Center for Naval Architecture
Department of Mechanics

NUMERICAL MODELLING OF SPRAY SHEET DEFLECTION ON PLANING HULLS

Linus Olin

A thesis presented for the degree of
Master of Science

Royal Institute of Technology
Sweden

Date 2015-09-14

Summary
The intent of this paper is to investigate the seemingly untested theories regarding modelling of spray formation on planing hulls. This is done by means of 2D and 3D CFD simulations. The motivation for testing these theories is due to the design factors of a conceptual spray deflection system used on planing hulls. This technology is presented and its performance evaluated by comparison with current spray rail systems. With an appropriate design of the cross section of the deflectors the spray can be redirected aftward and thus recover some of the energy in the spray. It was found that the theory presented by Savitsky and Morabito, where the spray field of a planing hull can be represented by a 2D flat planing plate, will overestimate the spray thickness. However a close match between the spray formation on planing hulls and the theory presented by Wagner was found, this theory is that of a wedge being dropped vertically into a calm water surface. The reduced frictional resistance, when using spray deflectors, accounts for up to 28% reduction of the total drag and redirecting the spray aftward can reduce the total drag an additional 4%. The spray deflectors was found to reduce the wet surface area due to spray to a higher extent than possible with spray rails.

Sammanfattning
Avsikten med denna artikel är att testa de till synes otestade teorierna beträffande modellering av spray bildning hos planande skrov. Detta görs med hjälp av 2D och 3D CFD simuleringar. Motiveringen att testa dessa teorier är på grund av design faktorer av ett konceptuellt spray deflektions system som används på planande skrov. Denna teknologi presenteras och dess prestanda utvärderas genom att jämföra den med nuvarande spray skenor. Med en lämplig design av tvärsnittet av deflektorerna kan sprayen riktas bakåt och genom detta återvinna en del av energin i sprayen. Det visade sig att teorin presenterad av Savitsky och Morabito, där spray bildningen hos planande skrov representeras av en 2D planande platta, överskattar spray tjockleken. Däremot hittades en bra överenstämmelse mellan spray bildningen hos planande skrov och teorin presenterad av Wagner, denna teori bygger på en kil som släpps vertikalt ner i en vågrät vattenyta. Det minskade friktions motståndet, vid användning av spray deflektorer, står för up till 28% minskning av det totala motståndet och bakåtriktning av sprayen minskar det totala motståndet ytterligare 4%. Användning av spray deflektorer visade sig minska den våta ytan från spray bildningen till en högre grad än möjligt med spray skenor.

Preface

The work presented in this thesis has been carried out on the commission of Petestep AB at the KTH Center for Naval Architecture and Department of Mechanics. It has been performed under the supervision of Anders Rosén, Mireia Altimira and Jonas Danielsson. Thy are acknowledged for their help, support and rewarding discussions during this project.

This thesis has been written in the format of a scientific paper and is put forward to public review by Linus Olin.

Stockholm, September 2015
Linus Olin

Introduction

Planing crafts have hull shapes designed to generate lift from hydrodynamic forces. The main purpose is to overcome the wave making barrier experienced by displacing hulls at hight speeds. As a planing hull moves forward, water under the hull is decelerated and pressure is increased on the bottom of the hull. Planing is achieved when the hydrodynamic lift is greater than the hydrostatic lift. Planing reduces wave making resistance, but with increasing speed comes higher frictional resistance.

A significant part of the resistance of a v-bottomed planing hull comes from the region in front of the stagnation line, the spray area. The stagnation line is the line separating the flow going under the hull from the flow going into the spray area. A thin spray sheet is formed along the bottom of the hull increasing the wetted surface and wasting energy in spray to the sides. Established theories for performance prediction of planing hulls [1] were initially only concerned with resistance components generated behind the stagnation line. In 2007 this method was subject to update by the authors of the paper, where the resistance from the spray region was included [2]. It was concluded that the spray region could contribute up to 15% of the total drag, however experiments have shown values as high as 18% [3]. This poses a significant opportunity to reduce the total drag of a planing boat.

Traditionally, longitudinal spray rails are adopted to detach the spray from the bottom, thus reducing the frictional resistance from the spray. These do however leave large areas of the spray still attached to the hull and their horizontal part of the geometry increases the vertical accelerations experienced in a boat travelling in waves. These spray rails have been subject to many studies [2], [3],[4], but the general arrangement has remained unchanged. In this study a new arrangement of spray rails is studied.

The studied deflector arrangements are patented by Petestep AB, who have done extensive full scale testing of this technology. The idea behind it is that the spray could be intersected parallel to the stagnation line and subsequently deflected aftward. By deflecting the spray aftwards, a forward force is generated on the hull, thereby potentially reducing the power requirement of the boat. By intersecting the spray parallel to the stagnation line the wetted area could be reduced to a higher degree than what is possible with conventional longitudinal spray rails.

In order to design these deflectors, detailed knowledge about the spray region is required, especially the position of the stagnation line and the thickness of the spray sheet.

Theoretical equations for predicting the spray thickness have been presented by Wagner [5], Pierson, Leshover [6] and Payne [7], these are 2D-theories assumed to represent a planing hull. These are cases of either a flat planing plate or a wedge being dropped vertically into a free water surface. They are based on either momentum theory or linearized theory, i.e. by only taking into account the linear terms of the equations of motions. Savitsky and Morabito [8] extended these theories to prismatic planing hulls and formulated additional equations to describe the appearance of the spray area. However, there appears to be limited effort made to validate these theories, especially the relation between 2D-theoretical equations and the case of a prismatic planing hull. This is not surprising, as accurate experimental measurements are hard to achieve in the spray area.

In recent years, advances in computational performance have allowed for complex flow cases to be studied, such as free surface flows around planing hulls. This poses an opportunity to test the previously mentioned theories, since accurate measurements during controlled circumstances are possible. However this method require validation and verification due to different sources of error associated to the mathematical modelling and discretization. For this paper towing tank experiments [9], [10], [11] is used for validation of the model. Computational resources provides the level of approximation of the physical flow case, Computational Fluid Dynamics (CFD) analysis is always a compromise between computational resources and modelling level. The question whether either of these theories gives and adequate representation of the 3D-flow case remains unanswered and will be investigated.

The objectives of this study can be divided into three subcategories. First, setting up an appropriate 2D CFD simulation in order to evaluate the previously mentioned 2D spray theories and perform simulations related to the deflector design. Secondly develop a 3D CFD simulation based on the previous 2D-simulations in order to evaluate the relationship between the 2D- and 3D-simulations and also to model the spray details that are important for the spray deflectors. And lastly, applying the developed 3D CFD model in the evaluation of the performance of the spray deflectors.

Background theory

Traditionally it has not been in designers interests to recover energy in the spray of planing hulls. It has been assumed that the increase in wet surface area was the only negative contribution and that the loss of energy could not be avoided. However looking at figure 1, one can see that the drag components of a planing plate are divided into wave-making drag and spray-making drag. The ratio of these being a function of the Froude number, Fn. Since the wave-making drag is proportional to 1/Fn [12], it tends to zero at high speeds. This means all momentum transmitted to the water will appear as kinetic energy in the spray, assuming potential, and surface energy, i.e. the energy required to brake up the flow into droplets, is very low. Or to quote Payne "The resistance of a two-dimensional planing plate is zero (for inviscid flow) if the forward moving spray is directed aft by a deflector, thus satisfying d'Alambert's Paradox." [13]. The same analogy can be made with planing hulls.

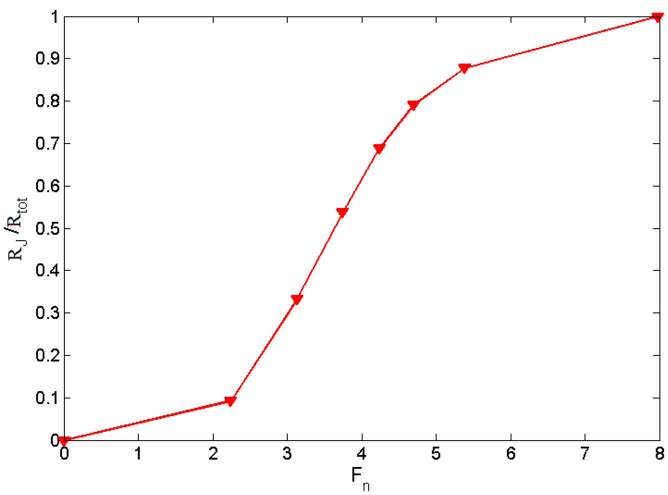


Figure 1: Ratio of spray making drag and wave making drag of a flat planing plate [12] for inviscid flow.

A brief description of pile-up and stagnation line on planing hulls is presented here to clarify concepts associated with the studied technology. As a planing hull moves forward the wetted length is slightly increased than what is defined by the intersection between the hull and the undisturbed water surface. The pile-up factor, f, is defined by

$$\frac{lw}{lu} = f \tag{1}$$

which is the ratio between the wet length due to the pile-up of water, lw, and the initial wet length, lu, see figure 2. In this paper the wetted length is defined as the length over which high hydrodynamic pressure is exerted, i.e. not including the spray area.

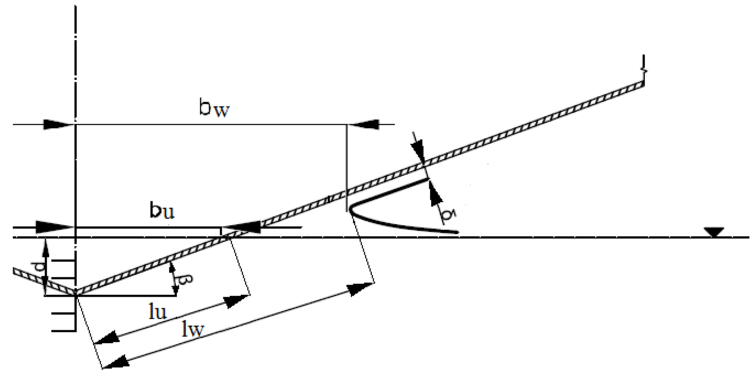


Figure 2: Dimensions of a cross section of a prismatic hull.

As mentioned earlier, the stagnation line is the high pressure zone separating the spray area from the flow going under the hull. A conceptual description is shown in figure 3. Considering a flat planing plate, the stagnation line is the line where the flow divides into a forward and aft component. It should be noted that for a deadrised hull this flow also has an sidewise component. The angle the stagnation line makes with the hull is determined by the running condition, trim and the deadrise angle of the hull. First, considering the calm water intersection with the hull at this running condition, the water surface forms a distinct v-angle with the hull. Subsequently taking into account the pile-up of water close to the stagnation line, known as the pile-up factor $\pi/2$ [1], the angle of the stagnation line can be described by

$$\alpha = \arctan\left(\frac{\pi \tan\tau}{2\tan\beta}\right). \tag{2}$$

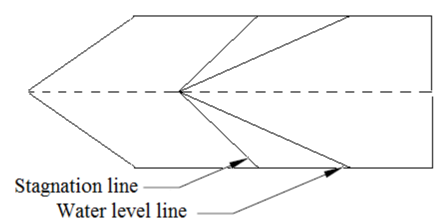


Figure 3: Water line intersection and stagnation line for prismatic hulls.

The idea behind the type of deflectors developed by Petestep AB is very simple and consists of two aspects. First reducing the wetted surface in order to reduce the frictional drag and secondly redirecting the spray in order to create a forward thrust force acting on the bottom of the hull. Thus, the conceptual idea of the spray deflectors is to cut off the spray parallel to the stagnation line, potentially eliminating the wetted area due to spray. The idea is illustrated in Figure 4.

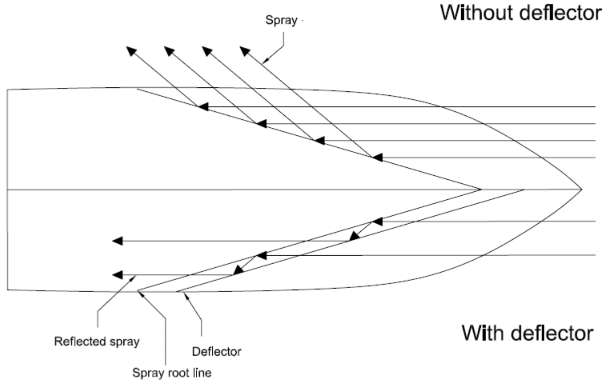


Figure 4: Conceptual redirection of flow.

Designers of high speed crafts are usually faced with the trade off between performance and comfort, i.e. resistance and vertical accelerations, the common variable being the deadrise angle, β. A high deadrise angle gives low accelerations and high drag, while a low deadrise angle gives high accelerations and low drag. Accelerations can also be increased by adopting traditional spray rails, due to the horizontal part of the geometry. This effect is not present with the spray deflectors, which are characterized by a vertical geometry.

(a) (b)

Figure 5: Difference in cross section of (a) deflector and (b) spray rail

The wetted surface of the hull can be divided into two regions, the pressure area, aft of the stagnation line and the spray area forward of the stagnation line. Until 2007 established theories for resistance prediction of planing hulls assumed the drag was only formed from behind the leading edge of the stagnation line. These resistance components were frictional and pressure drag. There appears to be some disagreement how these resistance components are calculated, especially the frictional drag from the spray area. In this section the different theories are presented with the intention to introduce and clarify, to the reader, how the resistance is calculated.

According to Savitsky [1] the added wet area due to spray can be described by (3). It is clear that at zero deadrise angle this approaches infinity, which means that no regard has been taken to the fact that at some point the spray will separate from the hull due to gravitational forces.

$$\Delta\lambda = \frac{\cos\Phi}{4\sin 2\alpha\cos\beta} \tag{3}$$

This is the effective wet length to beam ratio used to calculate the frictional drag, this is a non-dimensional measure, as opposed to the actual wet surface area. Effective wet length to beam ratio is different to actual length to beam ratio, since regard has been taken to the direction of the spray, i.e. for a spray velocity completely to the sides $\Delta\lambda$ will be zero, since the fictional drag is zero. In the first publication of Savitskys performance prediction method [1], the effect of the spray related frictional drag was mentioned but never included. However, other authors have included the frictional drag from the spray area to the Savitsky method, e.g. in 2000 Larsson and Eliasson [14] included the wetted area of the spray to the frictional drag according to:

$$R_f = \frac{1}{2}\rho V^2(\lambda + \Delta\lambda)\frac{b^2}{\cos\beta}C_f. \tag{4}$$

In this model, the spray area and the area behind the stagnation line is assumed to have the same velocity, equal to the speed of the boat. This is however a crude approximation, since the high pressure in the area behind the stagnation line will reduce the velocity, the frictional drag from this area should rather be computed with the average velocity estimated by Savitsky.

In [2] frictional drag from the spray area was included to the Savitsky model by simply taking into account the velocity and direction of the spray and the wetted areas due to spray, according to:

$$R_s = \frac{1}{2}\rho V^2\Delta\lambda b^2 C_f. \tag{5}$$

This is a fair assumption since it divides the frictional drag from the spray area and the pressure area onto two separate components, each with its own characteristic velocity. For full-scale applications it is usually assumed that the flow is fully turbulent, however considering the fact that the spray sheet can be less than the height of the boundary layer, one can question the validity of using general models for calculating the frictional coefficient in this formula.

Interesting behaviour of the spray drag is found for certain combinations of trim and deadrise, giving a negative spray drag area, shown in figure 6. This is due to the fact that the spray will move in the direction of the boat direction of travel, resulting in a negative frictional drag, it would thus be undesired to reduce such an effect.This is however only present at low deadrise angles and redirecting the spray with deflectors will still generate forward thrust.

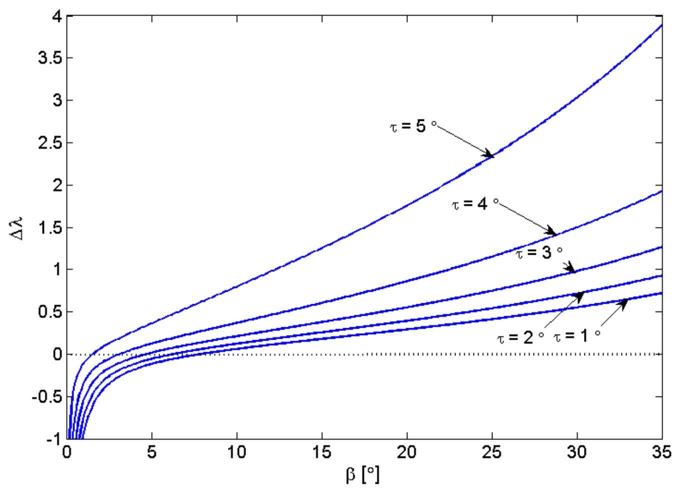


Figure 6: Effective added area due to spray according to (3).

Since a planing hull has a trim angle towards the free water surface, the hydrodynamic lift force will have an aft component, this is refereed to as pressure drag. Established theories model the pressure drag as:

$$R_d = \Delta \tan \tau. \tag{6}$$

Thus independent of the pressure distribution, and thus the pressure drag from the spray area is included, however small it may be. This also means pressure drag can not be reduced without changing parameters of the boat, i.e. trim and weight.

It is assumed that the velocity in the spray area is equal to that of the free stream velocity. According to the Bernoulli equation

$$\frac{1}{2}\rho V^2 + \rho g z + p_{atm} = constant \tag{7}$$

the pressure in the spray area has to be close to atmospheric pressure, disregarding any change in hydrostatic pressure since the height scales are generally very low, in the order of magnitude of the waves, compared to the velocity scale, in the order of magnitude of the speed of the boat.

The exact prediction of the spray details has until now not had any practical motivation with regard to the design of spray rails. However, detailed knowledge of the spray sheet thickness, pile-up, and position of the stagnation line will enable more efficient design of the here presented deflectors.

Many attempts have been made to come up with mathematical models describing the complex flow around planing hulls. One theory suggests using a flat planing plate as a representation of the 3D flow case, e.g. Pierson and Leshover [6]. They came up with a theory, where the draft and trim of a flat plate being towed in a towing tank would uniquely describe the spray thickness. Another theory suggests that dropping a wedge into a calm water surface is a more appropriate representation of the spray formation. In this theory, e.g. Wagner [5], the wedge would be a representation of a cross section of the hull. The idea is that the wedge would strike the water surface and thus generate some spray to the sides. The wedge would penetrate the water surface until it has a draft equal to the draft of the boat it is supposed to represent. Savitsky [8] presented a simplified version of the theory presented by Pierson and Leshover, but no motivation why this would be preferred over the theory presented by Wagner. Since experimental measurements of the spray thickness are difficult, only mathematical models basted on linearized theory and momentum theory are available, both for 2D-plates and wedges. It is suggested in [8] that the characteristics of the spray can be described by a 2D-cross section perpendicular to the stagnation line see figure 7, and following the equation from [6], the spray thickness can be described in therms of fraction of the wetted length. Thus in order to find the local spray thickness the local wet length must be known, and thus the pile up of water. The question whether to use the plate or the wedge theory remains, and will here be investigated, by means of CFD-modelling.

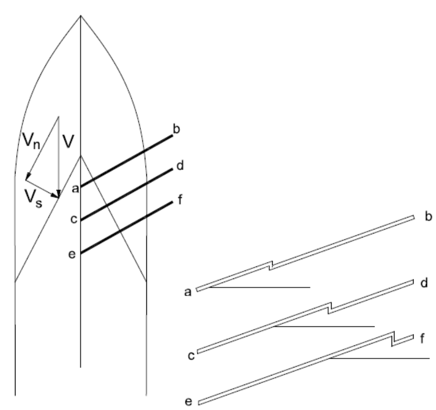
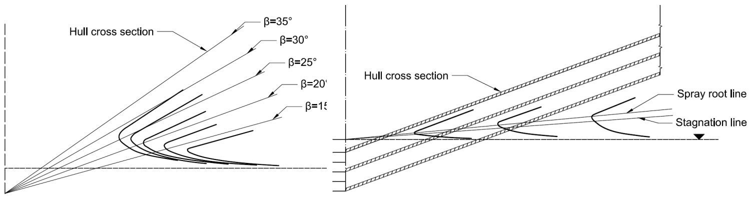


Figure 7: Cross sections used for 2D-flow.

A commonly used assumption when modelling the pile up in front of a flat planing plate, is that the only independent variable is the wetted length. In practise this implies that the spray thickness and pile up will be the same for any case with the same wetted length, regardless of whether this is due to a low angle and low draft or high angle and high draft both giving the same wetted length. As an initial assumption this somewhat hurts the generality of the model. Instead both the angle and draft of the plate will be used as separate variables. This assumption will be used in the 2D-flow case investigation, hence either the angle or the draft will be changed in each different flow case, according to figure 8.



(a) Cross sections with different deadrise angles. (b) Cross sections with different drafts.

Figure 8: Cross sections used in 2D-flow case

Mathematical model

In order to understand and to put the subsequent sections into context, a brief introduction CFD modelling is presented.

The governing equations for the conservation of mass and momentum in the flow field are the Navier-Stokes equations, N.S. These arise from applying Newtons second law of motion to a fluid. They consist of the continuity equation (8) and x,y and z-momentum equations (9).

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \overrightarrow{u}) = 0 \tag{8}$$

$$\rho(\frac{\partial \overrightarrow{u}}{\partial t} + \overrightarrow{u} \cdot \nabla \overrightarrow{u} = -\nabla p + \nabla \cdot (\mu(\nabla \overrightarrow{u} + (\nabla \overrightarrow{u})^T) - \frac{2}{3}\mu(\nabla \cdot \overrightarrow{u})\overrightarrow{I}) + \overrightarrow{F} \tag{9}$$

Due to the complexity of the equations they are very hard to solve analytically, except for some simple laminar cases. However, most engineering applications involving fluid flows operate in the turbulent regime. This regime is characterised by chaotic and random fluctuations of the flow fields. Usually the time average of the flow field quantities are sufficient for an engineering application. The equations representing this flow field is commonly referred to as the RANS-equations. The effect of turbulence is now represented by turbulence models.

In this work, the k-ω SST turbulence model has been adopted. This model resolves two equations for the turbulent kinetic energy, k and the specific dissipation rate ω to model the turbulent fluctuations. It was found to give a better representation of the separation of the spray sheet, compared to a k-ε model. The addition of the shear stress transportation equation to the k-ω model provides the best opportunity for a RANS-model to predict the separation region in areas where other turbulence models might fail [15]. However the lack of wall-functions requires a high resolution of the mesh to resolve the boundary layer.

An additional challenge of the simulation of spray formation in planing hulls is the need to define the interface between two fluids, air and water. For that purpose, the Volume of Fluid method, VOF, is used [16]. This method solves a volume fraction continuity equation for the volume fraction of water within the domain. Each cell can be filled with either air, volume fraction of 0, water, volume fraction of 1, or a combination of fluids, with a volume fraction between 1 and 0. The VOF model is shown to give a more physically correct representation of the thin spray sheet and separation on a flat planing plate compared to an Smooth Particle Hydrodynamics (SPH) approach [17].

When dealing with any type of spray modelling, surface tension forces should not be neglected if the Weber number (We) is less than 1. The Weber number is a dimensionless parameter of the ratio between inertial forces and surface tension forces. In this case it is in the order of 10 000, when using the spray sheet thickness and free stream velocity as characteristic length and velocity. A commonly used model, the continuum surface tension model (CFS) [18] will be used in this study. This numerical treatment of the surface tension forces is closely related to the transition region between two phases. The surface tension force is acting on the cells containing the interface, the CSF model transforms this surface force into a body force. Hence a sharp interface is required for the accurate modelling of surface tension forces.

2D-flow case

As described in the introduction the representation of the flow field of a 3D planing hull by a planing plate will be tested, as this seems to be the preferred model for modelling the spray area, as opposed to the wedge theory. The intention with the 2D-flow case is to examine the theoretical equations for the spray thickness as well as the pile-up. But also to come up with an appropriate computational set-up, i.e. solution schemes, turbulence model, and boundary conditions, which will be used in the 3D-flow case. This is achieved by systematically changing either the draft or angle of the plate.

The set-up of the 2D CFD model is depicted in figure 9. At the inlet boundary the free water surface level and the uniform velocity of 4 m/s is prescribed. At the top and outlet boundaries a constant static pressure is prescribed with zero normal gradients for the other flow fields. The bottom boundary has a slip condition. On the plate itself a non-slip condition is used.

The computational domain is 8x5 m and the plate is 2 m located in the middle of the domain. The water lever ranges from 2,78 m to 3 m depending on which flow case is studied, since this range was found to be sufficient to avoid any shallow water effects.

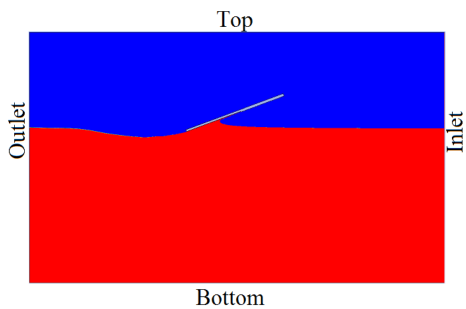


Figure 9: Computational domain for the 2D-flow case.

Additional properties of the computational set up regarding the discretization of the governing equations is shown in table 3.

Table 1: Computational set-up

Discretization of equations	
Momentum	Second order upwind
Volume fraction	Modified HRIC
Turbulent kinetic energy	Second order upwind
Specific dissipation rate	First order upwind
Transient formulation	First order implicit
Gradient	Least square cell based
Pressure	PRESTO
Pressure-velocity coupling	SIMPLE

The same models, boundary conditions and solutions schemes used in this 2D-case will be used in the 3D-case.

The mesh for the 2D-flow case is shown in figure 10. The mesh used is a quadrilateral mesh structured in the area of interest in front of the plate. The mesh size varies with the angle of the plate, but is in general around $130*10^3$ elements with the finest resolution closest to the plate. This refinement is varied depending on the spray thickness, and ranges from 10-17 elements through the spray sheet thickness, this is equivalent to a cell size on the plate between 0,01 mm to 10 mm. This gives a sufficient resolution of the viscous sub-layer with a $y^+ < 1$. The simulations are run with a time-step of 0,002 s and 5 iterations/time-step, iteration error was estimated to be less than 0,1%. An example of the scaled residuals of a simulations is shown in table 2.

Table 2: Residuals in 2D-simulations

Equation	Scaled residual
Continuity	$9*10^{-5}$
X-velocity	$9*10^{-6}$
Y-velocity	$8*10^{-6}$
K	$3*10^{-4}$
ω	$2*10^{-4}$
Volume fraction	$1*10^{-5}$

Mesh convergence is shown in table 3, convergence with a difference of less then 1 % is found with a mesh of $133*10^3$ cells.

Table 3: Convergence of 2D-mesh

Mesh	Cells(10^3)	Cell size on plate[mm]	δ[mm]	l_w[m]
Coarse	45	10	15.12	1.121
Medium	90	5	15.36	1.119
Fine	133	3	15.47	1.125
Finest	164	1	15.48	1.127

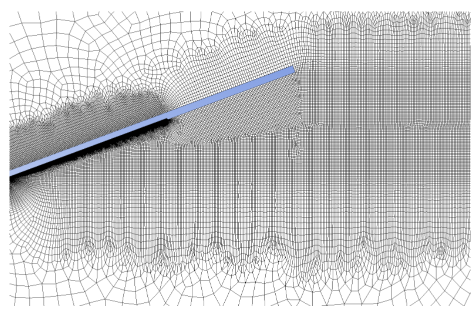


Figure 10: Example of 2D-mesh.

Since an accurate resolution of the spray area is important for the purpose of this study, the interface between the air and water has to be accurately described by a sharp interface. This also gives a better representation of surface tension forces. This is achieved by means of mesh resolution and by choosing a proper advection scheme for the water volume fraction that has minimum numerical dissipation. The effect of refining the mesh in the spray area is shown in figure 11, where a close up of the interface between air and water in the spray area of the plate is depicted. In reality the interface is a discontinuity between air and water, however this is numerically represented by a transition between 0-1 volume fraction.

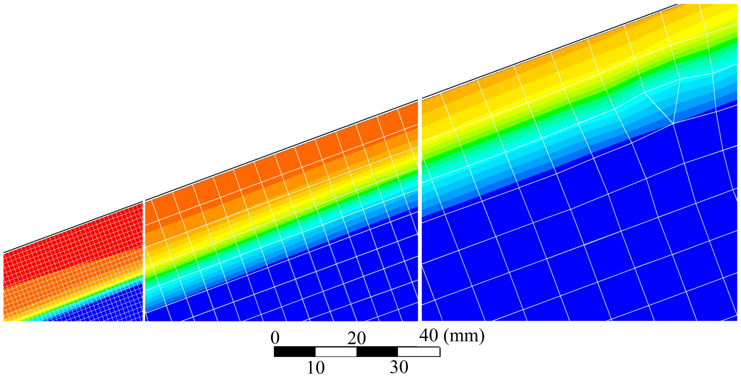
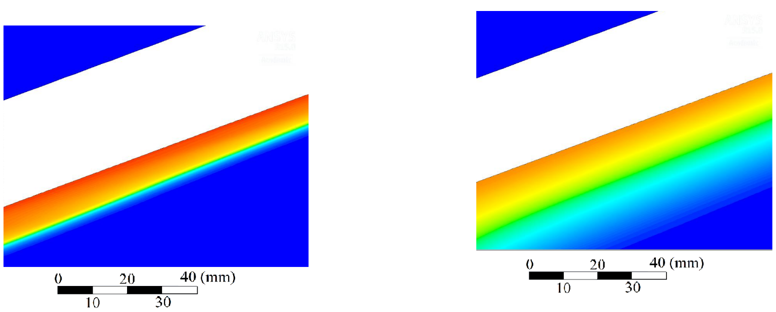


Figure 11: Resolution of the interface between air (blue) and water (red), with a mesh size from 1 mm (left) to 10 mm (right).

The difference between a second order upwind scheme and a modified HRIC scheme, which is a combination of both upwind and downwind schemes [19], is shown in figure 12. Obviously the modified HRIC scheme gives a sharper interface, due to its reduced numerical diffusion. This scheme is used in all subsequent simulations.



(a) Interface with modified-HRIC scheme.

(b) Interface with 2:nd order scheme.

Figure 12: Numerical diffusion of the interface between air (blue) and water (red) for different volume fraction advection schemes.

In total 35 different flow cases are studied with angles ranging from 5 to 35 degrees and drafts ranging from 0.016 to 0.216 m at a speed of 4 m/s.

3D-flow case

In this section a 3D CFD model is analysed, with the intention to investigate the validity of using the previous 2D-flow case as a representation of 3D-flow case, as described in the introduction. The hull is modelled as a prismatic wedge hull with constant deadrise angle and a straight keel, see figure 13.

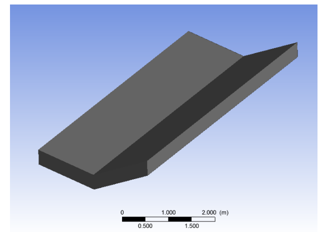


Figure 13: Geometry of the prismatic planing hull.

It was found that a structured hexahedral mesh poses the best opportunity for details of the flow to be captured, such as the spray area.

A common problem when modelling planing hulls using the VOF-model is referred to as numerical ventilation. This is the non-physical supply of air being pulled in under the hull and has been observed in previous studies, e.g. [20], [21]. The source of this being in the way the solver treats elements with a volume fraction between 1-0 i.e. elements with both air and water. These are present in the intersection with the hull and the water surface level. In the forward most spray area, the spray thickness approaches zero and at some point along the hull, the local element size will be in the same order of magnitude as the physical spray thickness. However in this area the refinement is insufficient to resolve the spray sheet, hence no spray sheet will form forward of the stagnation line. Instead the information in these elements will be supplied under the hull as seen in figure 14. This is the source of the numerical ventilation i.e. elements with both air and water, but with only one velocity vector.

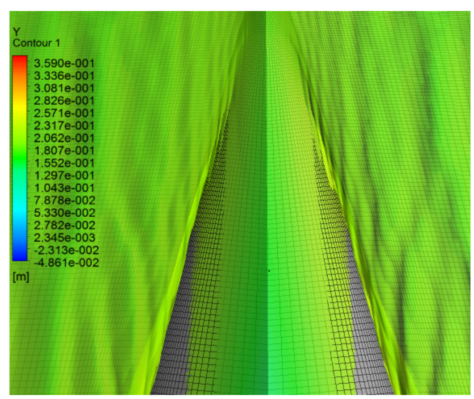


Figure 14: Air film underneath the hull.

This effect can compromise the shear stress on the bottom of the hull if the air film is thick enough compared with the local mesh size. The shear stress is calculated from the the velocity and viscosity of the elements closest to the hull. The viscosity is calculated with a weighted average of the air and water fraction in the element. If the amount of air is very low compared with the amount of water, the viscosity will be close to that of the water.

This effect can be reduced with a refined mesh, but it can not be completely suppressed. It was found that a refined mesh close to the hull is not sufficient to reduce this effect, but rather a refinement upstream of the hull along the water surface is required.

In total six different hulls are simulated, with deadrise angles ranging from 10 to 30 degrees, see Appendix A. Cross sections of the 3D-simulations according to figure 15 are compared with the 2D-simulations and the theoretical equations.

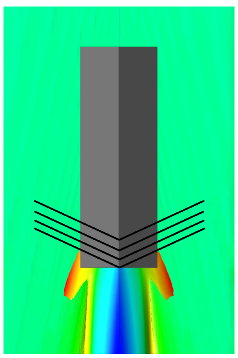


Figure 15: Cross sections used for comparison with 2D-flow case.

Mesh convergence is considered with respect to the variables of interests; pile-up and spray thickness, the mesh is refined in the spray area. The spray sheet thickness and pile up is measured in the cross section planes according to figure 15. Mesh convergence is shown in table 4, differences of less than 1 % are found with a mesh size of $3.54*10^6$ cells. The same number of elements through the spray sheet thickness used in the 2D-simulations are used in the 3D-simulations, however the element face size on the surface of the hull is varied according to table 4. These are slightly larger compared to the 2D-simulations in an effort to reduce mesh size.

Table 4: Convergence of 3D-mesh

Mesh	Cells (10^6)	Cell size on hull [mm]	δ[mm]	l_w [m]
Coarse	1.5	50	-	0.86
Medium	2.31	20	9.6	0.88
Fine	3.54	10	15.5	0.95
Finest	4.35	5	15.57	0.95

Comparison between 2D and 3D models

In this section results from the 2D- and 3D-simulations are compared with the theories presented in the Background section. In figure 16 the simulated spray thickness is compared with theoretical equations in terms of fractions of the wetted length, at each angle multiple drafts are tested. Good agreement between Pierson and Leshover [6] and Savitsky and Morabito [8], is found for the 2D flow case. This is a theoretical equation of a flat planing plate according to the set-up in the 2D-flow case.

As opposed to the initial hypothesis the 2D-plate theory does not appear to be a good representation of the 3D-flow case. Instead a closer match is found with the wedge theory by Wagner [5], which was briefly described in the background theory part of this paper, see Appendix C for comparison.

These results challenges the seemingly untested theory that the spray area can be modelled according to a flat planing plate, but suggests that a wedge drop theory is a closer match. The relative differences between these two theories are quite significant. Therefore, using the plate theory to design these deflectors, will result in an overestimation of the spray thickness as well as the height of the deflectors.

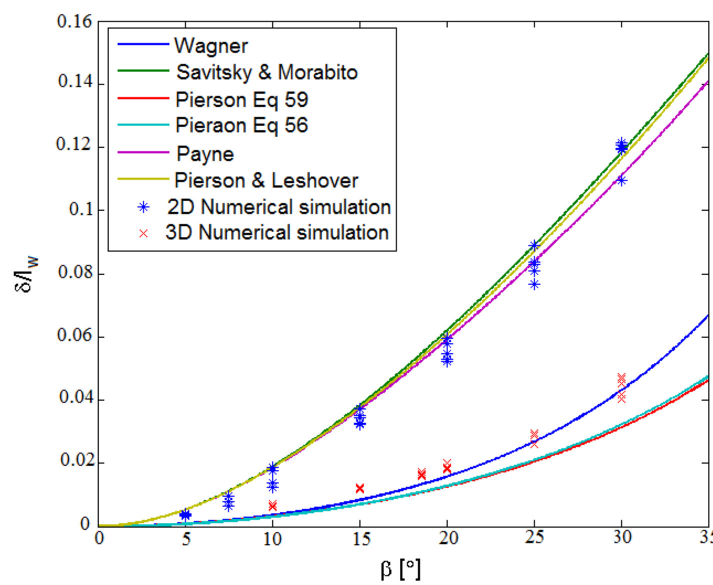


Figure 16: Spray thickness fraction of wetted length, simulations comparison with theory.

As for the pile-up factor, the most commonly used theory suggests that this factor is constant and equal to $\pi/2$ [1]. Again, there appears to be little practical motivation for the exact value of this factor. In performance prediction and design of spray rails, only the order of magnitude is in general of interest. However as for the spray thickness, the pile-up factor has an important role in the design of the deflectors.

Numerous experiments with flat planing plates have been made in an attempt to investigate pile-up, e.g. [9], [10], [11]. In general, the increase in wet length due to pile-up was found to be between 30-40 % of the beam of the plate. A summary of theories and experiments are shown in figure 17.

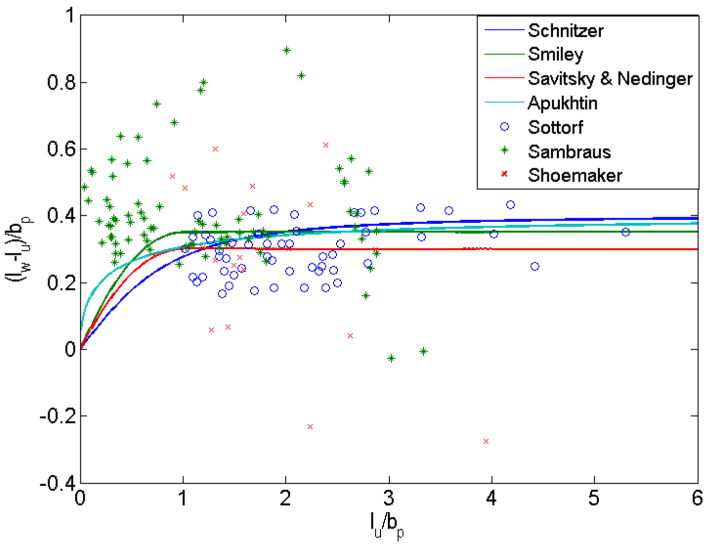


Figure 17: Summary of theory on pile-up on flat planing plate [22].

In figure 17 the pile-up is defined as the difference between the wet length due to the pile-up of water and the initial wet length, normalized with the beam of the plate, as seen on the vertical axis. Evidently, the data is very scattered and it is hard to justify any conclusions based on these results, especially considering some of the data points indicate a negative pile-up. Also the representation of the spray area of a prismatic planing hull by a 2D-flat planing plate appears to be invalid.

Again, the 2D-plate simulations appears to be a bad representation of the 3D-flow case, see table 5. In addition, the simulations does not suggest that the pile-up factor is independent of the angle. This was also noted by Payne [22]. In fact, the $\pi/2$ factor appears to be valid for only a small range of angles between 15 and 20 deg, according to this study. For higher and lower angles this factor is slightly higher, according to the 3D simulations. The trim angle was kept constant at 3.8°for all 3D-simulations, this does however have a very small effect on the angle of the cross section, which is very close to the deadrise angle.

Table 5: Pile-up factor from 3D- and 2D-flow case

Deadrise	Pile-up factor 3D	Pile-up factor 2D
5°	-	1.540
7.5°	-	1.693
10°	1.837	1.772
15°	1.658	1.862
18.6°	1.646	-
20°	1.668	1.913
25°	1.826	1.947
30°	1.953	1.944

Since there is a difference between the pile-up factor of 2D-plate simulations and that of 3D simulations, the stagnation line angle will also differ. The effect this difference has is presented in table 6. by using (2). In general the 3D-flow case suggests a slightly larger angle of the stagnation line. Which might result in the stagnation line crossing the deflector, compromising the performance.

Table 6: Angle of stagnation line with pile-up factor from 3D-flow case and $\pi/2$ factor

	Stagnation line angle with pile-up factor:		
Angle	$\pi/2$	From 3D case	difference
10°	31.9°	36.1°	4.15°(13 %)
15°	22.3°	23.4°	1.1°(4.9 %)
18.5°	18.1°	18.8°	0.7°(4.3 %)
20°	16.8°	17.8°	1°(5.8 %)
25°	13.3°	15.3°	2°(15.6 %)
30°	10.8°	13.3°	2.53°(23.6 %)

Application

The CFD set up in the previous sections will here be applied in investigating the mechanics and performance of the spray deflectors. The performance of the deflectors is evaluated in terms of reduction of viscous resistance and spray deflection related forward thrust in comparison to a bare hull.

Simulations with the use of deflectors are based on a boat with particulars according to table 7. The running condition is based on the equilibrium trim and draft found by using the equations provided by Savitsky [1], with the addition to forces associated with deflecting the spray. With the intention of isolating the effect the deflectors have on the resistance, the same running condition is used in the case without deflectors, the bare hull.

Table 7: Particulars of boat used in example

Length	Breath	β	draft	Velocity	τ	Δ
7 m	3.2 m	18.6°	0.2 m	40 knots	3.8°	1350 kg

Three different arrangements of deflectors are tested according to table 8 with deflector height h and cross section angle γ defined in figure 18.

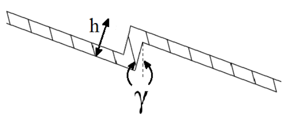
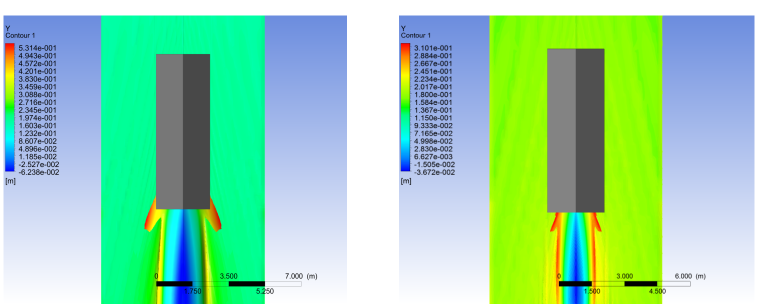


Figure 18: Dimensions of deflector.

Table 8: Dimensions of deflectors and their performance

Case	h	γ	Fwd thrust/total drag
1	22 mm	5°	4.9%
2	30 mm	5°	5.8%
3	22 mm	-5°	6.0%

Table 9 shows the potential deflectors can have on a bare hull. As seen the wet surface area due to spray can be more than 40% of the total wet surface area and the mass flow of water in the spray sheet can be in the order of 200 kg/s. In addition, one should not underestimate the visual appeal of a boat travelling through water with visually less disturbance to the water surface, see figure 19.



(a) Top view without deflector.

(b) Top view with deflector. Case 2.

Figure 19: Comparison of spray formation with and without deflector

Table 9: Kinetic energy in spray sheet and wet surface area due to spray in relation to total wet surface area

Deadrise	A_s/A_{tot}	Mass-flow spray [kg/s]	Energy spray [kW]
10°	23.3 %	215	10.8
15°	34 %	226	13.1
18.6°	42 %	238	14.4
20°	43 %	231	15.8
25°	36 %	229	13.8
30°	35 %	210	12.7

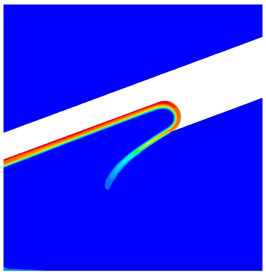
By using the bare hull as a reference, the reduced resistance when using deflectors can be studied and compared with model test with and without spray rails. Previous studies of spray rails concluded that a reduction of the total resistance by 18 % was possible [3]. In this study a reduction of the total resistance up to 32% was found, when using deflectors, as seen in table 10. The deflection of spray can account for 4 % of this improvement. This component is found by integrating the pressure over the deflectors and projecting the force in the forward direction, the rest is due to reduced frictional resistance. The deflectors recover about 20 % of the energy in the spray sheet into forward thrust. Considering the theoretical efficiency can not be grater than 60 %, see Appendix B, this appears to be a promising result. The highest efficiency of 60 % is however only possible if the spray has a direction completely forward with a speed ratio of the spray and deflectors of 1/3, and is redirected 180 deg. It is however unlikely that this is possible in practice.

Although the geometry of the deflectors will act to increase the wet surface area, this effects appears to be negligible. This is due to the fact that the aft velocity component on the surface of the deflector is very small and is mostly accounted for by a downward velocity component.

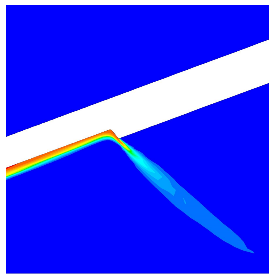
Table 10: Resistance comparison.

Case	Reduction due to reduced wet surface area	Reduction due to deflection of spray	Total
Spray rail	18.0%	-	18.0 %
Case 1	28.0%	3.3%	31.3 %
Case 2	28.0%	3.9%	31.9 %
Case 3	28.0%	4.1%	32.1 %

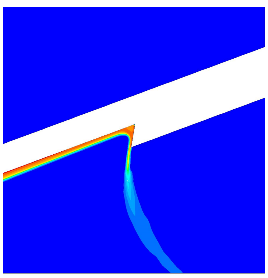
The forward thrust by deflecting the spray sheet is uniquely described by the change in momentum of the spray i.e. change in direction and velocity. Due to the reduced computational time of the 2D-simulations, numerous cross sections of the deflectors could be investigated and performance judged by deflection angle. Both height and cross section angle of the deflectors are tested, also a semicircle cross section of the deflectors is investigated. A few examples are shown in figure 20.



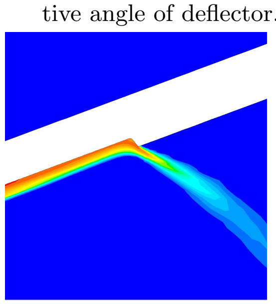
(a) Curved deflector.



(b) Deflector height 150% of spray sheet thickness, positive angle of deflector.



(c) Negative angle of deflector.



(d) Deflector height equal to the spray sheet thickness, positive angle of deflector.

Figure 20: Effect of different cross section of deflectors, water is represented by red and air by blue.

It was found that a semicircle deflector gives the highest deflection angle of close to 180 deg deflection of the spray sheet. For a straight angled deflector the direction of the deflected spray will be equal to the angle of the deflector, if the height of the deflector is in the order of 150 % of the spray sheet thickness. The effective height of the deflector is somewhat reduced by a small recirculation region being generated in the corner of the deflector, this effect was also noticed in the 3D flow cases.

Conclusions

The objectives of this study was: to evaluate the previously mentioned 2D spray theories; to examine the relationship between the 2D- and 3D-simulations; to perform simulations related to the design of deflectors; and to study the performance of the spray deflectors. Based on the simulations presented in this study the following conclusions can be drawn:

1. The representation of a prismatic planing hull by a flat planing plate will overestimate the spray thickness.
2. The representation of a prismatic planing hull by a vertical wedge drop will give a good prediction of the spray thickness.
3. According to these simulations the $\pi/2$ pile-up factor only good for angles around 15 degrees
4. Deflectors have the potential to further reduce the wet surface area compared to conventional spray rails at the design speed of the deflectors.
5. Using deflectors can reduce the total drag up to 32 % of the total drag.
6. Deflection of the spray can generate a forward thrust responsible for up to 4 % reduction of the total drag
7. Numerical ventilation is only a problem if the air film thickness is in the same order of magnitude as the local mesh size. To reduce this effect refinement of the mesh has to be done close to the hull as well as upstream of the hull along the water surface.
8. The deflectors should be in the order of 1.5 times the local spray thickness, to achieve a deflection of the spray equal to the angle of the deflector.

Discussion and future work

The work presented in this paper is limited to input for the design of the deflectors and validation of hypothesis related to the characteristics of the spray area. However a lot of work can still be done as to identify the effect of the deflectors. Especially vertical accelerations levels when travelling through waves. Full scale testing have indicated lower noise levels and lower accelerations in a boat fitted with deflectors compared to a boat with spray rails. The air film ventilation of hull could be interesting from a modelling point of view, but also for a performance application. A 2-degree of freedom simulation to investigate effect on trim and heave due to deflection of spray, would be a more fair simulation, in this paper the running condition was calculated with a modified Savitsky method. The resistance characteristics of submerged deflectors are important to investigate in cases where multiple deflectors are used being effective at different speed intervals. By parameterising the geometry of the deflectors and using total resistance as objective function, an optimization extension to the CFD solver could possibly provide the optimum arrangement of the deflectors.

List of symbols

A	Cross section area of spray sheet
A_s	Wet surface area due to spray
A_{tot}	Total wet surface area
b	beam of hull
b_u	Wetted beam at undisturbed water line
b_w	Wetted beam due to pile up of water
C_f	Frictional coefficient
d	Draft
f	Pile-up factor
F_x	Forward thrust on deflectors
h	Height of deflector
L	Length of hull
l_u	Wetted length at undisturbed water line
l_w	Wetted length up to spray root line
$\dot{m}$	Mass flow of spray
P_{in}	Energy in spray sheet
P_{out}	Recovered energy from spray sheet
R_d	Total pressure drag
R_f	Total frictional resistance
R_j	Drag due to spray
R_s	Frictional resistance due to spray
R_{tot}	Total drag
u	Velocity of spray sheet
V	Velocity of hull
V_n	Velocity component normal to stagnation line
V_s	Velocity component parallel to stagnation line
W_e	Weber number
α	Angle between stagnation line and keel
β	Deadrise angle
γ	Cross section angle of deflector
Δ	Displacement of hull
δ	Thickness of spray sheet
$\Delta\lambda$	Wet surface area due to spray/beam
η	Efficiency in recovery of energy
λ	Mean wet surface area/beam
ρ= 996 kg/m^3	Water density
τ	trim angle of hull
Φ	Angle between keel and spray edge

References

[1] D. Savitsky, *Hydrodynamic Design of Planing Hulls*, Marine Technology 1, 1964

[2] D. Savitsky, M.F. DeLorme, R. Datla, *Inclusion of Whisker Spray in Performance Prediction*, Marine Technology 44, 2007

[3] E. P. Clement, *Reduction of Planing Boat Resistance by Deflection of the Wisker Spray*, AD0454407, 1964

[4] T.C. Fu, T. Ratcliffe, T.T. O'Shea, K.A. Brucker, R.S. Graham, D.C. Wyatt, D.G. Dommermuth, *A Comparison of Experimental Measurements and Computational Predictions of a Deep-V Planing Hull*, 28th Symposium on Naval Hydrodynamics, California, 2010

[5] H. Wagner, *Über Stoss und Gleitvorgänge an der Oberfläche von Flüssigkeiten*, Zeitschrift für Angewandte Mathematik und Mechanik, Vol.12, 1932

[6] J.D. Pierson, S. Leshnover, *An Analysis of the Fluid Flow in the Spray Root and Wake regions of Flat Planing Surfaces*, 1948

[7] P. R. Payne, *On the Spray Sheet Thickness of a Planing Wedge Hull*, Ocean Engineering 13 p. 1-36, 1986

[8] D. Savitsky, M. Morabito, *Origin and Characteristics of the Spray Patterns*, Davidson Laboratory, Stevens Institute of Technology, Hoboken, Report No 2882, 2010

[9] A. Sambraus,*Planing-Surface Tests at Large Froude Numbers-Airfoil Compraison*, 1936

[10] J.M. Shoemaker,*Tank Tests of Flat and V-Bottom Planing Surfaces*, National Advisory Committee for Aeronautics, Langley Aeronautical Lab, 1934

[11] W. Stofford,*Experiments with Planing Surfaces*, National Advisory Committee for Aeronautics, 1932

[12] H. B. Squire, *The motion of a simple wedge along the water surface*, Proceedings of the Royal Society of London, Series A, Mathematical and Physical Sciences 243, p.48-64, 1957

[13] P. R. Payne, *The Spray Volume Shed by an Uncambered Planing Hull in Steady Planing*, Ocean Engineering 9 p. 373-384, 1982

[14] L. Larsson R. Eliasson, *Principles of Yacht Design*, Adlard Coles Nautical, 2000

[15] R.F. Menter, *Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows*, Mashgiz Press,AIAA Paper 93-2906, 1993

[16] C. W. Hirt, B. D. Nichols, *Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries*,Journal of Computational Physics 39, p.201-225, 1981

[17] P. Ghadimi, A. Dashtimanesh, M. Farsi, S. Najafi, *Investigation of free surface flow generated by a planing flat plate using smoothed particle hydrodynamics method and FLOW3D simulations*, Journal of Engineering for the Maritime Environment, 2012

[18] J.U. Brackbill, D. B. Kothe, C. Zemach, *A Continuum Method for Modeling Surface Tension*, Jurnal of Computational Physics p.335-354, 1992

[19] V. Dostal, V. Železný, P. Zacha, *Simulations of Two-Phase Flow in Fluent*, ANSYS FEM Users' Meeting & 14. ANSYS CFD Users' Meeting Luhačovice 5. - 7. listopadu, 2008

[20] D. Villa, S. Gaggero, M. Ferrando, *An OpenScourse Approach for the Prediction of Planing Hull Resistance*, University of Genova, 2014

[21] S. Brizzolara, F. Serra, *Accuracy of CFD Codes in the Prediction of Planing Surfaces Hydrodynamic Characteristics*, 2nd International Conference on marine Research and Transportation, ICMRT07. ISCHIA. 28-30 Giugno. (vol. 1, pp. A-1-A-12). ISBN: 88-901174-3-5

[22] P. R. Payne, *The Water Rise in Front of a Model Planing Hull*, Experiments in Fluids 17, 1996

Appendices

Appendix A - Top view of 3D-flow cases

This appendix shows all 3D-flow cases Figure 22-27 shows the bare hull cases and figure 28-30 shows the cases with deflectors. The water surface is represented by 0.5 volume fraction contour levels plotted are local y-position.

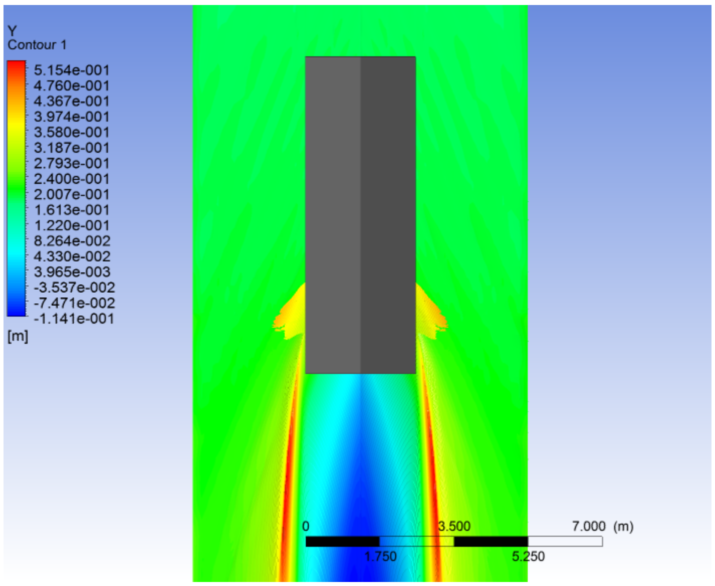


Figure 21: 10 degree deadrise angle.

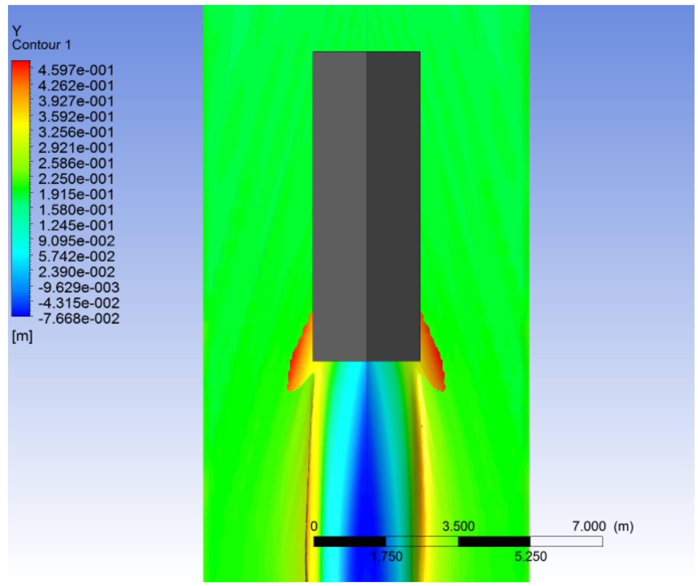


Figure 22: 15 degree deadrise angle

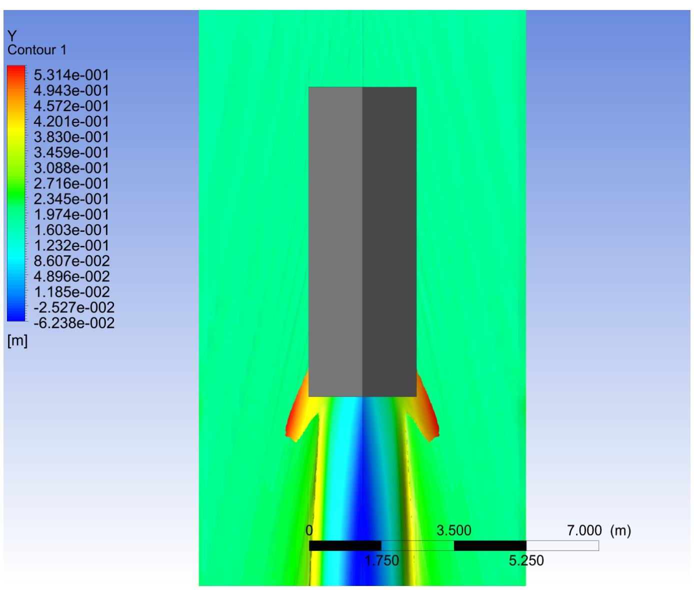


Figure 23: 18,6 degree deadrise angle.

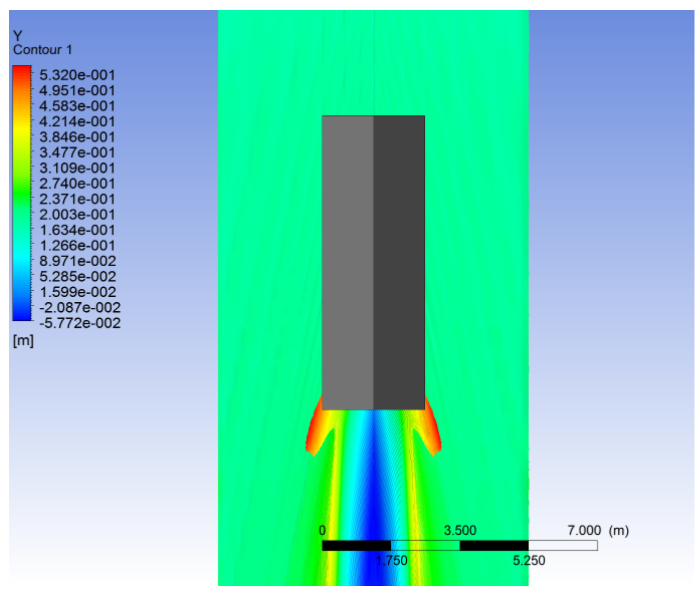


Figure 24: 20 degree deadrise angle

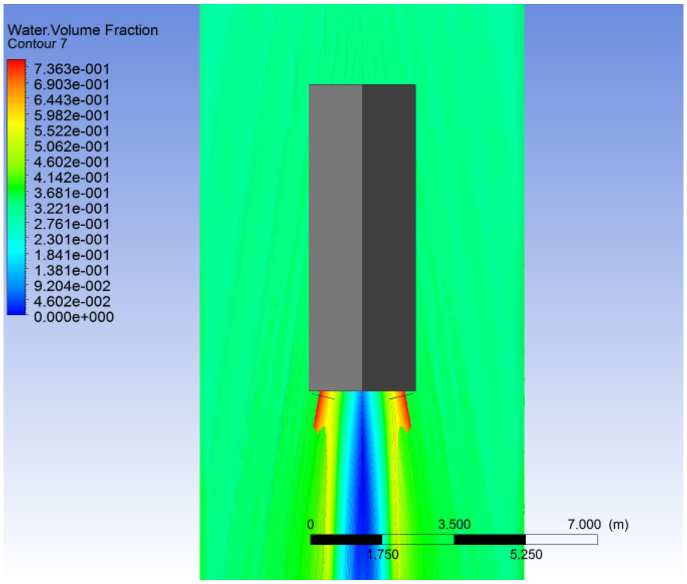


Figure 25: 25 degree deadrise angle.

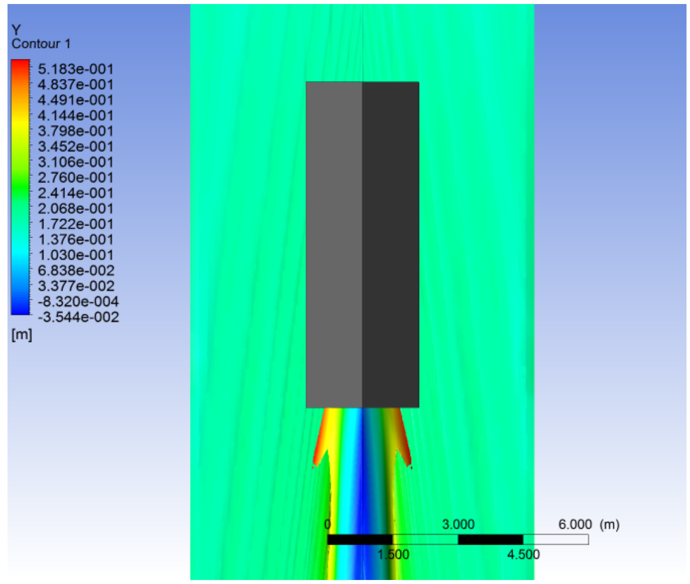


Figure 26: 30 degree deadrise angle

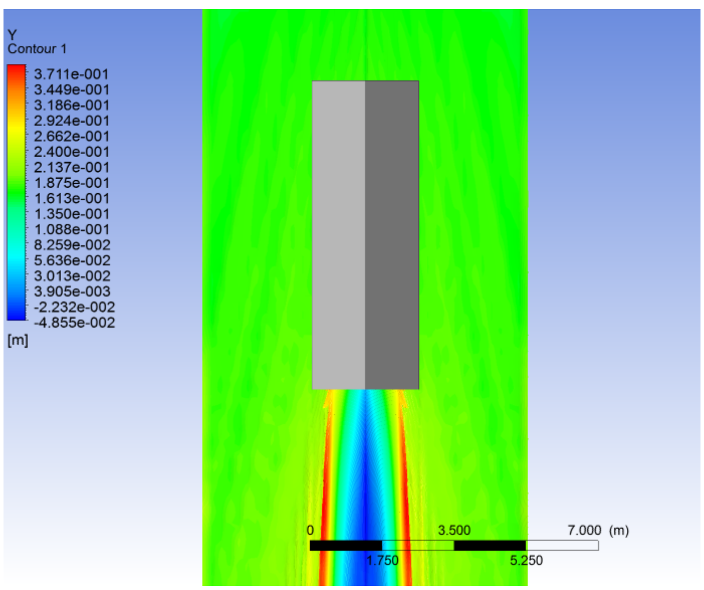


Figure 27: Case 1.

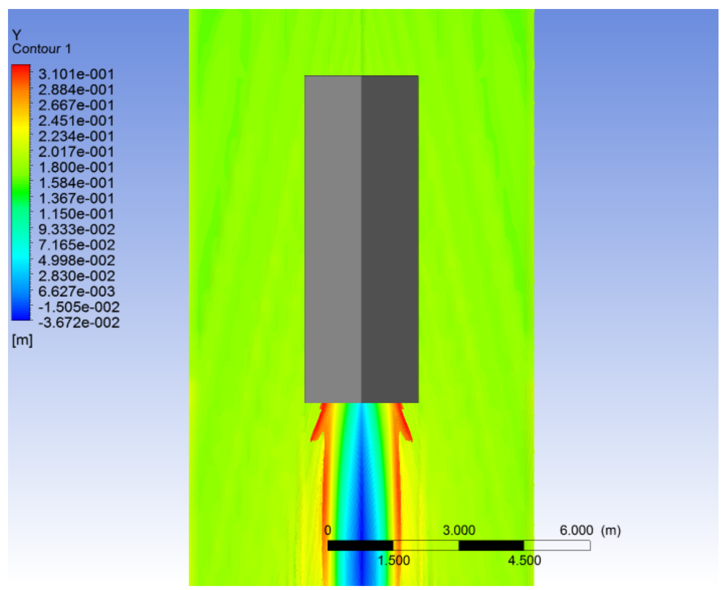


Figure 28: Case 2

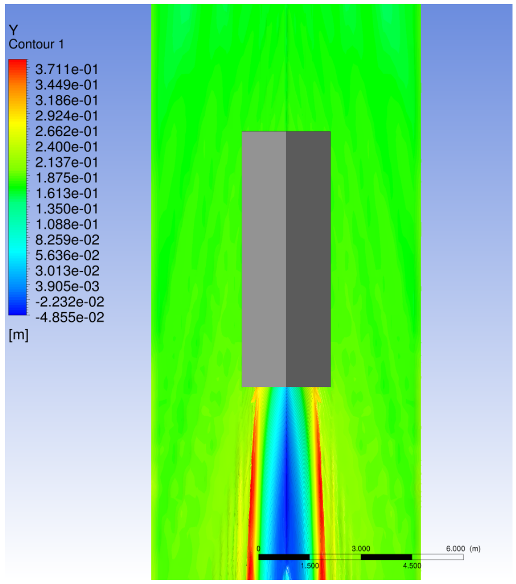


Figure 29: Case 3.

Appendix B - Recovery of energy

The energy content in the spray provides an opportunity to recycle this energy into favourable forces on the hull. Crucial to understanding the possibility to recover energy in the spray is the characteristics of the spray, i.e. thickness, velocity and direction of the spray sheet. As described by Savitsky [8] the direction of the spray is merely a reflection of the velocity vectors about the stagnation line, analogous with a beam of light reflected in a prism.

Since the boat is moving in a room fixed reference plane, a Gallilieo transformation allows the boat to be fixed and the water to flowing past it, the transformation is a fair representation since the forces acting on the hull are all invariant.

Applying the conservation in momentum over the deflectors, see figure 30, a theoretical efficiency of the recovered energy is established.

$$\nearrow: \hat{x}\vec{F} = -F_x = \hat{x}\dot{m}[V_{out} - V_{in}] \tag{10}$$

$$\hat{x}V_{out} = -(V - u) \tag{11}$$

$$\hat{x}V_{in} = (V - u) \tag{12}$$

$$\Rightarrow F_x = 2\rho A(V - u) \tag{13}$$

Thus the generated power is found by:

$$F_{out} = F_x u \tag{14}$$

And the power content in the spray is given by:

$$F_{in} = \frac{1}{2}\dot{m}V^2 = \frac{1}{2}\rho V^3 A \tag{15}$$

The theoretical efficiency is now found by:

$$\eta = \frac{P_{out}}{P_{in}} = 4(1 - \frac{u}{V})\frac{u}{V} \tag{16}$$

The efficiency is thus a function of the relative velocity of the spray and the deflector. Maximum is found by differentiate with respect to this velocity fraction.

$$\frac{d\eta}{d\frac{u}{V}} = 4(1-\frac{u}{V})(1-3\frac{u}{V}) = 0 \tag{17}$$

Extreme values found when:

$$\frac{u}{V} = 1 \tag{18}$$

$$\frac{u}{V} = \frac{1}{3} \tag{19}$$

(18) being the trivial solution and (19) gives the global maxima. Inserting (19) into (16) gives the efficiency η=60% i.e. 60% of the energy content in the spray can be recovered into forward thrust.

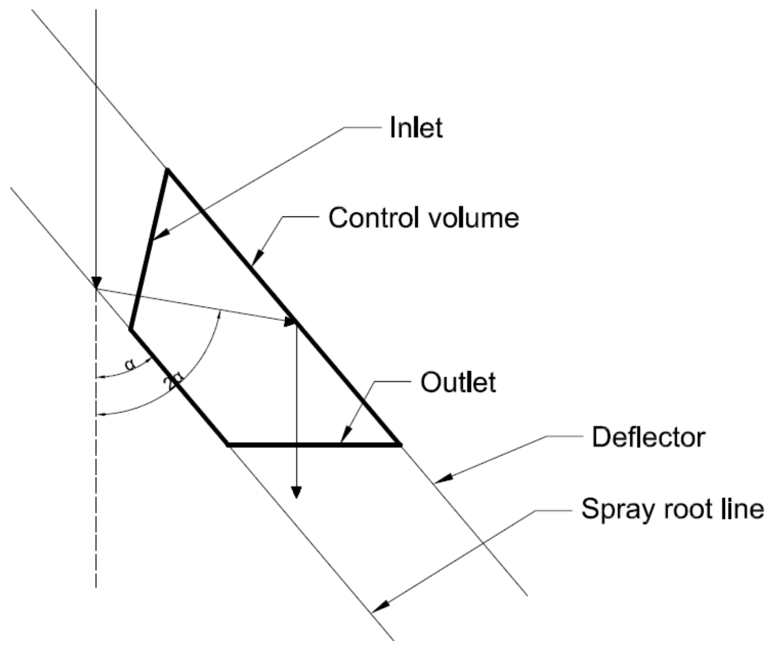


Figure 30: Control volume.

Appendix C - Difference between theory and simulations

This appendix shows the difference between the simulated spray thickness fraction of the wetted length and corresponding theoretical values. This is done in terms of mean average error (M.A.E) and standard deviation.

Table 11: Difference between 2D-simulation and theory by Savitsky

Angle	MAE	Savitsky	Mean value	Standard deviation
5°	0.67 %	0.0051	0.0037	2.858e-4
7.5°	1.1%	0.011	0.0087	0.0014
10°	1.64 %	0.0186	0.0152	0.0027
15°	1.64 %	0.0381	0.0344	0.0020
20°	2.88%	0.0618	0.0555	0.0031
25°	2.93%	0.0888	0.0826	0.0045
30°	1.88%	0.0118	0.1123	0.0146

Table 12: Difference between 3D-simulation and theory by Wagner

Angle	MAE	Wagner	Mean value	Standard deviation
10°	1.22 %	0.0034	0.0065	3.40e-4
15°	1.87 %	0.0082	0.0119	2.88e-4
18.5°	1.62 %	0.0134	0.0164	5.03e-4
20°	1.58 %	0.0135	0.0187	7.68e-4
25°	0.84 %	0.0265	0.0283	0.0016
30°	1.49 %	0.0425	0.0442	0.0032

KTH Royal Institute of Technology
Centre for Naval Architecture, Teknikringen 8, SE-10044 Stockholm, Sweden
Department of Mechanics, Osquars Backe 18, SE-10044 Stockholm, Sweden
www.kth.se