundary conditions used are shown in Figure 13, with the boundaries in the fore and side of vessel being velocity inlets and the top, bottom and aft boundaries being pressure outlets. In the initial case with the static surge both the vessel and background domain are stationary with the vessel velocity being modeled with the inlet velocity. In the self-propulsion case the velocity at inlet is set as zero, the vessel is moving in surge direction by its own propulsion and the background domain is following with same velocity in horizontal plane with only that directional motion allowed.

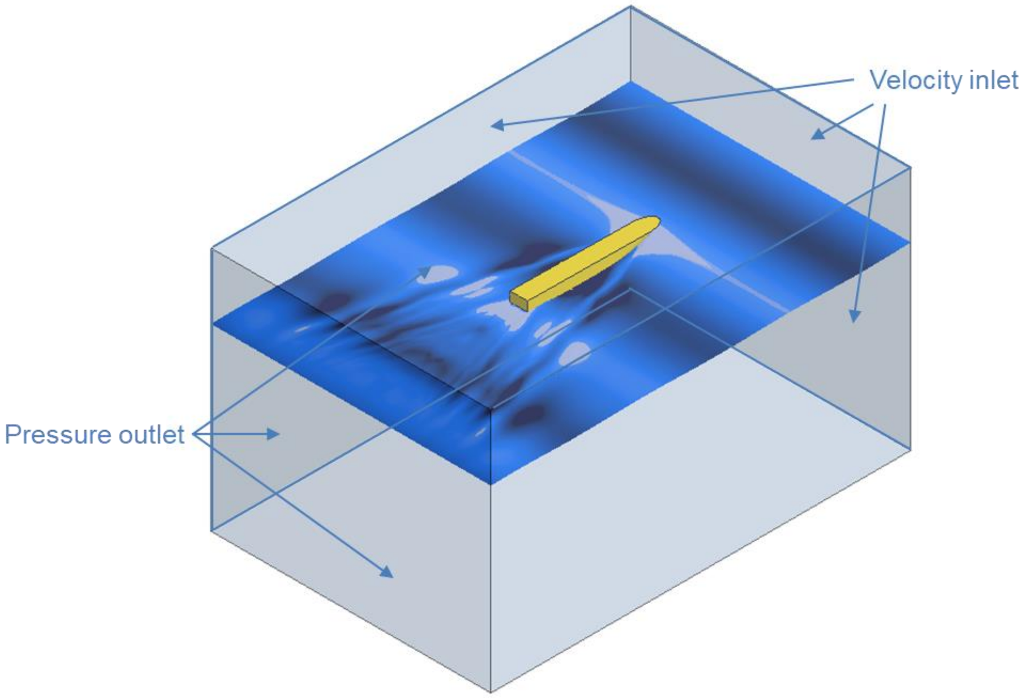


Figure 13 Boundary conditions for regular wave simulation

Handling pressure relfections from the boundaries is a crucial part of even the calm water simulations. With the waves added these can be even more problematic. To combat the pressure waves bouncing back and forth, there is an added wave damping method. This is only used in the boundary in the aft of the vessel, so that it smooths out the unneeded reflections, but doesn’t affect the solution in other ways. The damping method adds a source term that dampens the motions, increasing in effect nearing the boundary.

On the side and fore boundaries there is added wave forcing method, which helps the wave to keep its shape inside the domain and to reduce the diffusivity of the mesh.

Without such method the waves diffuse into flatter waves inside the domain, unless a much denser mesh is used. Even with unfathomably large mesh, there is more diffusivity than with the forcing method due to the boundary modeling.

The domain itself is very small compared to what we would use in a calm water simulation. In those simulations the aim is to have the boundaries far enough so that they do not have an effect on the solution. In wave simulations the damping and forcing methods are used to negate this effect. Larger domain downstream would be beneficial in some ways, but could also provide additional problems, as well as increase the CPU time required.

Figure 14 has the approximate damping and forcing areas marked, they are gradual in nature, so their effect near the marked lines is very minor increasing near the boundary.

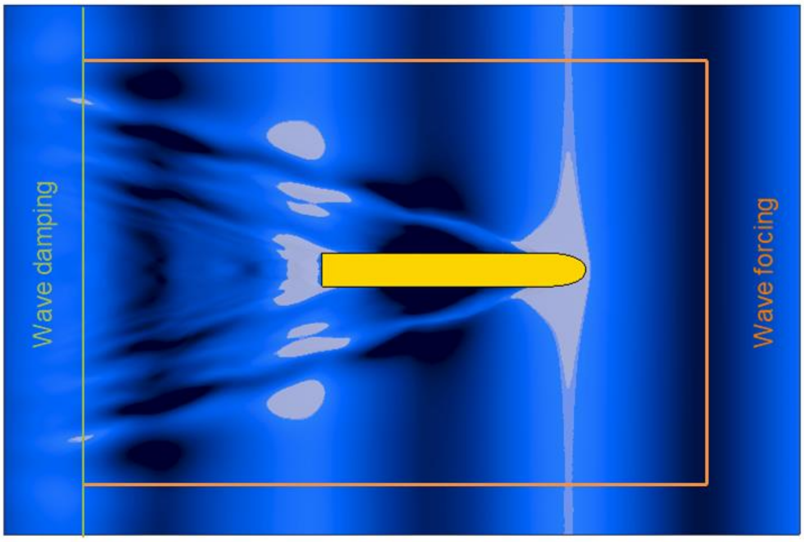


Figure 14 Wave damping and forcing areas

6.3 Results

Visually the StarCCM+ cases seem reasonable, with the wave being modeled nicely and being able to catch some of the wave breaking and splashing in the fore. Being able to accurately simulate the fore splashing would be beneficial, since the green water amount is often a critical consideration in ship design and it is difficult to estimate using model tests. In Figure 15 we can visually inspect and see that the motions and angles of the vessel are similar between StarCCM+ and Aqwa.

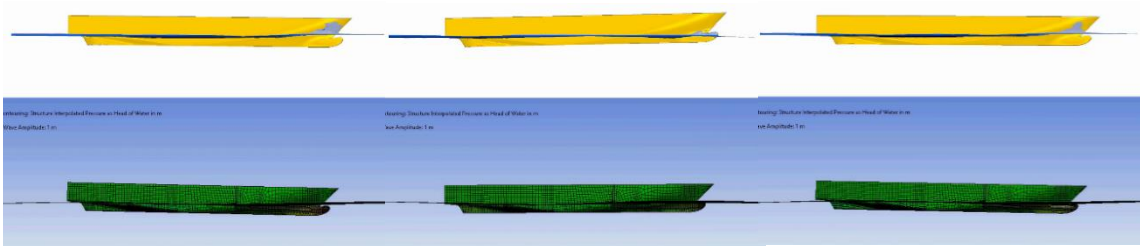


Figure 15 StarCCM+ and Aqwa side view visualization

Figure 16 and Figure 17 has comparison between the 4 simulation cases and model test results for regular fore waves. In both pitch and heave the Aqwa simulations differ on the lower frequencies, nearer the natural period of the hull (0.08 Hz, 12.5 s). Both of them match the model test results in higher frequency range.

StarCCM+ with the forced forward motion is closer to the Aqwa results than the model tests, but the self-propulsion case is very near the model test results even near the natural period.

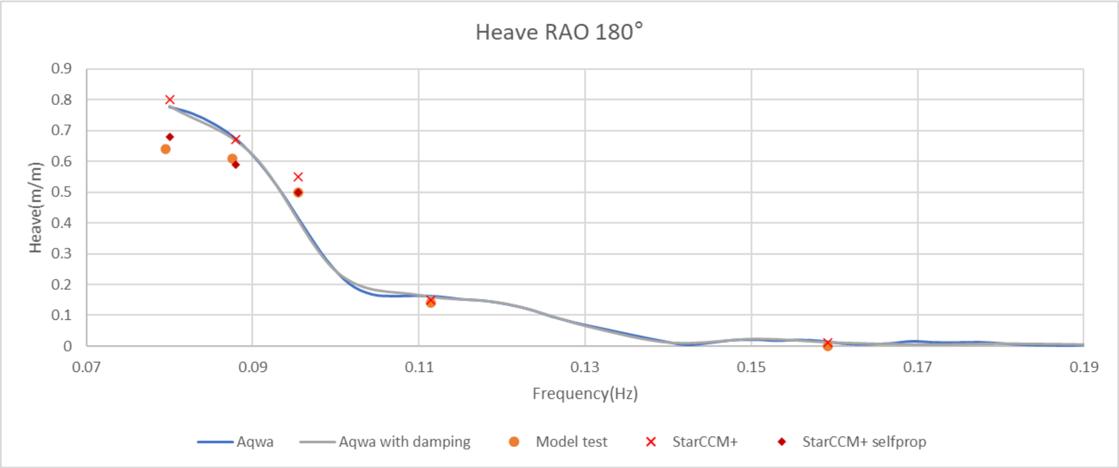


Figure 16 Heave RAO comparison for fore waves [11]

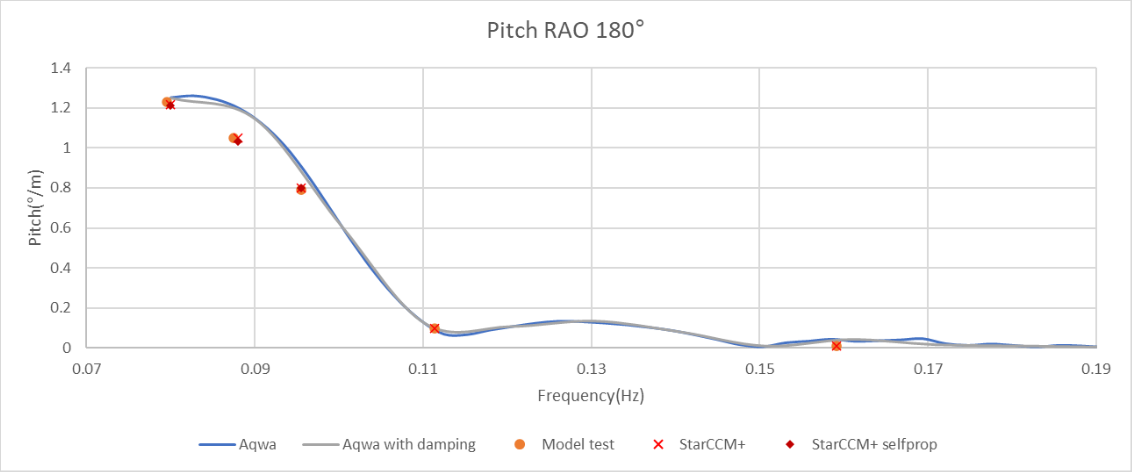


Figure 17 Pitch RAO comparison for fore waves [11]

The simulation results are in good agreement with the model test result, with the self-propulsion StarCCM+ model being clearly the best solution. Still there is some difference, but there are many contributing factors that could have caused that.

Even with the decades of fine tuning and experience on the model testing, they still do have their inherent limitations on accuracy, so some of the difference can be due to the error margin of the model test itself.

The simulations were done without the appendixes, so that could be another contributing factor. Their effect is usually fairly minimal in general and especially in fore waves, but additional research should be done on their effects.

Mass inertias were extrapolated from the geometry model, so there will be some difference to the model test, but both of them will be somewhat different from reality. This too should be fairly small influence, but together with the other factors it could be noticeable.

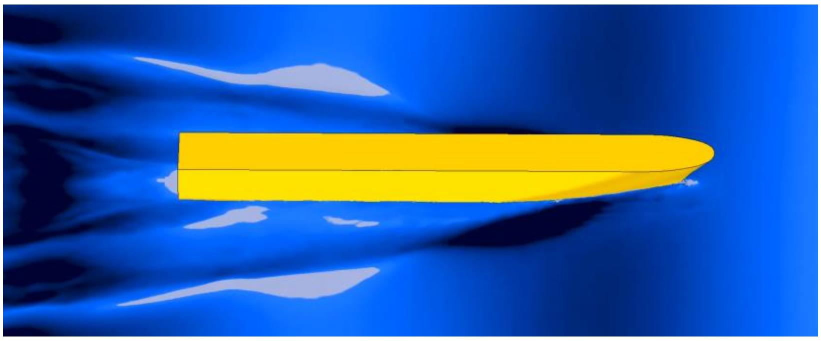


Figure 18 Regular wave visualization from StarCCM+

7 SEA FARING VESSEL IN IRREGULAR WAVES

The same vessel that was used in chapter 6 had also model tests done in irregular waves, with estimation on its performance in two different sea states. The sea states are the same as the ones defined in the model test report, but due to the irregular nature of the waves there is inherent difference in the cases. This could be compensated with averaging results over longer simulation time, but due to time constraints in the project the simulation time was limited. This means that the results are not that scientifically accurate, but more serve as a proof-of-concept that it is possible to simulate ship performance in irregular waves.

7.1 Problem setup

Both of the simulation cases are for fore waves, with different peak periods and significant wave heights to represent the different sea areas. The fore wave direction is to get the wave added resistance, which will provide good comparison point for the results. The speed listed in Table 6 are for the model test, the simulation model aimed to have velocity as close as possible, but due to the self-propulsion model it varied slightly.

HEADING(DEG)	SPEED(KN)	SEA STATE	SIGN. HEIGHT(M)	PERIOD T_0(S)
180	14.7	S.North Sea	8.5	12.9
180	23.8	Adriatic	4.0	9.8

Table 6 Irregular wave cases

As discussed in chapter 2.4 JONSWAP is the most commonly used spectra for modeling common sea states, as such it was used by MARIN in the model test. The model test basin cannot completely accurately reproduce the theorical spectra and the difference can be seen in Figure 19. The simulation also has some diffusivity in the waves inside the simulation domain, so it also has some discrepancy from the theoretical spectra, although much closer than the model test.

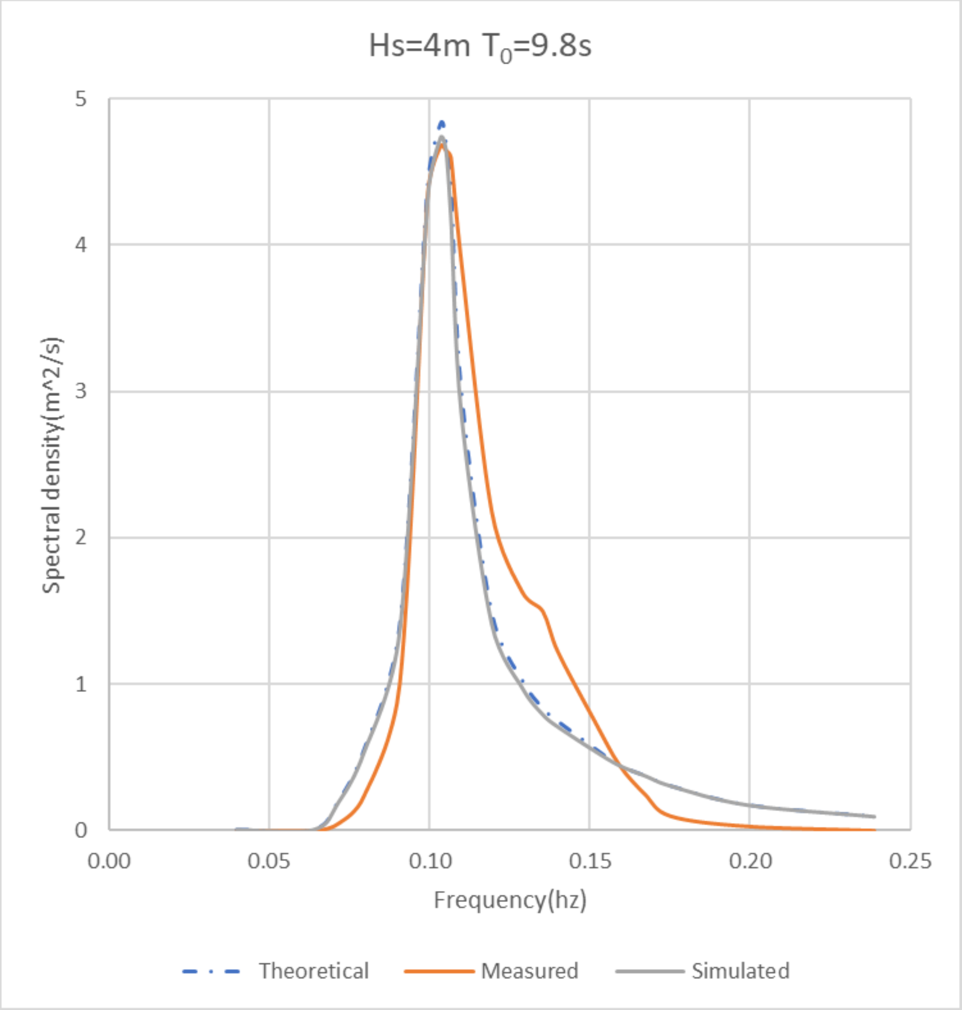


Figure 19 Comparison of used spectra

7.2 Simulation setup

The simulation set-up built upon the self-propulsion case of chapter 6.2, with the majority of the case being the same as previously. The major addition is the usage of automatic mesh adaptation. This method periodically remeshes the case during simulation, generating denser mesh where needed.

Without the adaptive mesh refinement model, the mesh generated especially for the larger irregular waves would be massive. This would in turn make it computationally very expensive to run the simulation in a reasonable time frame. With the AMR method the simulation model had 4 million cells, which is nearly the same usually used in calm water simulation. The adaptation of course slows down simulation time, but still being reasonably fast. Running the case overnight on 200 cores led into around 100-150 seconds of simulation time.

The adaptation used two criteria in the simulation: Overset and VOF-interface criteria. Overset adaptation refines the mesh around the overset interface area, checking to see if the hole cutting and interpolation is working on the interface. VOF-interface adaptation refines the mesh near the wave surface, trying to keep sharp interface between the phases. This method also has a swept through function, which means it will refine the mesh also in the direction the wave surface is going. This makes it so that it doesn't need to run the refinement cycle as often. In Figure 20 areas one can see the overset refinement working in the areas marked as '1.' And the VOF-interface refinement marked '2.'.

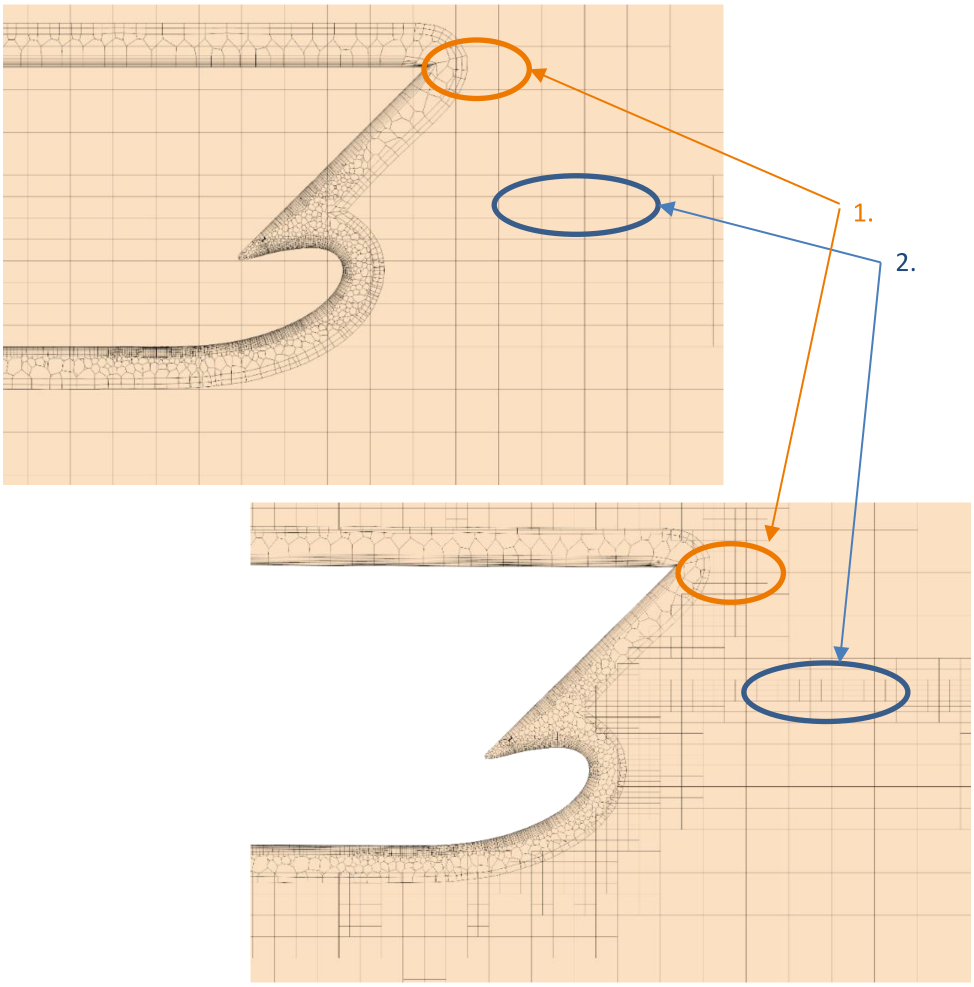


Figure 20 Adaptive mesh refinement example

7.3 Results

In Appendix 1 there are animations of the two irregular wave simulations, with Figure 21 being a snapshot of one of them. Visually inspecting the simulations they look reasonably realistic, with the wave remaining tighly in the proper form and with kelvin wake showing nicely. During slamming event the water rises on the fore, but greenwater (i.e. the water entering the fore deck in waves) isn't really detected in these cases. Realistically in the S.North sea case atleast some wetting of the fore deck should be seen, but it is not seen in the animation. This is probably due to a combination of two things: the mesh on the deck might not be dense enough to properly model the phenomena and the post processing using a isosurface isn't well suited to show it.

The amount of greenwater is of high interest for the shipyards, being a crucial factor in the fore structural design. Model test cannot be used to estimate that, so accurate estimation with CFD would be highly beneficial for ship design. Certainly in the future this is one important area for further research.

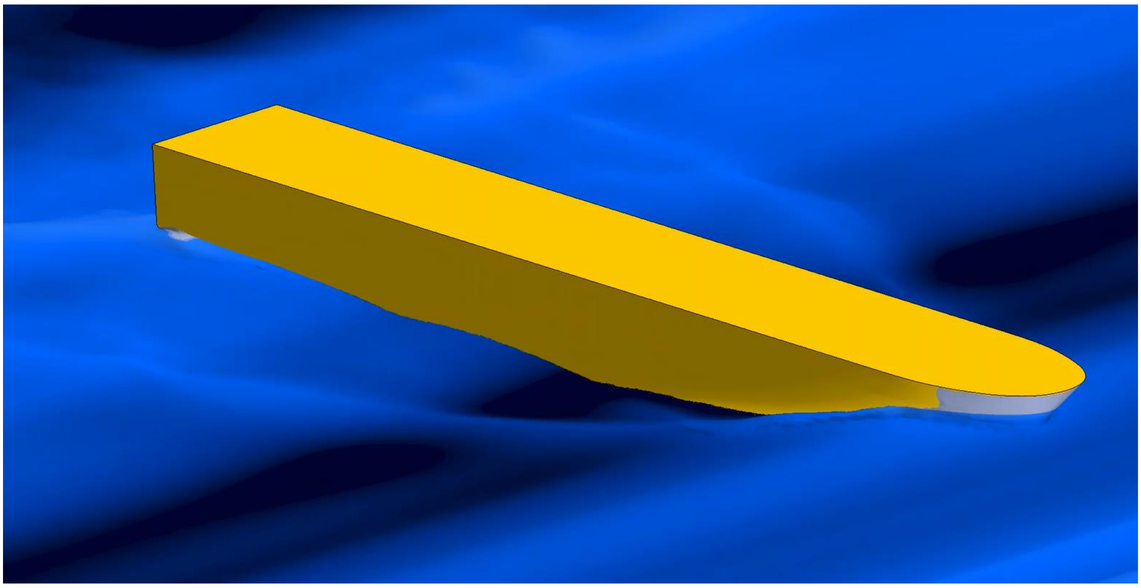


Figure 21 Irregular wave simulation visualization from StarCCM+

The main comparison is done by comparing the thrust requirement in the two sea states and the wave added requirement. In Figure 22 is the calm water results between the model test, simulation and sea trial corrected model test results.

The model test results are the estimation of the performance during the finishing phases of the vessel design, with the corrected ones being done later corrected with the results of the sea trial. The disparity between the two is due to two main factors: The new wave damping after body and them being rather conservative. Back then when this vessel was being developed, the wave damping after body was something new, so there wasn't any data to use for extrapolating the model test results. Also it is often better for the vessel to surpass the expectations in sea trials, so the reported performance estimation is overly conservative.

Using StarCCM+ simulation for estimating the calm water performance is common, with well validated and often used methods. So the simulation results being near the sea trial corrected results was to be expected.

All of the results shown in figures Figure 22, Figure 23, Figure 24 are for the hydrodynamic resistance/thrust, the aerodynamics are left out of this comparison. The aerodynamics is a separate topic that doesn't affect this comparison, but would be necessary for estimating the actual power requirement of the vessel. The estimated deliverable thrust is shown in the figures just to give idea of the maximum velocity of the vessel, in reality with the aerodynamics being considered the actual response would be lower than the ones shown.

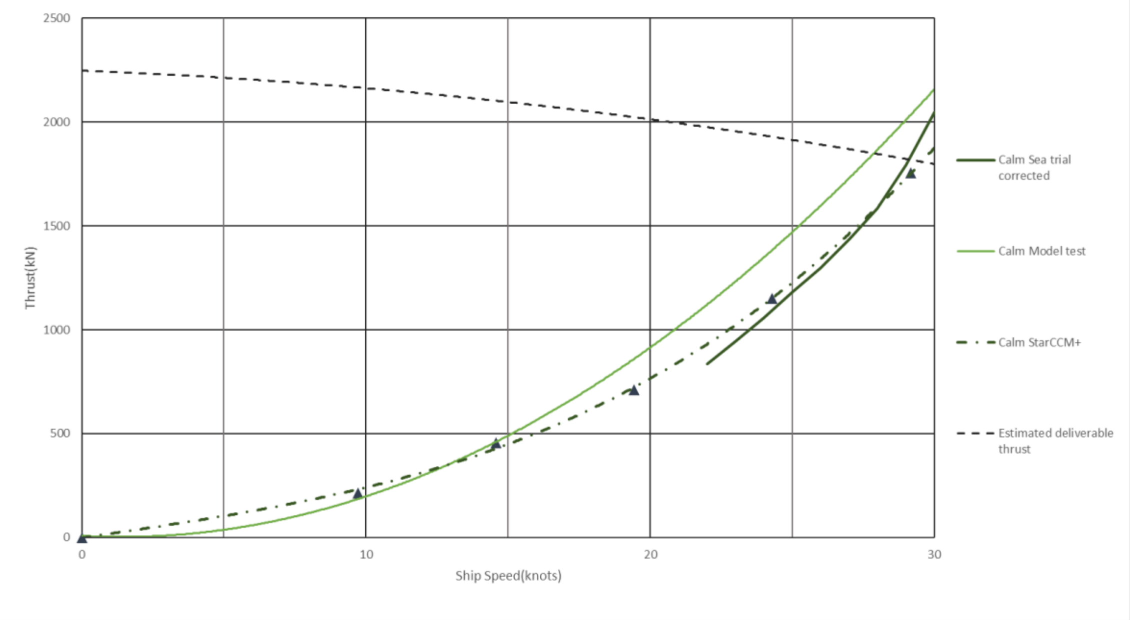


Figure 22 Thrust in calm water [11]

The wave added resistance, I.E. the difference between calm water resistance and resistance in waves, is very closely the same between model test and simulation as seen in Figure 23 in the Adriatic sea case. The wave added resistance is visualized with wave brackets in the following figures, with the distance between the top and bottom being the added resistance.

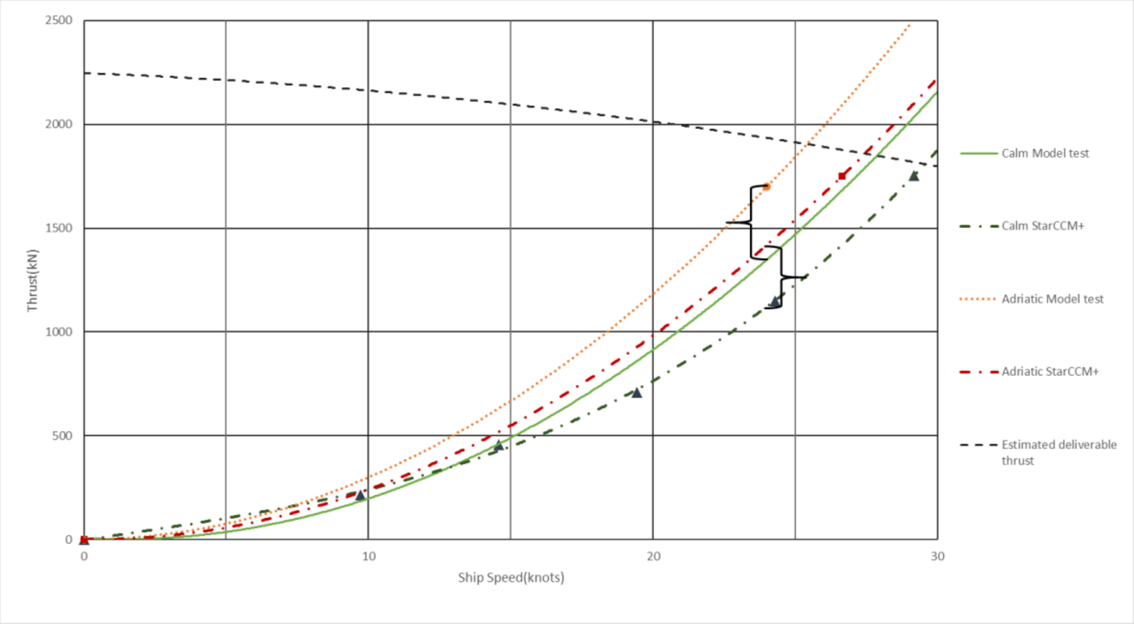


Figure 23 Additional thrust required due to waves in Adriatic sea [11]

A much larger difference can be detected in the S.North sea case, as seen in Figure 24, with the wave added resistance in model test being twice of what the simulation would predict. The larger waves are harder to both simulate and to test in model scale, so there is larger error margin in both of those results.

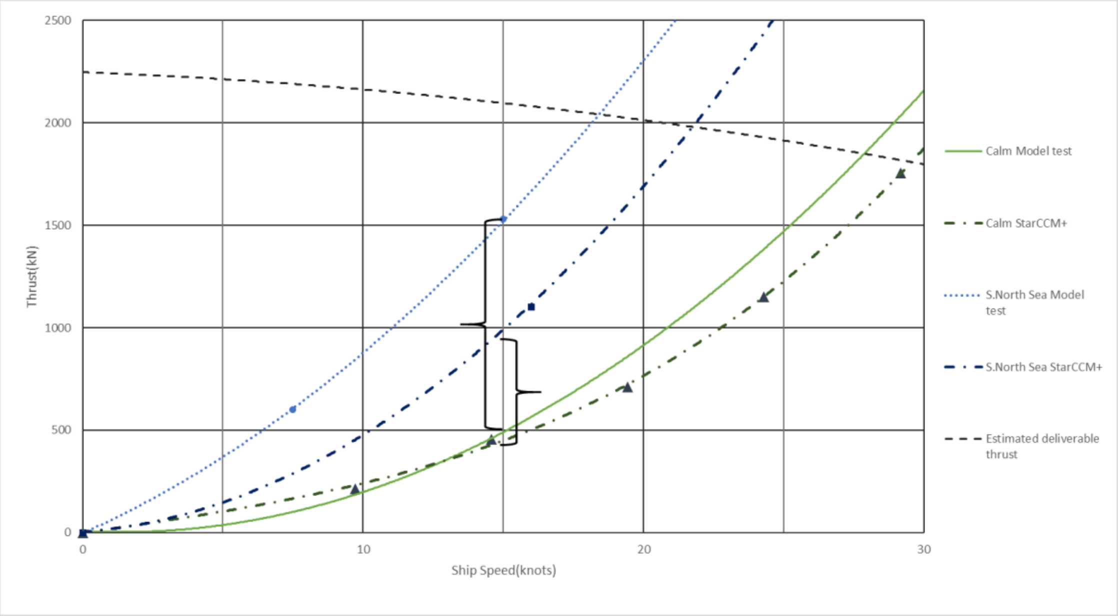


Figure 24 Additional thrust required due to waves in S.North sea [11]

There are a multitude of factors contributing to the difference in results between simulation and model test, considerin that the Adriatic sea results are surprisingly close. In that case it could be that a couple of different errors are cancelling each other out, so there it isn’t really clear if these results can be fully trusted.

The model test result itself has an error margin. As seen in the calm water results it can be quite significant too. The larger waves are harder to evaluate as well as the slower velocity makes it more difficult to accurately predict the performance.

As discussed in 7.1 the irregular waves are modeled using a spectra, the model test cannot match the theoretical spectra exactly so there is difference between the model test and simulation too. The simulation is closer to the theoretical spectra, but that too misses the mark slightly.

The appendices are missing from the simulation model, this too contributing to the difference in model test. Usually the appendices effects are negligible in model test, but that can be a charasteristic of doing it in model scale. This can be corrected in the extrapolation functions the model testing facility is using, but it can be inaccurate. In reality they can have an effect on these results, but there isn’t data to make any comparisons against.

Mass inertias were extrapolated from the geometry using a rule-of-thumb and compared against the ones used in model test. Both of these are probably different from the actual finished vessel's values, but they should be on the correct magintude. Their effect should be a minor one, but this should be checked.

Due to limited resources during the project the simulations were run only for around 100-150 seconds. The model test reports values over 700 seconds, with more recently the standard becoming around 3 hours. Averaging over only the 100 seconds might not provide enough data for accurate prediction.

Considering all of these contributing error factors, the results are in surprisingly good agreement with the model test results. As an initial proof-of-concept, these simulations have provided high amount of trust for the feasibility for its usage in the future.

8 FUTURE RESEARCH

There are several different avenues for future research, where this research project has provided some ground work for. The major split is between the regular and irregular wave simulations.

For the regular waves the natural next steps would be to simulate additional wave directions, thus providing additional insight on the vessels response on those situations. This could be useful comparison also to Ansys Aqwa, since it is more lightweight approach for the same problems. If the simulations can be done quickly enough they could be used by themselves, or running a few key simulations and using those in conjunction with Aqwa simulations for more accurate predictions.

Comparative cases with different turbulence models would be a highly interesting future study. Choice of turbulence model is very impactful and finding the optimal balance between stability and accuracy would be beneficial.

Geometrically modeling the appendices would be another interesting simulation set. In model scale their effect is usually negligible, but in reality it could have some effect. There isn’t really a way to get solid data to compare against, but even seeing the change if any would be of interest.

In irregular waves the cases should be run longer for more statistical data, and some additional velocities would be beneficial. Similarly as with the regular waves, the appendices effects would be interesting to see.

Additionally for both the regular and irregular waves a different reference case should be done. A different hull with similar data available would provide additional trust and insight into the reliability of the simulations.

9 CONCLUSIONS

Calm water simulations for hull performance have been on the rise for the past decade, with them rapidly becoming the industry standard. Increase in accuracy and computational capacity have made it cheaper and faster, all the while the trust in the results have increased too. Further research efforts are then placed upon more complex simulations, with the hopes that better methodologies could provide improved insight into the hull’s performance in more complex situations.

In this thesis the feasibility of simulating vessel performance in waves had been studied, starting with simple static wave breaker simulated in regular waves. The simulation worked well, but without adequate comparison data from measurements comprehensive reporting on the final values was not possible. Comments from experienced designers and rough estimation values evaluated that the simulation results were atleast on the correct magnitude and visually correct.

In the second phase simulations were carried out for a ~170 meter ferry, first in regular waves with irregular waves in the final phases of the study. Several different types of simulations were compared in regular waves to the ones reported in model test report, with the self-propulsion type CFD simulation being very well aligned with those results.

In irregular waves two different sea states were simulated, with the smaller wave height of Adriatic sea and larger ones in S.North sea. In the smaller waves the wave added resistance was close to the one in model test report, but in the larger waves the difference was large. There could be several different factors contributing to the differences in thrust requirement as discussed in chapter 7.3.

Undoubtedly much more research is needed before simulations can replace model test in wave performance evaluation, but the result presented in this thesis give good indication that it should be possible at some point.

10 REFERENCES

[1] J. D. Fenton, "A Fifth-Order Stokes Theory for Steady Waves," in *Coastal and Ocean Eng.*, Port, J. Waterway, 1985, pp. 216-234.

[2] W. Thomson, "On ship waves," in *Proceedings*, Institution of Mechanical Engineers, 1887, pp. 641-649.

[3] G. P. T. R. J. W. R. J. Sobey R J, "Wave Theories," in *Waterway, Port, Coastal & Ocean Eng.*, 1987, pp. 565-587.

[4] I. R. M, "A Revised Parameterisation of the JONSWAP Spectrum," in *Applied Ocean Research*, 1987, pp. 47-50.

[5] F. O. M, Sea Loads on Ships and Offshore Structures, Cambridge University Press, 1990.

[6] "Hs-Tz scatterplots," GlobWave, 2013. [Online]. Available: http://globwave.ifremer.fr/products/demo-products/item/421-hs-tz-scatterplots. [Accessed 2021].

[7] Siemens Support Center, "What are the best settings for a VOF wave model simulation?," 2019.

[8] E. B. A. R. O. H. V. M. Houzeaux G, "A Chimera Method for the Incompressible Navier-Stokes Equations," *International Journal for Numerical Methods in Fluids,* pp. 1-45, 2013.

[9] V. H. K, Introduction to Computational Fluid Dynamics: The Finite Volume Method, Malalasekera: Weeratunge, 2007.

[10] v. K. Th, "Mechanical Similitude and Turbulence," *Tech. Mem. NACA,* no. 611, pp. 58-76, 1931.

[11] MARIN, "Seakeeping test for a 176m fast RO-RO ferry," 1997.

Additional files

Animations of simulations can be found in ship_animations.pptx power point slideshow, attached as a separate file