3D CFD-analysis of conceptual bow wings

PAUL NIELSEN
paulni@kth.se
0707-384496

Master Thesis
Ver. 0.1

KTH Center for Naval Architecture
February 2011
ABSTRACT

As a small step towards their long-term vision of one day producing emission free vessels, Wallenius employed, in 2009, Mårten Silvanius to carry out his master thesis for them in which he studied five different concepts to reduce the overall fuel consumption using wind powered systems. The vessel on which his study was performed is the 230 m LCTC vessel M/V Fedora. One of the concepts studied was the bow wing which is thought to generate enough force in the ship direction to profitably reduce the overall wind resistance. His calculations showed that the wing would be the preferred method of the different concepts studied since it was determined cheapest to build, had good payback, had good global drag reducing effects and had a predicted performance of a reduction in fuel cost between 3-5% on a worldwide route.

This thesis is conducted mainly to verify the results of Silvanius numerical study. The method chosen is to perform a fully viscous 3-D CFD study on the entire flow around the above water portion of the ship in full scale. A 3-D model is created and the wing is placed using suggestions given by Silvanius.

One major limitation in this project was the computational capacity available at the time this thesis was conducted. In order to run some of the viscous grids created the grids had to be severely coarsened. This had a negative impact on the reliability on some of the results.

Since it has been difficult to obtain satisfactory solutions, no work has been done to optimize the shape and position of the wing.

Nevertheless, one it has been shown that the wing does in fact affect the resistance in a positive way, however nowhere near as much as predicted by Silvanius. This effect needs to be further determined through further calculations, both using CFD and also through experimental wind tunnel testing where alternatives to the wing profile should be tested, e.g. replacing the wing with a vortex generator to further delay the point of separation.
ACKNOWLEDGEMENTS

First and foremost I would like to thank one of my supervisors in this project, Maximilian “Mio” Tomac at KTH, without whom I would be completely lost in the jungle that is CFD.

I would also like to thank my other supervisors, Karl Garme (KTH) and Mikael Huss (Wallenius), for their contributions during the different stages of this thesis.

Furthermore I would like to acknowledge the support that Exjobbsgruppen has given me, especially one member, Carl Hedgren, for his spontaneous disruptions and rewarding discussions which in no way helped me forward in the project but were nevertheless highly appreciated.

Lastly I also would like to express my gratitude to the staff at the department of Ship design at Wallenius Marine for providing me with a pleasant working environment during my short stay at their offices at the start of this project.

Paul Nielsen, February 2011.
6.1.1. Concerning the bow effect.................................................................43
6.2. Effects when adding the wing..........................................................44
   6.2.1. Effects on the flow structure ....................................................45
   6.2.2. Force generated from wing .....................................................47
6.3. Effects of altering mounting position and sheet angle .....................47
7. Conclusions..........................................................................................49
8. Recommendations..................................................................................49
Appendix A – Pressure distribution comparisons .....................................50
Appendix B – Streamline comparisons ....................................................52
Appendix C – Velocity field comparisons ...............................................55
References...............................................................................................57
### Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-D/3-D</td>
<td>Two-/Three-Dimensional</td>
</tr>
<tr>
<td>BC</td>
<td>Boundary Conditions</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Friedrichs-Levy condition</td>
</tr>
<tr>
<td>DES</td>
<td>Detached Eddy Simulation</td>
</tr>
<tr>
<td>E#</td>
<td>Euler mesh</td>
</tr>
<tr>
<td>ECMWF</td>
<td>European Center for Medium-range Weather Forecasts</td>
</tr>
<tr>
<td>FOI</td>
<td>Swedish Defence Research Agency (Sw. Totalförsvarets Forskningsinstitut)</td>
</tr>
<tr>
<td>GB</td>
<td>Gigabyte</td>
</tr>
<tr>
<td>GUI</td>
<td>Graphical User Interface</td>
</tr>
<tr>
<td>IGES</td>
<td>Initial Graphics Exchange Specification</td>
</tr>
<tr>
<td>KTH</td>
<td>Royal Institute of Technology (Sw. Kungliga Tekniska Högskolan)</td>
</tr>
<tr>
<td>LCTC</td>
<td>Large Car and Truck Carrier</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>M/V</td>
<td>Merchant Vessel</td>
</tr>
<tr>
<td>N-S</td>
<td>Navier-Stokes</td>
</tr>
<tr>
<td>NACA</td>
<td>National Advisory Committee for Aeronautics</td>
</tr>
<tr>
<td>PLM</td>
<td>Product Lifecycle Management</td>
</tr>
<tr>
<td>R#</td>
<td>RANS mesh</td>
</tr>
<tr>
<td>RAM</td>
<td>Random Access Memory</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes equations</td>
</tr>
<tr>
<td>Rhino</td>
<td>Rhinoceros</td>
</tr>
<tr>
<td>RORO</td>
<td>Roll-On Roll-Off</td>
</tr>
<tr>
<td>RK</td>
<td>Runge-Kutta</td>
</tr>
<tr>
<td>WB</td>
<td>ANSYS Workbench</td>
</tr>
<tr>
<td>WT</td>
<td>Wind Tunnel</td>
</tr>
<tr>
<td>ZERO</td>
<td>Zero Emission RoRo</td>
</tr>
</tbody>
</table>
## Nomenclature

<table>
<thead>
<tr>
<th>Latin symbols</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_F$</td>
<td>Projected frontal area of above water portion of ship</td>
<td>$[m^2]$</td>
</tr>
<tr>
<td>$A_S$</td>
<td>Projected side area of above water portion of ship</td>
<td>$[m^2]$</td>
</tr>
<tr>
<td>$A_w$</td>
<td>Projected area of wing</td>
<td>$[m^2]$</td>
</tr>
<tr>
<td>$B$</td>
<td>Maximum width of ship</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$C_D$</td>
<td>3-D drag coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>$C_d$</td>
<td>Non-dimensional force coefficient in ship direction</td>
<td>[-]</td>
</tr>
<tr>
<td>$C_f$</td>
<td>Skin friction coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>$C_L$</td>
<td>3-D lift coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>$C_n$</td>
<td>Non-dimensional side force coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Non-dimensional moment coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>$C_X$</td>
<td>Non-dimensional force coefficient in ship direction</td>
<td>[-]</td>
</tr>
<tr>
<td>$\Delta C_X$</td>
<td>Difference of the non-dimensional force coefficient in ship direction</td>
<td>[-]</td>
</tr>
<tr>
<td>$D$</td>
<td>The height of the ship measured from the keel line to weather deck</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$D$</td>
<td>Drag force generated by wing</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$F_i$</td>
<td>Arbitrary force</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$F_{x_{hi}}$</td>
<td>Aerodynamic force acting on hull in ship direction</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$F_{y_{hi}}$</td>
<td>Aerodynamic side force acting on hull</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$F_{x_{hi}}$</td>
<td>Aerodynamic force acting on wing in ship direction</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$F_{y_{hi}}$</td>
<td>Aerodynamic side force acting on wing</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$F_{h_i}$</td>
<td>Hydrodynamic force acting on hull</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$F_{h_p}$</td>
<td>Hydrodynamic force acting on propeller</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$F_{h_{ri}}$</td>
<td>Hydrodynamic force acting on rudder</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$G$</td>
<td>Function loosely dependant on $\ln(Re)$</td>
<td>[-]</td>
</tr>
<tr>
<td>$k$</td>
<td>Kinetic energy</td>
<td>$[J]$</td>
</tr>
<tr>
<td>$k-\omega$</td>
<td>Turbulence model type</td>
<td>[-]</td>
</tr>
<tr>
<td>$L$</td>
<td>Lifting force generated by wing</td>
<td>$[N]$</td>
</tr>
<tr>
<td>$L$</td>
<td>Characteristic length</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$L_{OA}$</td>
<td>The overall length of the ship</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$L_{PP}$</td>
<td>The length between the ships perpendiculars</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$n$</td>
<td>Number of time steps</td>
<td>[-]</td>
</tr>
<tr>
<td>$P_i$</td>
<td>Arbitrary power</td>
<td>$[W]$</td>
</tr>
<tr>
<td>$Q$</td>
<td>Dynamic pressure</td>
<td>$[Pa]$</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
<td>[-]</td>
</tr>
<tr>
<td>$Re/m$</td>
<td>Reynolds number over length</td>
<td>$[m^{-1}]$</td>
</tr>
<tr>
<td>$T$</td>
<td>The design draft of the ship</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$U_{\infty}$</td>
<td>Magnitude of free stream velocity</td>
<td>$[m/s]$</td>
</tr>
<tr>
<td>$(u_e, v_e, w_e)$</td>
<td>Free stream velocity components in the $(x, y, z)$ direction</td>
<td>$[m/s]$</td>
</tr>
<tr>
<td>$u_{\tau}$</td>
<td>Wall friction velocity</td>
<td>$[m/s]$</td>
</tr>
<tr>
<td>$V_A$</td>
<td>Apparent wind speed</td>
<td>$[m/s]$</td>
</tr>
<tr>
<td>$V_S$</td>
<td>Ship speed</td>
<td>$[m/s]$</td>
</tr>
<tr>
<td>$V_T$</td>
<td>True wind speed</td>
<td>$[m/s]$</td>
</tr>
<tr>
<td>$x$</td>
<td>Characteristic length</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$(X, Y)$</td>
<td>Global coordinates</td>
<td>[-]</td>
</tr>
<tr>
<td>$(x, y)$</td>
<td>Local coordinates</td>
<td>[-]</td>
</tr>
<tr>
<td>$y$</td>
<td>Normal distance to nearest wall</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$y^*$</td>
<td>Dimensionless viscous wall distance</td>
<td>[-]</td>
</tr>
<tr>
<td>Greek symbols</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>---------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>--------------------</td>
</tr>
<tr>
<td>α</td>
<td>Vertical angle of attack</td>
<td>[°]</td>
</tr>
<tr>
<td>β</td>
<td>Horizontal angle of attack</td>
<td>[°]</td>
</tr>
<tr>
<td>β</td>
<td>Apparent wind angle</td>
<td>[°]</td>
</tr>
<tr>
<td>β_{local}</td>
<td>Local apparent wind angle</td>
<td>[°]</td>
</tr>
<tr>
<td>γ</td>
<td>True wind direction</td>
<td>[°]</td>
</tr>
<tr>
<td>δ</td>
<td>Sheet angle</td>
<td>[°]</td>
</tr>
<tr>
<td>ζ</td>
<td>Mounting position of wing measured from the ships centerline</td>
<td>[°]</td>
</tr>
<tr>
<td>η_{ship}</td>
<td>Ship propulsion efficiency</td>
<td>[-]</td>
</tr>
<tr>
<td>κ</td>
<td>Karmans constant, = 0.41</td>
<td>[-]</td>
</tr>
<tr>
<td>λ</td>
<td>Drift angle</td>
<td>[°]</td>
</tr>
<tr>
<td>μ</td>
<td>Dynamic viscosity</td>
<td>[Pa·s]</td>
</tr>
<tr>
<td>ν</td>
<td>Kinematic viscosity</td>
<td>[m²/s]</td>
</tr>
<tr>
<td>ν′</td>
<td>Quantity equivalent to eddy viscosity</td>
<td>[m²/s]</td>
</tr>
<tr>
<td>ρ</td>
<td>Fluid density</td>
<td>[kg/m³]</td>
</tr>
<tr>
<td>τ_{wall}</td>
<td>Shear stress at the wall</td>
<td>[N/m²]</td>
</tr>
<tr>
<td>ω</td>
<td>Specific dissipation</td>
<td>[m²/s³]</td>
</tr>
</tbody>
</table>
1. INTRODUCTION
During 2009 Wallenius Marine conducted a project called ZERO (Zero Emission RORO) where a plan was created for how to reach their long-term vision of one day producing emission free vessels. A major part of their vision is the increased exploitation of emissionless sources of energy, such as solar, wave and wind energy. The source in which the largest amount of energy can be extracted is presumed to be from wind [1].

In 2009, Mårten Silvanius conducted his master thesis for Wallenius in which he studied how different wind powered systems could decrease the fuel consumption for a 230 m LCTC (Large Car and Truck Carrier) vessel. The vessel which served as norm was the M/V Fedora (Figure 1) which is a relatively new addition to the Wallenius fleet. This vessel is assumed to operate on a world-wide route, 220 days each year, with an average daily fuel consumption of 56 tonnes/day. The average fuel consumption is based on wind statistics provided by the ECMWF (European Center for Medium-range Weather Forecasts) when going at the ships design speed of 20 knots.

![Figure 1. The M/V Fedora [2].](image)

Silvanius studied a total of five different wind powered systems; the Flettner rotor, the wing sail, the kite, the horizontal and vertical axel wind turbine. His studies showed that the preferred system would be to install vertical wings at the bow of the ship as this had a good payback investment, was cheap to build and maintain and had a measurable performance. The results in his thesis [3] showed that the vertical bow wings could save up to 3-5% of the fuel consumption. The potential of these wings are further investigated in this thesis.

1.1. PURPOSE AND OBJECTIVE
Silvanius thesis only covered a minor 2-D CFD (Computational Fluid Dynamics) analysis showing the airflow around the bow area (without wings) of the ship. In order to completely understand how bow wings would affect the flow around the entire ship a complete 3-D CFD-analysis is a good start, preferably followed by wind tunnel testing. A 3-D analysis will also be able to take in to account some aspects that were not considered in the 2-D analysis, such as the reduced effect to global drag due to delayed separation and how the new airflow affects the wake.

The overall goal of this thesis is to determine how large the potential is for a decrease in fuel consumption for a LCTC vessel whilst fitted with bow wings. Furthermore the shape and position of these wings are to be evaluated in order to reach maximum effect. Also, the results from Silvanius thesis are to be verified through this thesis. By performing a complete 3-D analysis of the above water portion of the ship, all the aforementioned objectives can be reached.

1.2. METHOD
The main objective is to analyze the air flow around the above water portion of a LCTC vessel. One method of doing this is to perform a complete 3-D CFD-analysis of the vessel in question. This basically means solving the Navier-Stokes (N-S) equations, which define any single-phase fluid flow [4], using computer software specially developed for these types of problems.
Firstly, a model of the ship without bow wings (later referred to as the naked model) will be created and run through the CFD process (briefly covered below) mainly to be used as reference to the case with wings. Also the results of the naked model will provide information regarding where and in which area of the bow that it would be beneficial to position the wings. Furthermore the results for the naked model will be compared to those from the 2-D calculations performed by Silvanius.

1.2.1. Basic CFD Procedure

For starters a 3-D model needs to be created featuring only the above water portion of the ship in question, where all details that has insignificant impact on the overall results are removed since these details would only add to the amount of computer work needed further on in the CFD process. One can use any 3-D modeling tool to complete this task providing that the software used is capable of exporting the model in the appropriate file format.

When satisfied with the geometrical 3-D model it needs to be discretized properly to form a volume mesh befitting the case under investigation. Firstly an Euler-mesh is created (used for inviscid Euler calculations) and then run through the flow solver in order to evaluate the quality of the mesh. For the viscous RANS (Reynolds-Averaged Navier-Stokes) calculations a RANS-mesh is created where prismatic layers are added to the Euler-mesh in order to be able to better resolve for the boundary layer. Thereafter the appropriate method of modeling the turbulence needs to chosen and boundary conditions need to be set along with a number of parameters in order to reach convergence. The residuals and relevant results are thereafter visualized and analyzed.

1.2.2. Software

The vessel under investigation is a relatively large object that will require a large number of elements to accurately describe the flow. Also this requires that the dimensions are described as precisely as possible. As of today there exists no software which includes all necessary parts of the CFD process, therefore the job is divided into four kinds of software, each designed to carry out one specific task; 3-D modeling, mesh generating, flow solving and data visualization.

The 3-D model of the hull from the baseline up to the weather deck is provided by Wallenius. The desired superstructures are created in Solid Edge (from Siemens PLM Software) and the wings are created using Rhinoceros (developed by Robert McNeel & Associates and later referred to as Rhino). The main difference between the two programs is that Solid Edge creates solids (as the name implies) and Rhino primarily creates surfaces which can later be made into solids. However, since only the surfaces are going to be meshed it will not matter if the model is solid or only a shell. Although both programs are equally capable of creating all intended objects on its own, Solid Edge was chosen to create the superstructures as it is easier and quicker to create large blocks whilst the wing was created in Rhino simply because it has a function that can import lines and curvatures from a PDF-file, which greatly simplifies the task of creating an airfoil with exact dimensions and curvatures.

The computational domain and volume net is generated using ANSYS ICEM CFD (later referred to as ICEM), which is a flexible mesh generating tool for the ANSYS Workbench [5]. It is capable of creating large volume or surface meshes with a wide range of shell format variations and mesh generating methods. Although this software is also capable of creating and modifying geometric models, it is not as accurate as software specifically designed for this purpose. The geometric modeling tools are mainly used to patch up and modify existing models created elsewhere that may have been damaged (e.g. disrupted surface connectivity, gaps etc.) during the exporting and importing process.

The flow solver used to perform the computations is the Edge-code. Edge is a CFD flow solver for unstructured grids that solves the three-dimensional, compressible RANS equations on hybrid grids. It solves problems in both viscous (using Navier-Stokes equations) and inviscid (using Euler equations which are Navier-Stokes equations simplified) flow on arbitrary elements [6]. Edge was developed by FOI (the Swedish Defence Research Agency) and is one of the leading commercial flow solvers being used. The governing equations are integrated explicitly towards steady state with Runge–Kutta time integration. Results like pressure distributions and aerodynamical coefficients are obtained and can be graphically pre-
sented in visualization applications such as Paraview. Edge can be used for steady state calculations as well as time accurate ones including manoeuvres and aeroelastic simulations. Edge is a purely aerodynamic flow solver as it is intended for aircraft calculations. This means that for hydrodynamic calculations one would have to use another flow solver. However since only the above water portion of the ship is under investigation, Edge will be enough to complete the task at hand.

In order to be able to compute several cases (one case being either an Euler or RANS computation, naked or winged model and one specific apparent wind angle) at the same time the computations will be run on a high-performance computer available at KTH (the Royal Institute of Technology) that utilizes parallel computing. Each case is allocated to a node connected to this computer. The computer has a total of 640 nodes [7], where each computer node is allocated 8 GB of RAM (Random-Access Memory).

The results are visualized using ParaView, which is a multi-platform data analysis and visualization application where one can analyze their data using quantitative and qualitative methods. ParaView is capable of analyzing datasets of all sizes and can therefore be used on supercomputers as well as on laptops [8].

1.3. RESOURCE LIMITATIONS

At the time when this thesis was conducted there was a limit on how fine the mesh could be, or rather how much RAM that was available at computing nodes. The required memory needed is directly proportional to the amount of grid points present in the mesh [9], e.g. a mesh containing 5 million grid points would require around 5 GB of RAM. Therefore any mesh used on the high-performance computer needs to contain ≤ 8 million grid points. This criterion is met by most cases except for the RANS-mesh for cases where the wing is present which contains around 12-17 million grid points.

One effect is that the number of prismatic layers in the RANS-mesh for the winged model had to be reduced from 50 (as for the naked model) to 30 layers since the addition of the wing significantly increases the total amount of grid points and thusly greatly increases the amount of RAM necessary to handle the model appropriately. Also the entire surface mesh had to be coarsened (as shown in Figure 2) in order to be able to run computations on the high-performance computer.

![Figure 2. The surface mesh at the bow as it is for most cases (left) and the coarsened mesh for the winged RANS calculations (right).](image)

Computations will however be done on the original mesh (as well as on a further refined mesh), one case at a time, on a local computer with a greater RAM capacity. Since only one case can be computed at a time only a few selected number of apparent wing angles will be investigated.

2. STUDIED SHIP & THE BOW WING

The panamax vessel Fedora is chosen to serve as norm for comparative reasons since the same vessel was used in Silvanius studies. With a maximum capacity of 8000 car units (1 car unit = 8.4 m²) the M/V Fedora makes for one of the largest car carriers in the world [2] and, for Wallenius, it is of primary interest to lower its daily fuel consumption in any way possible. The principal particulars of Fedora can be seen in Table 1.
Table 1. Principal particulars of the M/V Fedora.

<table>
<thead>
<tr>
<th></th>
<th>[m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length over all, (L_{OA})</td>
<td>227.80</td>
</tr>
<tr>
<td>Length between perpendiculars, (L_{PP})</td>
<td>219.30</td>
</tr>
<tr>
<td>Breadth, (B)</td>
<td>32.26</td>
</tr>
<tr>
<td>Depth, (D)</td>
<td>34.70</td>
</tr>
<tr>
<td>Design draft, (T)</td>
<td>9.50</td>
</tr>
<tr>
<td>Projected frontal area, (A_F)</td>
<td>1132</td>
</tr>
<tr>
<td>Projected side area, (A_S)</td>
<td>6885</td>
</tr>
</tbody>
</table>

The value of the projected frontal and side areas of the ship are based on only the above water portion of the ship. A blunt body such as this generates a large wind resistance. However the Fedora comes with the characteristic semicircular bow shape which, in Silvania studies, proved to be advantageous in terms of local acceleration and redirection of the airflow around it (see Figure 3).

![Figure 3. Topview of the bow where the different angles are defined [3].](image)

\(V_A\) is the apparent wind speed and the angle in which it attacks from is the apparent wind angle, \(\beta\) (measured from the centerline). The local apparent wind angle \(\beta_{local}\) refers to the altered apparent wind angle which comes from the assumed redirection of the airflow due to the bow. The assumption is that as long as \(\beta<90^\circ\) the flow will remain parallel to the hull until separation occurs and that the wind direction only depends on the mounting position, \(\zeta\) (measured from the centerline), through

\[
\beta_{local} = 90^\circ - \zeta .
\] (1)

The angle \(\delta\) is the sheet angle of the wing and \(L\) and \(D\) are the resulting lift and drag forces from the wing.

Mounting wings at the bow is thought to have mainly three advantages that are to be confirmed with CFD calculations. Firstly, in favorable conditions, the wings are thought to generate a contributing lifting force in the ship direction, large enough to profitably affect the overall resistance of the ship. Secondly the heeling moment generated from wings mounted at the bow is thought to be less than would they be mounted on deck. Lastly it is mentioned but not investigated that the wings improve the airflow around the vessel by delaying the point of separation and thusly reducing the drag.
2.1. THE INVESTIGATED SETUP

Besides calculating the vessel without wing, this thesis primarily focuses on the setup of the wing suggested by Silvanius. This setup is presented in Table 2.

Table 2. Wing setup as suggested by Silvanius.

<table>
<thead>
<tr>
<th>Setup</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Mounting position, $\zeta$</td>
<td>$20^\circ$</td>
</tr>
<tr>
<td>Sheet angle, $\delta$</td>
<td>$13^\circ$</td>
</tr>
<tr>
<td>Distance from hull</td>
<td>4 m</td>
</tr>
<tr>
<td>Wing profile</td>
<td>NACA 6412</td>
</tr>
</tbody>
</table>

Apart from the suggested setup, two other wing positions, $10^\circ$ and $30^\circ$ from the centerline, and also one other sheet angle, $20^\circ$, was investigated. The calculations were done on a full-scale 3-D model of the ship and the apparent wind angle, $\beta$, was taken as every $5^\circ$ from $0^\circ$-$45^\circ$ from the centerline with an apparent wind speed, $V_A$, of 20 knots in every direction.

3. NUMERICS

This chapter covers the basic numerics used in this thesis. The first section (Section 3.1) loosely covers how Silvanius approached his study through basic ship theory. The objective of this section is to describe how Silvanius results will be evaluated numerically. This is done by first describing shortly how he approached his assignment followed by which aspects are considered in this thesis. The second section (Section 3.2) discusses the governing equations and numerical methods of solving them in the CFD process through brief descriptions and only the default methods in the flow solver is mentioned. For more detailed information of how the governing equations are derived, how the default numerical methods are composed and also information on other methods available, see [15] or [17].

3.1. BASIC SHIP THEORY: SILVANIUS APPROACH

Basically what Silvanius did was determining at which angle of attack a certain wing profile generated maximum lifting force, subsequently the effect gained from this force, and then subtracted its component in the ship direction from the total engine power needed to power the ship (which was calculated separately) whilst taking into account for some aspects that affect the total power gained from the wing.

The lifting, $L$, and drag, $D$, force of the wing seen in Figure 3 are calculated through

$$ L = qA_wC_l $$
$$ D = qA_wC_D $$

(2)

where, $A_w$ is the projected area of the wing in the direction of the force, $C_l$ and $C_D$ are dimensionless 3D lift and drag coefficients and $q$ is the dynamic pressure which, for incompressible fluids (which is assumed for air due to the relatively low speed of the ship), is calculated

$$ q = \frac{\rho V_A^2}{2} $$

(3)

where $\rho$ is the fluid density and $V_A$ is the apparent wind speed which is calculated through

$$ V_A = \sqrt{V_T^2 + V_S^2 - 2V_TV_S \cos \pi - \gamma} $$

(4)

and the apparent wind angle, $\beta$, is calculated through

$$ \beta = \cos^{-1} \left( \frac{V_T^2 - V_S^2 - V_S^2}{-2V_TV_S} \right) $$

(5)
where $V_T$ is the true wind speed, $V_S$ is the ship speed and $\gamma$ is the true wind direction. All speeds and angles are defined in Figure 4.

The lift and drag forces from the wing are not of primary interest as much as how they each contribute to the total aerial resistance in the ship's direction. Using the apparent wind direction these forces are translated to form the generated force in the ship direction $F_{x_A}$ and the side force $F_{y_A}$ (both of which are defined in Figure 4) through

$$
\begin{bmatrix}
F_{x_A} \\
F_{y_A}
\end{bmatrix} =
\begin{bmatrix}
\cos \beta & \sin \beta \\
-\sin \beta & \cos \beta
\end{bmatrix}
\begin{bmatrix}
-D \\
L
\end{bmatrix}
$$

Assuming that the flow is redirected as discussed in the previous chapter the apparent wind angle $\beta$ in (6) is replaced with the local apparent wind angle, $\beta_{local}$ as defined in Figure 3. Furthermore, the assumption provides that the wing will always be attacked from the same angle, the sheet angle $\delta$, and thusly always have the same lift and drag coefficients, $C_L$ and $C_D$ (determined through an online Java script, JavaFoil [10]), regardless of the apparent wind angle as long as $\beta<90^\circ$.

The aerodynamic force, $F_A$ (defined in Figure 4), acting on the hull is calculated separately in a similar fashion as the lift and drag forces in (2) through its components in the side and ship direction

$$
\begin{align*}
F_{x_A} &= qA_F C_x \\
F_{y_A} &= qA_S C_y
\end{align*}
$$

where again $q$ is the dynamic pressure determined through (3), $A_F$ and $A_S$ are the projected frontal and side area respectively of the above water portion of the ship and $C_x$ and $C_y$ are the force coefficients in the ship and side direction. The values used in Silvanius thesis for the dimensionless coefficients were determined through wind tunnel tests on a 1:200 scale model of the ship in question by DSME at Force technology in Denmark [3].

To translate any calculated force, $F_s$, into its corresponding propulsion power, $P$, the following equation is used

$$
P = \frac{FV_S}{\eta_{ship}}
$$

where $\eta_{ship}$ is the ship propulsion efficiency set to 0.75.

Other forces, variables and angles calculated in Silvanius thesis are defined in Figure 4 and Table 3.
Figure 4. Definition of forces, variables, angles and coordinate systems [3].

<table>
<thead>
<tr>
<th>Variable</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F_{H}$</td>
<td>Hydrodynamic force generated by rudder</td>
</tr>
<tr>
<td>$F_{H}$</td>
<td>Hydrodynamic hull forces</td>
</tr>
<tr>
<td>$F_{P}$</td>
<td>Propeller propulsion force</td>
</tr>
<tr>
<td>$(X, Y)$</td>
<td>Global coordinates in which the ship moves</td>
</tr>
<tr>
<td>$(x, y)$</td>
<td>Local coordinates</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Drift angle</td>
</tr>
</tbody>
</table>

The hydrodynamic forces mentioned in Table 3 will not be evaluated any further in this thesis.

The aerodynamic solver is capable of calculating the absolute forces acting on the entire ship, with or without wing, as well as the forces acting on defined sections of the model (e.g. the wing alone, the hull and the superstructures separately). However it does this by first determining the dimensionless force coefficients in the defined directions, i.e. $C_x$ and $C_y$. The translation in (6) is done automatically in the flow solver when properly defining the direction of the force coefficients.

This thesis will mainly present results showing the force coefficients and any comparison made will therefore be a comparison between the coefficients used in Silavnius thesis and those calculated here, the difference being that the force coefficients determined through CFD will contain information on how the wing affected the ships resistance whenever it is present.

3.2. THE GOVERNING EQUATIONS

The core of any CFD application could be to solve either potential equations, Euler equations or the Navier-Stokes (N-S) equations. The N-S describes how the pressure, temperature and density are related for a moving fluid. They comprise of a set of coupled partial differential equations; one continuity equation for the conservation of mass, three equations for the conservation of momentum and one equation for the conservation of energy, all of which are time-dependant [16]. Although theoretically possible, these equations are very difficult to solve analytically and are therefore more commonly solved on computers through approximations.

Normally one is only interested in knowing the time-averaged properties of the flow (e.g. mean velocities, mean pressure etc.) and not interested in considering all of the small-scale fluctuations in the atmosphere. Therefore some averaging operators are introduced adding some additional terms to the governing equations which represent the effects of the eddy fluxes created by the scales of motion which have been removed in the averaging process [17]. The new system of equations is more commonly referred to as the RANS (Reynolds-Averaged Navier-Stokes) equations and the additional terms are called the Reynolds stresses.
Turbulence models are developed and used in order to predict and approximate these stresses as well as the scalar transport terms and to close the system of mean flow equations so that turbulent flow can be computed using the RANS equations.

By neglecting the viscous effects altogether, the governing Navier-Stokes equations are reduced to the Euler equations, only the continuity equation remains identical [17]. The flow is assumed inviscid and therefore the no-slip condition at the wall boundaries is disregarded.

3.2.1. Steady state calculations

The flow solver used in this thesis (the Edge code) utilizes explicit Runge-Kutta (RK) schemes in order to integrate the governing equations towards steady state. The Runge-Kutta method is a numerical technique to approximate the solution of stochastic differential equations and advances in time in a step-by-step or “marching” manner [17]. The RK method is a multistage method which has the advantage that it does not require any specific starting procedure [22], other than the initial condition required by the differential equation itself, and the extra stages are used to improve the accuracy of the solution and to extend the stability region. A maximum of 10 stages is foreseen in Edge. The default scheme in Edge [15] is a 3-stage, first order accurate scheme which provides good smoothing for both upwind and central schemes. The numerical dissipation is only calculated once in this scheme in the first stage, which reduces the computational cost substantially. The dissipative terms are however stored separately. Convergence is also accelerated using agglomerated multigrids and implicit residual smoothing.

3.2.2. Time accurate calculations

Whenever the case under investigation is determined unsteady in nature, time accurate computations are applied. The flow solver used in this thesis has two methods of carrying out this type of calculations; explicit RK time marching with a global time step or implicit time marching with explicit subiterations [15].

For explicit time accurate calculations a global time step is automatically computed from the minimum local time step in all domains. A RK scheme of at least second order is recommended, however a fourth order scheme is default when explicit time accurate calculations are applied. Explicit calculations are not compatible with the convergence acceleration methods used for steady state calculations, such as multigrid methods or residual smoothing. Explicit calculations have however not been used in this thesis when time accurate calculations have been applied.

Instead implicit time accurate calculations have been used (since this is the default method when switching to time accurate calculations). For these calculations the domain has been divided into a number of user defined real time steps, within which the set of governing equations are solved through explicit RK time marching through a number of subiterations for each real time step. The default scheme for implicit time marching in the flow solver is a second order accurate backward difference scheme which provides good smoothing also for large implicit time steps [15].

4. Aspects regarding the CFD process

This chapter describes the CFD-process for this investigation. The first two subchapters (Sections 4.1 and 4.2) cover the modeling and discretization process and the final subchapter (Section 4.3) covers the Edge process. However this section only describes a general approach which was done in the initial stage of this project when the flow calculations were just started. Alterations to this approach is covered in the following chapter (Chapter 5) where methods to improve convergence are discussed and how this affected the model in this thesis.

4.1. 3D-modeling

A complete model comprising of the hull (shown in yellow in Figure 5) from the design draft to the weather deck of the ship was provided by Wallenius. Added to the model are the superstructures (shown in blue in Figure 5) that are necessary in order to gain a satisfactory understanding on how the airflow behaves.
Figure 5. The 3D model of the ship showing the hull (yellow) and the added superstructures (blue).

Note that all minor details, e.g. antennas and railings, have been removed (compare the model in Figure 5 with the ship depicted in Figure 1). The dimensions were taken from a detailed general arrangement of the vessel provided by Wallenius. The wing is added with a NACA 6412 profile at three different mounting positions, $\zeta; 10^\circ, 20^\circ$ and $30^\circ$ from the centerline, at a distance of 4 m from the hull (also a suggestion from Silvanius). It is not in any way connected to the hull in the model as presented in Figure 6. The reason for this setup is that it is not known how the wings will be connected if they are realized and that it would be sufficient this way to at least determine if it is a feasible solution.

Figure 6. The wing (purple) shown "floating" in the air at a position of 20° from the centerline.

The models are thereafter exported in the IGES-format (Initial Graphics Exchange Specification) and imported to ICEM where some further reparations and alterations are done. It is very common that defects in the geometry occur during this process. Such defects could be gaps, overlaps and discontinuities between surface patches [11]. As can be seen in Figure 7 a surface patch on the port side of the lower aft section of the hull (right) and the entire weather deck (left) had to be recreated in ICEM as it was lost during translation.

Figure 7. Defects on the model after importation.

Another side effect of the conversion is that multiple points, curves and surfaces occupying the same space may occur and at least the excess curves and surfaces need to be cleaned up (i.e. removed) since this could affect the discretization process and computations later on.

The vessel is oriented so that the point of origin is placed at the front most tip of the bulb and the coordinate system is similar to that of the flow solver. The orientation can be seen in Figure 5.
A vertical plane (shown in purple in Figure 8) is then added in ICEM at the lower aft section of the hull. This area has a narrow angle against the “waterline” where the hull is cut and would have required a large amount of small elements to properly reproduce the geometry in this area and since this area is not as important as other areas of the hull (e.g. the frontal area) and there is a limit on how many elements that may be used, the vertical plane was added.

![Figure 8. The vessel with (left) and without (right) the vertical plane (purple).](image)

Lastly the computational domain (or rather the farfield boundaries) is created (see Figure 9). The size of which has to be large enough as not to have any influence on the future computations [11]. The domain is shaped like a flat cylinder with a height of 500 m (i.e. \( \sim 2L_{OA} \)) where the ship is centered on the lower surface which is later defined as the symmetry plane. Initially two different radii were created to study whether or not this would have an effect on the results. These are 5.5\( L_{OA} \) and 8.5\( L_{OA} \), later referred to as 5L and 8L respectively. The volumetric mesh is assigned to a point created between the upper surface of the farfield and the symmetry plane (marked as FLUID in Figure 9).

![Figure 9. The farfield boundaries (green) and the symmetry plane (purple).](image)

When satisfied with the geometries the discretization process can take place.

### 4.2 Discretization

The objective of discretization is to divide the physical space where the flow is to be computed into a large number of geometric elements called grid cells [11] inside which the governing equations are solved for each element. The different geometric element types available in ICEM are presented in Figure 10.
Figure 10. The different element types; a) triangle, b) quadrilateral, c) tetrahedron, d) hexahedron, e) prismatic and f) pyramid.

Triangular and quadrilateral elements are used in 2-D grids and also on the surface mesh of a 3-D grid. The other elements presented in Figure 10 are used in volume meshing in 3-D grids.

4.2.1. Mesh type

Basically there exist two types of grids; structured and unstructured grids. Structured grids are characterized by regular connectivity that can be expressed by a two or three dimensional array which decreases the storage requirement and makes for efficient solver algorithms. Structured grids are restricted to quadrilateral elements in the surface mesh and hexahedral elements in the volume, an example of a structured grid is shown in Figure 11. A well designed structured grid gives a solution of high accuracy, however the process of generating such a grid is a tedious task (and can take months when dealing with complex geometries [9]).

Figure 11. Example of a structured grid.

Unstructured grids however could comprise of all elements shown in Figure 10 (in these cases the grid is more commonly referred to as a mixed or hybrid grid), but mostly they only consist of triangular elements in the surface mesh and tetrahedrons in the volume. The elements have no particular ordering and are placed in an irregular fashion, as shown in Figure 16. This type of grid is the most common in commercial solvers today. Due to the irregularity of the grid distribution the neighborhood connectivity of the different cells must be explicitly stored which in turn requires significantly more storage space than for structured grids. The generation process can be highly automated and offers high flexibility when dealing with local refinements of the grid without disturbing the overall grid distribution. Mainly for these reasons plus the somewhat complex geometry of the case, an unstructured grid is chosen for this thesis.

4.2.2. Mesh requirements

When creating a mesh there are some aspects that need to be considered since the future results highly depend on the quality of the grid. Both numerical stability and accuracy could be affected by a poor quality grid [9]. Naturally the resolution should be as high as possible, but higher resolution costs more in computing resources and generates slower turnaround times. Primarily there should be no gaps in the grid or overlapping of elements. Furthermore grid points should be clustered around areas of interest such as regions of large gradient (e.g. boundary layers, separation points and shocks), areas where one could expect changes in pressure and also around sharp corners or curves.
Also the transition from small to large elements should be smooth so that there are no abrupt changes in the volume of the grid cells. Areas of low interest could have relatively large elements so as to keep the total number of elements to a minimum. When present there should be no large kinks in the grid lines of quadrilateral and hexahedral elements as this could lead to a significant increase in numerical errors [11]. Lastly one needs to make sure to check orientation of the cell faces so that they are normal to the flow gradient.

4.2.3. Mesh method

A typical approach is to start by specifying the global maximum element size allowed to exist in the entire domain. This number is usually quite large in order to minimize the amount of elements needed to fill the entire computational domain. For this project the global element size is set to 200 meters. Obviously this is too large to properly reproduce the geometry of the ship, thus one need to specify element parameters for each surface on the ship. One can also specify element parameters on the curves, but it is not necessary to do this on every curve present. The elements sizes on the ship ranges from 0.05 m at the bulb to 4 m on the large horizontal surfaces on the superstructures and hull sides. The element sizes on the wing ranges from 0.01 m on the leading and trailing edges to 0.2 m on the sides. Figure 12 shows the different element parameters on the ship, unless specified otherwise the curves and surfaces have a maximum element size specification of 0.5 m.

For every curve and surface one can also specify a parameter denoted as tetra width. This ensures that the element size for the curve or surface in question does not increase in size until a number of tetrahedron, corresponding to the tetra width, away from the said curve or surface. As seen in Figure 12 this is only applied on the ship where a higher resolution is necessary; along the upper frontal edge of the bridge, bow upper and frontal lower edges, the midline from the weather deck all the way down to the bulb and along the edges at the aft. For the wing on the other hand a tetra width is applied on every surface as well as the leading and trailing edges, the result of which can be seen in Figure 16.

It is also possible to create or manipulate a mesh density volume in areas that are not adjacent to the geometry (e.g. the wake of an airfoil). This is done by creating what ICEM calls a mesh density (later referred to as a density field), shaped like a polyhedral (a rounded cylinder), in which one can prescribe a local maximum element size. For comparative reasons one such polyhedral is created surrounding the entire vessel, with a maximum element size of 3 m within the density field, in order to create a denser mesh close to the vessel. The density field is presented in Figure 13. The presence of a density field of this size naturally increases the number of elements significantly (see Table 5).
Prior to choosing generation method one must decide whether or not it should be patch dependent or independent. Patch dependent meshing creates a series of loops which are automatically defined by the boundaries of surfaces and/or a series of curves. The patch independent option on the other hand uses the geometry to associate the faces of the boundaries of the mesh to the regions of interest, thusly ignoring gaps, overlaps and other defects that would otherwise be problematic when generating a mesh. It is also usually slower than the patch dependent option. The choice is highly connected to the geometry appearance and quality. For complex geometries and geometries of poor quality (e.g. regarding surface representation etc.) it is recommended to choose the patch independent option [13].

There are several mesh generation methods available in ICEM. For unstructured meshes there are three methods to choose from; the Octree, Delaunay and Advancing front method.

The quickest and most robust method used throughout most of this thesis is the Octree method which is the only method that does not require an existing surface mesh in order to compute, thus saving time not having to do surface and volume mesh separately. The Octree method ensures refinement of the mesh where specified by the user and maintains larger elements where possible, allowing for faster computation. It starts out with a root tetrahedron covering the entire geometry and then subdivides it into eight tetrahedrons which in turn are each divided into eight new tetrahedrons and so on until all element size requirements are met. The default subdivision ratio is a factor of 2 [13], meaning that elements sharing an edge or face do not differ in size by more than a factor of 2, however in some areas it is preferred to specify a lower ratio (in ICEM this is referred to as the height ratio for a specific curve or surface) so that the changes in volume do not increase too rapidly.

One other method of generating a mesh was also used for comparison, namely the Advancing front. This method generates triangles by marching a front of free sides into the unstructured volume domain. The initial front is created by the prescribed boundary edges according to the element parameters previously determined for every surface, therefore it needs an existing surface mesh in order to operate. By extending this initial front, new triangles are created by using existing nodes or creating new ones. The front is thereafter updated once the new element is created in order to reflect changes due to this creation. This process continues until all sides in the front are removed and the domain is meshed.

Figure 14 shows a comparison between the two described methods.
The advancing front method shows a higher refinement close to the ship but with a faster increase in element size. Both methods result in a satisfactory overall grid distribution, however with relatively large volume steps away from the ship. This could affect the solution reliability. This however can only be investigated once they have gone through the flow solver.

4.2.4. Euler mesh

The unstructured Euler-mesh consists only of triangular elements on the surfaces and tetrahedrons in the volume. All surface meshes as well as volume meshes are generated using the robust Octree method with patch independency. The Euler-mesh is used for the inviscid Euler calculations. These are done in order to check the behavior of the surface mesh (e.g. no unexpected pressure spikes in unexpected areas), which later will be used to generate the RANS-mesh, and also to give a hint of where one may expect pressure gradients, where (if possible) the mesh needs further refinement and where the end results may land. Initially the grid was much coarser than is presented in Figure 12 and was refined after having it run through an Euler calculation. Table 4 shows a grid comparison between the different Euler grids created.

<table>
<thead>
<tr>
<th>Mesh nr.</th>
<th>Farfield radius</th>
<th>Mounting position, ζ</th>
<th>Total elements [x10^6]</th>
<th>Number of nodes [x10^6]</th>
</tr>
</thead>
<tbody>
<tr>
<td>E1</td>
<td>5L</td>
<td>-</td>
<td>~3.3</td>
<td>~0.58</td>
</tr>
<tr>
<td>E2</td>
<td>-</td>
<td>20°</td>
<td>~18.8</td>
<td>~3.23</td>
</tr>
<tr>
<td>E3</td>
<td>8L</td>
<td>-</td>
<td>~3.3</td>
<td>~0.58</td>
</tr>
<tr>
<td>E4/E5/E6</td>
<td>-</td>
<td>10°/20°/30°</td>
<td>~18.5</td>
<td>~3.17</td>
</tr>
</tbody>
</table>

4.2.5. RANS mesh

Using the surface mesh produced for the Euler-mesh, the RANS-mesh is created by extruding prismatic and quad elements from the surface elements as shown in Figure 16. This procedure is done in order to ensure that no information is lost regarding the boundary layer due to poor resolution [12]. There are several ways to go by when extruding prisms in ICEM. The typical approach is to specify the growth law and at least three of the following parameters; the initial height of the first layer, the total amount of layers present, the prism growth ratio and the total height of the prismatic layers.

The growth law determines the height of each particular layer given the previously mentioned parameters. Three different laws are available in ICEM; the linear, exponential and WB-exponential growth law (this is the exponential growth law as defined in ANSYS Workbench [13]) and Figure 15 shows how the prism layers are extruded with the different growth laws.

Figure 15. The different available growth laws for prism meshing [13].
The initial height is directly connected to the dimensionless viscous wall distance, \( y^+ \), which is defined as

\[
y^+ = \frac{yu^*_z}{\nu}
\]  

(9)

where \( y \) is the normal distance to the nearest wall, \( \nu \) is the kinematic viscosity and \( u^*_z \) is the wall friction velocity which in turn is defined as

\[
u^*_z = \sqrt{\frac{\tau_w}{\rho}} = U_\infty \sqrt{\frac{1}{2} C_f}
\]  

(10)

where \( \tau_w \) is the shear stress at the wall, \( \rho \) is the fluid density, \( U_\infty \) is the free-stream velocity, \( C_f \) is the skin friction coefficient which is calculated, using turbulent boundary layer theory for a flat plate at zero incidence [18], through

\[
\frac{C_f}{2} \approx \left[ \frac{\kappa}{\ln \operatorname{Re}_x} G \ln \operatorname{Re}_x \right]^2
\]  

(11)

where the quantity \( \kappa = 0.41 \) is the Karman constant, \( G \ln \operatorname{Re}_x \) is a function which is weakly dependant on \( \ln \operatorname{Re}_x \) and is usually set to \( G = 1.5 \) and \( \operatorname{Re}_x \) is the Reynolds number for a characteristic length \( x \). The turbulent Reynolds number is calculated using the full length of the ship, i.e. \( x = L_{OA} \), through

\[
\operatorname{Re}_x = \frac{\rho U_\infty x}{\mu}
\]  

(12)

where \( \rho \) is the fluid density and \( \mu \) is the fluids dynamic viscosity.

A simpler way of calculating the viscous grid spacing is used through an online Java script [19] which utilizes the same theory mentioned above.

At least 5-10 grid points should be within \( y^+=20 \) and the initial layer should be within \( y^+=1 \) [9] in order to properly catch all details in the flow inside the boundary layer. The number of prismatic layers is defined by the user and is typically between 30-50 layers within the boundary layer. One method is to extrude all desired layers at once, however since the element sizes vary a lot on the ship this proved to have undesirable results in the prismatic layers, also extruding all layers at once can require much computational time when dealing with a large amount of elements. A more robust approach was used where the initial height and total height were left floating (i.e. varying along the surface of the ship) and only five thick layers were initially extruded (see Figure 16) using the exponential growth law and in a later step each layer is split into the desired amount of layers and the grid points redistributed to better satisfy the desired initial height and growth ratio. Furthermore prismatic elements are capped off by pyramids in areas where the quality of the element would otherwise be to poor.
All viscous grids created are listed and compared in Table 5. Common for all viscous meshes is that they all have the larger farfield radius, i.e. 8.5 $L_{OA}$, and that they have been created using patch independent meshing.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>R1</td>
<td>-</td>
<td>Robust Octree</td>
<td>-</td>
<td>50</td>
<td>~13.0</td>
<td>~5.8</td>
</tr>
<tr>
<td>R2</td>
<td>20°</td>
<td>-</td>
<td>-</td>
<td>50</td>
<td>~33.9</td>
<td>~12.1</td>
</tr>
<tr>
<td>R3/R4/R5</td>
<td>10°/20°/30°</td>
<td>-</td>
<td>-</td>
<td>30</td>
<td>~22.2</td>
<td>~7.7</td>
</tr>
<tr>
<td>R6</td>
<td>20°</td>
<td>-</td>
<td>-</td>
<td>30</td>
<td>~22.0</td>
<td>~7.0</td>
</tr>
<tr>
<td>R7</td>
<td>-</td>
<td>Advancing front</td>
<td>Density field</td>
<td>50</td>
<td>~15.6</td>
<td>~6.4</td>
</tr>
<tr>
<td>R8</td>
<td>20°</td>
<td>-</td>
<td>-</td>
<td>50</td>
<td>~41.5</td>
<td>~17.1</td>
</tr>
</tbody>
</table>

Note that both mesh R2 and R8 are too large to be computed on the supercomputer and thusly only one apparent wind angle can be calculated at a time. Therefore only a selected number of measure points are calculated for these meshes.

4.2.6. Mesh quality
In order to avoid any numerical problems further on in the CFD process, where the meshes will be run through the flow solver, it is most desirable to have mesh with high quality. This decreases the chances for divergences due to poor mesh quality. Depending which type of element is considered, ICEM calculates the quality of the element differently. A listing of the different methods for the elements used in this project is presented in Table 6.
Table 6. Quality calculation methods for the different element types [13].

<table>
<thead>
<tr>
<th>Element type</th>
<th>Quality calculation method</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Triangle</td>
<td>The minimum ratio of height to base length of each side of the element</td>
<td>Normalized to 1</td>
</tr>
<tr>
<td>Tetrahedron</td>
<td>Aspect Ratio of element</td>
<td>Value ranges from 0 to 1. 0 indicates no volume in the element and 1 that the element is perfectly regular</td>
</tr>
<tr>
<td>Quadrilateral</td>
<td>Determinant of element</td>
<td>Value ranges from -1 to 1 where negative values indicates inverted element, 0 indicates element degenerates and 1 a perfectly regular element</td>
</tr>
<tr>
<td>Pyramid</td>
<td>Minimum of the determinant and warpage of the element.</td>
<td>Warpage is normalized to a factor between 0 and 1 where 0 indicates a warpage of 90° and 1 a warpage of 0°</td>
</tr>
<tr>
<td>Prismatic</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The determinant is calculated as the ratio of the smallest and largest determinant of the Jacobian Matrix where the determinant is computed at each node of the element.

The aspect ratio is calculated as the ratio between the radius of an inscribed sphere and a circumscribed sphere for each element, as shown in Figure 17.

![Figure 17. Definition of the aspect ratio for tri and tetra elements [13].](image)

The warpage is determined for the quadrilateral faces of a prismatic element by calculating the maximum distortion of said faces.

Typically the quality of the mesh is satisfactory if the quality value is higher than 0.001. However this is no guarantee that things will run unhindered during calculations. The worst element qualities for every mesh created are listed in Table 7. When several different elements are used in the same mesh, the lowest quality is presented.

Table 7. Worst element qualities for the different meshes.

<table>
<thead>
<tr>
<th>Mesh nr.</th>
<th>Worst element quality</th>
<th>Element type present</th>
</tr>
</thead>
<tbody>
<tr>
<td>E1</td>
<td>~0.17</td>
<td>Triangles and tetrahedrons</td>
</tr>
<tr>
<td>E2</td>
<td>~0.17</td>
<td></td>
</tr>
<tr>
<td>E3</td>
<td>~0.12</td>
<td></td>
</tr>
<tr>
<td>E4/E5/E6</td>
<td>~0.11</td>
<td></td>
</tr>
<tr>
<td>R1</td>
<td>~0.0012</td>
<td>Triangles, tetrahedrons, quadrilaterals, prismatic and pyramids</td>
</tr>
<tr>
<td>R2</td>
<td>~0.0022</td>
<td></td>
</tr>
<tr>
<td>R3/R4/R5</td>
<td><del>0.0013/</del>/0.0041/~0.0055</td>
<td></td>
</tr>
<tr>
<td>R6</td>
<td>~0.0018</td>
<td></td>
</tr>
<tr>
<td>R7</td>
<td>~0.0011</td>
<td></td>
</tr>
<tr>
<td>R8</td>
<td>~0.0023</td>
<td></td>
</tr>
</tbody>
</table>
If the mesh quality is unsatisfactory (which often is the case, one rarely creates a good quality mesh from the start) there are several tools available in ICEM in order to improve the quality. One such tool is the smoothing operator which can move and/or merge grid points, swap edges and also delete entire elements if needed. It is also possible to choose if all or only some of the different element types are to be smoothed. One can also manually choose specific elements and alter their appearance (e.g. splitting them or moving grid points). If the mesh still is unsatisfactory one should consider changing the mesh generating method or altering the geometry (if possible) to make meshing easier in some areas.

When satisfied with the mesh distribution and quality the mesh is converted into a format compatible with the flow solver.

4.3. CFD SOLVER PROCESS

As previously mentioned the flow solver used for this thesis is the Edge-code. In order to simplify the process of defining different parameters and get a better overview of the case under investigation the code comes with a GUI (Graphical User Interface) called Xedge [14].

Generally there are five steps that need to be taken when making a computation [6];

1. Set up an input file with relevant parameters for the desired computation.
2. Set proper boundary conditions.
3. Running the preprocessor.
4. Running the flow solver.
5. Post process results.

The last step is done both in Edge and ParaView.

4.3.1. The input file

The general approach is to start by loading a default input file. This file contains all the default values for every parameter which controls the flow solver, preprocessor and other programs within the Edge-code. This is followed by defining case specific parameters in the external flow wizard (as the name suggests, the wizard is primarily used for steady state, external flow calculations). The wizard is divided in two parts where in the first part free stream values of primitive variables (such as the free stream velocities and pressure) are related to derived variables, e.g. Mach number, Reynolds number over length (Re/m) and angles of attack etc. The free stream coordinates and angles of attack are defined in Figure 18. It is also possible to define in the first part of the wizard whether or not the computations should be 2-D or 3-D.

The components $u$, $v$, and $w$ (the parameter names in Edge for these are UFREE, VFREE and WFREE) are the free stream velocity components in the $x$, $y$- and $z$-directions of the magnitude of the free stream velocity, $U_\infty$. These are related through the angles of attack through [14]
\begin{align*}
u_e &= U_\infty \cos \beta \cos \alpha \\
v_e &= -U_\infty \sin \beta \\
w_e &= U_\infty \cos \beta \sin \alpha
\end{align*}

and

\begin{equation}
U_\infty = \sqrt{u_e^2 + v_e^2 + w_e^2}.
\end{equation}

Throughout all calculations in this thesis the vertical angle of attack, \(\alpha\), is kept at 0° and only the horizontal angle of attack, \(\beta\), is allowed to vary every 5° between 0° and -45° (one input file is created for every angle and for every mesh). The magnitude of the free stream velocity, \(U_\infty\), is kept at 20 knots.

The second part of the wizard gives access to crucial parameters for external flow calculations such as whether the calculations are to be viscous or inviscid (INSEUL in Edge). For viscous computations one can further define which turbulence model to use along with other parameters connected to turbulent flow calculations.

A turbulence model is a computational procedure that closes the system of mean flow equations so that a wide variety of flow problems can be solved [11]. The model needs to have a wide applicability, be accurate, simple and economical to run in order to be useful in general CFD purposes. Since turbulence is very difficult to model, there exist several different models using different approaches to choose from. The one used for these calculations is the default RANS-based turbulence model for Edge which is a two equation, \(k-\omega\) model which uses two additional transport equations that represent the turbulent properties of the flow [9]. In this case one of the equations represents the turbulent kinetic energy, \(k\), and the other one the specific dissipation, \(\omega\). It also utilizes an EARSM approximation (Explicit Algebraic Reynolds Stress Model) in which the Reynolds stress tensor is explicitly expressed in terms of velocity gradients and turbulence scales. A more detailed description of this model and several other models available in Edge (e.g. models for detached eddy simulations, DES, and large eddy simulations, LES) and their respective derivations can be seen in [15].

Besides the turbulence model and properties one can further specify in the second part of the wizard the number of multigrids and the amount of full multigrid cycles, initial condition (if one is available, if not this is created in the preprocessor), number of iterations and the logarithmic convergence criterion.

The Full Multigrid method (turned on or off with the parameter IFULMG in Edge, default is turned on, i.e. IFULMG = 1) uses a number of coarser grids (defined in Edge by the parameter NGRID, default is 4 grids which is used mostly throughout the thesis) in order to drive the steady state solution on the finest grid faster to steady-state convergence [11]. Calculations are started on the coarsest grid where the approximate solution is interpolated to the next finer grids so that a good starting solution is obtained on the finest grid with low numerical effort. The maximum number of iterations on the coarser grids is determined by the parameter NFMGCY, here the default value of 500 iterations is used for all steady-state calculations. IFULMG needs to be turned off (i.e. IFULMG = 0) for time-accurate calculations.

The number of iterations done is determined by the parameter ITMAX (default setting is 1000 iterations, however for most cases in this thesis this number is changed to 4000) and should it turn out that the defined number is too low or that the case is unsteady in nature and one wants to switch to time-accurate calculations, it is possible to use the solution from the previous calculation as an initial solution for the new calculation by turning on the parameter INPRES (i.e. =1 or 2 depending on the calculation).

A calculation could be stopped before computing every iteration if the residuals have reduced with an order of magnitude equal to that defined in the parameter RESRED. The default is that computations be stopped when the residuals have reduced with an order of magnitude -5.5, this is however changed for the calculations in this thesis to -10 in order to ensure that every iteration is computed for higher accuracy.
The wizard only provides control over a fraction of the amount of parameters available (more than 100 different parameters), however since these are usually case specific these are, for simple calculations, the only parameters that are changed from the default values. Regarding the numerical parameters, the default settings usually provides with good convergence rates and stable solutions.

Some parameters not considered in the wizard are however changed to fit the calculations for the ship. These are defined in Table 8.

<table>
<thead>
<tr>
<th>Edge parameter</th>
<th>Description</th>
<th>Value (default value)</th>
<th>Misc.</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFIMSH</td>
<td>The input mesh file, must be in the format “.bmsh”.</td>
<td>Name-of-file.bmsh (Edge.bmsh)</td>
<td>Important that a mesh file with this exact name exists in the current working directory</td>
</tr>
<tr>
<td>CFIBOC</td>
<td>The name of the generated boundary condition file.</td>
<td>Name-of-file.aboc (Edge.aboc)</td>
<td>Name of the boundary condition file later generated by Edge. Changed from default for PDC-purposes in order to distinguish which BC file belongs to which mesh.</td>
</tr>
<tr>
<td>INSEUL</td>
<td>Determines the governing equations.</td>
<td>0 = Euler (default) 1 = N-S</td>
<td>Can be chosen in the Wizard</td>
</tr>
<tr>
<td>IFULNS</td>
<td>Thin layer approximation or full N-S.</td>
<td>0 = Thin layer 1 = Full N-S</td>
<td>Only available for viscous calculations.</td>
</tr>
<tr>
<td>ITURB</td>
<td>Turbulence calculation type</td>
<td>0 = Laminar 2 = RANS models</td>
<td>Only available for viscous calculations. Other available options: 3 = DES models 4 = LES models</td>
</tr>
<tr>
<td>NPLAYER</td>
<td>Number of prismatic layers</td>
<td>30 or 50 (1)</td>
<td>Only for viscous calculations</td>
</tr>
<tr>
<td>NPART</td>
<td>Number of partitions for parallel computing</td>
<td>1-8 (1)</td>
<td>Varies depending on which computer, or rather on how many processors, the calculations are run</td>
</tr>
<tr>
<td>IPOST</td>
<td>Post processing options</td>
<td>11 (0)</td>
<td>Generates an additional solution file containing information regarding ( y^+ ), ( C_f ) and other interesting data. For viscous calculations only</td>
</tr>
<tr>
<td>IDCLP</td>
<td>Direction for lift force coefficient, ( C_l )</td>
<td>( x, y, \xi = 0, 1, 0 ) ((0, 0, 1))</td>
<td>For plotting purposes, since Edge only plots the lift and drag force coefficients these parameters are switched so that the actual side force in this case is plotted</td>
</tr>
<tr>
<td>IDCCP</td>
<td>Direction for side force coefficient, ( C_C )</td>
<td>( x, y, \xi = 0, 0, 1 ) ((0, 1, 0))</td>
<td></td>
</tr>
<tr>
<td>SREF</td>
<td>Reference to normalize force and moment coefficients</td>
<td>1132 (1.0)</td>
<td>The projected frontal area of the ship is used. Unit [m²]</td>
</tr>
</tbody>
</table>

As stated above, the parameter IFULNS determines if a thin layer approximation or a fully viscous approach should be applied. The thin layer approximation assumes [21] that for high enough Reynolds numbers the shear layer is very slender and thusly the grid describing it is packed in the surface normal direction whilst being relatively coarse streamwise and therefore only apply viscous terms in the normal derivatives and dropping these terms in the streamwise direction. The full viscous approach is however chosen for this project due to the relatively low speeds (i.e. low Reynolds numbers) being calculated.

The parameters presented in Table 8 are basically the same for all calculations.
For low-speed calculations it is recommended to apply preconditioning (controlled by the parameter IPREPA in Edge, default setting is off, i.e. IPREPA = 0) which helps the calculations to reach convergence faster. However using preconditioning for the cases in this project led to divergence or the solution blowing up and therefore was not applied for further calculations.

For time-accurate calculations one must switch on the parameter ITIMAQ, i.e. setting it so that ITIMAQ = 1 for implicit time stepping or = 2 for explicit time stepping. Time-accurate calculations were only done for the inviscid calculations, where implicit time stepping was used, since it would have been too time-consuming to apply time-accurate calculations to the viscous grids (i.e if there had been a need to do so since very few of the viscous calculations indicated that time-accurate calculations were needed). Implicit time-accurate calculations are computed through a number of global iterations (ITMAX) where each iteration undergoes a set of inner iterations before moving on to the next global iteration. The amount of inner iterations computed are controlled by three parameters; RESTAQ, which controls the convergence criterion on the residuals of each inner iteration (see Figure 19), ITMNAQ and ITMXAQ, which are the minimum and maximum (respectively) amount of inner iterations to be performed. The amount of inner iterations should be large enough to ensure sufficient convergence of the flow variables at each physical time step before moving on to the next [6]. There is no way of telling beforehand how many inner iterations that are needed. The default ITMXAQ amount is 100 iterations and the default ITMNAQ is 3 both of which have remained unaltered whenever time-accurate calculations have been applied. The ITMNAQ value ensures that at least this many iterations will be performed and the ITMXAQ value forces the calculations to move on to the next global iteration when the amount of inner iterations has reached said number.

![Figure 19. Principal description of the two convergence criteria, RESRED and RESTAQ.](image)

Another parameter that is crucial for time-accurate calculations and usually very case specific is the parameter DELTAT which controls the size of the global time step. If the time step is too small, then one might be calculating sound waves, on the other hand if the time steps are too large then the solution might be equivalent to the steady-state solution. The physical time scale usually gives a hint of how large the time step should be [6]. An estimation of the time step is calculated through

\[
\text{DELTAT} \approx \frac{L}{nU_\infty}
\]

where \(L\) is the characteristic length of the object under investigation (in this case, \(L = L_{\text{ref}}\)), \(n\) is the desired amount of time steps and \(U_\infty\) is the free stream velocity. There should at least be 30-60 [6] time steps, however when applied for this thesis 100 time steps are used leading to a DELTAT value of \(~0.22\) sec (default setting for DELTAT is 0.005 sec).
After finishing defining the parameters for the flow calculations, the boundary condition needs to be set.

### 4.3.2. Boundary conditions

The amount and type of boundary conditions (BC) varies from case to case depending on how the physical geometry is built. Regardless of which BC is chosen there is usually no need to further edit the BC since the default usually are sufficient for all specifications. Four different classes of BCs exist in Edge; connectivity, wall, symmetry and external BCs, each class having a set of different BC types. Connectivity BCs have however not been used in this project and will not be commented upon here, see [6] for information about connectivity BCs.

Three different types of wall BCs exist in Edge; weak Euler, weak adiabatic and the weak isothermal BCs where the Euler condition is only used for inviscid flow calculations. All variables in the weak Euler condition are unknown and specified conditions are implied through the flux [6]. The adiabatic and isothermal conditions are both intended for viscous calculations where the former (and most commonly used for viscous flow and also used in this thesis) assumes zero normal derivative of the temperature whilst the latter assumes a constant wall temperature. Both use strong conditions for the velocities, i.e. the velocities are specified (usually through no-slip conditions) and are therefore no longer unknowns. The no-slip condition implies that the velocity at the wall is zero. The viscous BCs further assume a well resolved grid in which $y^+ \approx 1$ in the first layer of nodes outside the wall. If this is not the case then the velocity at the wall may likely be other than zero.

The symmetry BC is the same as a weak Euler condition and is only defined in a separate class in order to separate it from a wall boundary. This is chosen for the symmetry plane which represents the “water surface”. Since, for this investigation, a simplification was made in which the boundary layer generated on this surface will not be considered, this BC should be sufficient.

There exist several different types of external BCs in Edge. Generally these too are of a weak formulation, i.e. variables on the boundary are unknown and specified conditions are implied through the flux. The weak characteristic BC (also denoted as a Farfield BC) can be used for all external boundaries. It can handle both sub-/supersonic in-/outflow boundaries but is mainly intended for external aerodynamic calculations. The characteristic BCs are based on one-dimensional analysis of the Euler equations in direction normal to the wall and are either specified or extrapolated depending on the signs of the eigenvalues [6]. Other external BCs describe specific boundaries for internal flow calculations, sub-/supersonic in-/outflow, propulsion in-/outlets, mass flow in-/outlets etc. Usually constant free stream values are used at the boundaries, however local variations can also be applied which makes it possible for this BC to adapt if necessary to any variation that may occur in the wake.

Table 9 lists the different BC classes and types used in this thesis along with a brief explanation of their function.

<table>
<thead>
<tr>
<th>BC type</th>
<th>Edge notation</th>
<th>Function</th>
<th>Applied to</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wall</td>
<td>Week Euler</td>
<td>Zero normal velocity. For inviscid calculations only.</td>
<td>Hull, Superstructures and wing</td>
</tr>
<tr>
<td></td>
<td>Weak Adiabatic</td>
<td>No-slip condition. Assumes zero normal derivative of the temperature.</td>
<td></td>
</tr>
<tr>
<td>Symmetry</td>
<td>Week Euler</td>
<td>Same as Euler wall.</td>
<td>Symmetry plane</td>
</tr>
<tr>
<td>External</td>
<td>Weak characteristic</td>
<td>Characteristic boundary conditions.</td>
<td>Farfield</td>
</tr>
</tbody>
</table>

When the BCS are properly defined the preprocessor is initiated.

### 4.3.3. Preprocessor

Initially the preprocessor further discretizes the elements in the grid to form control volumes around each vertex. These control volumes are then fused to create coarser grids for the multigrid process. This step is only done for steady state calculations as the multigrid method is not applied to time-accurate calculations.
Next step is to create partitions of the grid, if the parameter NPART is greater than 1, in order to allow for parallel computing. Each partition is read by a separate processor. Furthermore the preprocessor reorders every node and edge in order to improve performance of the solver, checks the orientation of every element in every boundary and (if applied) enables vector colorization to improve performance on vector computers.

For viscous calculations the preprocessor also calculates wall distances and laminar regions.

Lastly it generates a file with an output name determined by the parameter CFIEDG (default is Edge.bedg) and for partitioned files the generated files will have an extension _p1, ..._p"NPART".

When the preprocessor has finished the flow calculations can take place.

4.3.4. Flow solver

When initiated the flow solver is left on its own and will be terminated if:

- The amount of iterations computed has reached the maximum amount defined in parameter ITMAX. This is the most common reason for termination.
- The convergence criterion RESRED has been reached (however, whenever this happened in this project the solution had diverged).
- The solution has diverged (Edge reports this as the solution blowing up).

4.3.5. Post-processing

When the calculations are completed the results are mainly analyzed through three different steps; by plotting the convergence of the different force coefficients and the residuals, analyzing the general flow picture (e.g. the pressure distribution on the ship and in the volume) and lastly by checking the $y^+$ value in the first grid points. The last step is only eligible for viscous calculations as there are no viscous boundary layers in inviscid calculations. Plotting force convergence and residuals is easily done through Xedge whilst the flow picture and the $y^+$ value can be analyzed using ParaView. The results should also be compared with independent data such as results from wind-tunnel testing if available.

The ideal case when looking at the residuals is that they have reduced at least 5 decades [9] and that the fluctuations in the force coefficients have leveled out and varies less than 10 % over the final 1000 iterations. Figure 20 shows a comparison between cases with good and bad force convergence.

![Figure 20. Example of a plot showing a) better and b) worse convergence for the residuals and the different force coefficients plotted against the number of iterations done. Residuals of the primitive and (in this case) turbulent variables are plotted on the left. The side force coefficient (here denoted as $C_l$) is plotted on the top right. The force coefficient in the ship direction (here denoted as $C_d$) is plotted in the middle to the right and lastly the moment coefficient (here denoted as $C_m$) is plotted at the bottom right.](image)
Since only one reference area can be used at a time in Edge only the values responding to the force coefficient in the ship's direction ($C_d$ in Figure 20) are the actual values. For the actual side force ($C_l$ in Figure 20) these results are later normalized using the projected side area of the ship. The moment coefficient ($C_m$ in Figure 20) is not considered at all in this thesis.

The general flow picture should be smooth with no abrupt or unexpected changes in the gradient. However increasing complexity of the case makes it harder for the solver to converge properly which was the case for the calculations on the ship. This is further discussed in the following chapter as the solutions have been far from satisfactory.

5. THE ROAD TO CONVERGENCE AND RELIABLE RESULTS

This chapter will discuss the problems that occurred when running calculations on the ship and how (or if) these were overcome. For starters it may be in place to comment on the denotations for the different grids in the following plots and also what the plots represent. Firstly the plots (e.g. as in Figure 22) show the calculated total force coefficient in the ship's direction and is denoted $C_X$ which determines the arial resistance. The value of which is the mean value presented with the maximum deviation from this mean value determined through the final 1000 iterations. The deviation is presented as bars at every measuring point. Furthermore the value itself is of no interest here other than what it says about the force convergence and only the behavior of the solution will be commented upon. The actual value is only interesting when discussing the final results in the following chapter (Chapter 6).

The different grids are denoted in the plots (e.g. as in Figure 22) using the following structure:

<table>
<thead>
<tr>
<th>Size of FF-radius.</th>
<th>Viscous or inviscid calculation.</th>
<th>Whether or not the ship is equipped with a wing and if so its position.</th>
<th>Misc. information, e.g. coarsened/refined mesh etc.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Options: 5L or 8L</td>
<td>Options: E or R</td>
<td>Options: nw (no wing), 10°, 20° or 30°</td>
<td>Options:</td>
</tr>
</tbody>
</table>

If nothing is else is written in the misc. field concerning the mesh then the surface mesh used is that presented in Figure 12.

5.1. SOLUTION DEPENDENCY

The goal of any CFD calculation is to produce results independent of a number of factors. The most important of which are listed in Table 10 along with the strategy employed to avoid any dependency of them.

<table>
<thead>
<tr>
<th>Factor</th>
<th>Reason</th>
<th>Strategy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Farfield and BC</td>
<td>The size of the computational domain should be large enough as not to influence the end results and the BCs should be sufficient to describe the boundaries.</td>
<td>Increase the domain size to see the effects. If unexpected variations occur consider changing BC on symmetry plane.</td>
</tr>
<tr>
<td>Numerical method and turbulence model</td>
<td>The solution should be independent on which method is used to compute and which dissipative operators are used.</td>
<td>Alter some dissipative parameters to investigate the effects. Also try different turbulence models.</td>
</tr>
<tr>
<td>Primitive variables (e.g. free stream velocity)</td>
<td>Since the calculations deliver what should be dimensionless coefficients, the solution should be independent of e.g. which speed is used to calculate them.</td>
<td>Scale the primitive variables whilst maintaining the Reynolds number.</td>
</tr>
<tr>
<td>Grid size</td>
<td>The mesh should be fine enough as not to produce grid dependent results.</td>
<td>Further refine the mesh.</td>
</tr>
</tbody>
</table>

Table 10. Factors of which the end results should be independent.
5.2. THE COMPUTATIONAL DOMAIN

The first study was to see whether or not the size of the computational domain had any effect on the solution. Preferably the solution should move towards an asymptotic value, as illustrated in Figure 21, when increasing the domain radius until the solution no longer is affected.

![Figure 21. Illustration of how the solution could behave when increasing the domain radius.](image)

The investigation was done by running Euler calculations on the different sizes of the computational domain to see if there were any significant differences in the solution. The resulting force coefficient in the ships direction, $C_X$, is presented in Figure 22.

![Figure 22. Inviscid calculations for different radii on the computational domain, $C_X$.](image)

Note that only three points were calculated for the smaller radius. The comparison above shows a difference in the results. This leads to the decision to only focus the larger domain size to increase the reliability (as well as the accuracy) in the results for the following viscous calculations.

Since the difference was so small between the solution for 5L and 8L no further work was put into checking the dependency of the radius. However towards the end of the project a grid was created anyway in which the radius was increased further (up to 10L). After having run inviscid calculations on this new grid the solution showed a significant deviation from the results for 5L and 8L grids without wing. Therefore the following solutions should be considered with care since the BC chosen for the “water surface” (i.e. the symmetry BC) may not be optimal for this case and must be further investigated.
5.3. **CONVERGENCE THROUGH COMPARISONS AND TESTING**

When computing the initial viscous calculations it quickly became evident that there was something lacking in the general flow picture when analyzing this in ParaView, as seen in Figure 23 which depicts the pressure distribution, $C_p$, on the ship and in the volume through a horizontal cut plane at 19 m above “sea level”.

![Figure 23. The distribution of the pressure coefficient, $C_p$, for case 8L-R-nw with an apparent wind direction, $\beta$, of 20°.](image)

Note that the color scaling is set from -2 to 1. This is just to easier observe the pressure distribution and is not the actual maximum/minimum pressure present. Again, the actual value is not under consideration, however the behavior is. The distribution is somewhat smooth on the surface but when observing the distribution in the volume some relatively large jumps in pressure are visible, also the distribution is somewhat jagged. This is expected along the sharp edges of the ship since it is very difficult to obtain good solutions in these areas due to high accelerations and other dynamic happenings that occur there.

The reason for this poor distribution is thought to be one (or all) of three things (investigated in the order presented):

1. A numerical phenomenon causing the solution to not dissipate enough making the solution grid dependant.
2. The speed of the object under investigation is too low causing the flow to become incompressible which is difficult for Edge to handle properly as it was intended for higher speeds.
3. The grid is too coarse and needs to be further refined.

Note that had it not been that the grid size already was at the limit of what was possible resourcefully, the natural first step would have been to further refine the mesh. Since this was the least favorable option at the time this thesis was conducted the order of investigation became as presented above.

Also the inviscid solutions had the same poor $C_p$ distribution but no effort was put in to acquiring better solutions since, in the end, only the viscous results are of interest.

5.3.1. **Numerical alterations**

Theoretically, by altering some parameters in the input file one can obtain better convergence in cases that initially produced poor (or in some cases diverged) solutions with the default settings. The alterations made on the calculations for the ship are all based on recommendations from [6]. Furthermore it should be mentioned that no cases were run in which several parameters were altered at the same time. All alterations are calculated separately in order to observe whether or not this specific parameter change has any effect. Also, the general objective of altering the numerical parameters is to increase the numerical dissipation.
One of the first tests is to see if a reduction of the multigrid levels (i.e. lowering the parameter NGRID) has an overall influence on the solution. The solution should preferably not have a grid level dependency, however too many grids could have the effect that the coarsest grid produces a diffusive solution, i.e. the solution is dissipated too much initially which in turn could create problems further on in the calculations. The default amount of grid levels is 4, which is sufficient for most cases. When testing to see if the solution was multigrid dependant the parameter NGRID was reduced to 3 levels.

Next, a reduction of the inviscid CFL (Courant-Friedrichs-Levy) number (denoted CFL in Edge and has a default value of 1.25) was tested to observe if this could stabilize or improve the convergence. The CFL number (which is equivalent to the Courant number) determines the amount of cells a fluid-element travels in one time step [11]. By decreasing this number one thusly increases the overall computational time as it takes more time and more iterations to calculate every cell, at the same time however one could increase the accuracy of the results. When testing if the CFL number had any influence on the general flow picture the CFL number was reduced to 1.

The parameters VIS2 and VIS4 are artificial viscosity terms adding second and fourth order dissipation terms respectively and are used in central schemes only. By increasing the value of these parameters one can increase the numerical dissipation. Default setting is 0.5 for VIS2 and 0.02 for VIS4, which were increased to 1.0 and 0.04 respectively during testing.

Another parameter tested was the parameter TURFIX which controls the entropy fix for 2-equation turbulence models. A negative value is recommended for low Mach numbers (default is -1.1) in order to decrease the numerical dissipation. Since the opposite is desired the value was changed to 1.0.

It is also of interest to see if the solution is dependant of the turbulence model used. If the solution varies greatly when altering the turbulence model, the solution is deemed unreliable. The only other turbulence model tested is the one-equation Spalart-Allmaras model. This model is one of the most widely used models and utilizes only one turbulent transport equation for a quantity $\tilde{\nu}$ equivalent to the eddy viscosity [15].

The parameters mentioned so far had no influence whatsoever regarding the pressure distribution. They did however affect the behavior of the residual and force convergence, however without generating any dramatic differences in the end results. Naturally there are several more ways to alter the solutions numerically but since none of the above mentioned parameters had any influence it was determined that the distribution could not be improved numerically.

5.3.2. Scaling the speed and density

Since Edge was initially intended for high speed aircrafts with high compressibility, the relatively low speed of the ship was determined to be outside of the operable field, meaning that the flow is incompressible which is difficult for Edge to handle properly. By increasing the speed of the vessel it is possible to gain better results. It should not matter what velocity is used in the calculations since, in the end, the calculations yields dimensionless coefficients that are independent of the velocity. However simply increasing the free stream velocity would yield unrealistic flow conditions and therefore it is desirable to increase the speed whilst maintaining the Reynolds number, Re. Besides the free stream velocity, $U_\infty$, the Reynolds number is dependent on three other quantities; the fluid density, $\rho$, a characteristic length, $L$, and the dynamic viscosity, $\mu$ through

$$\text{Re}_L = \frac{\rho U}{\mu} \frac{L}{\rho}. \quad (16)$$

The common method utilized during wind tunnel testing is to downscale the geometry (for obvious physical reasons) of the object under investigation, thusly decreasing $L$. This is possible to do in Edge through a program called Scale. However another way to maintain the Reynolds number in Edge is to decrease the free stream pressure, $P_\infty$, thusly decreasing the fluid density. The speed and density for the ship calculations were scaled by a factor of ~6.5, see Table 11.
Table 11. The different speed and density values when scaling.

<table>
<thead>
<tr>
<th>Mach [-]</th>
<th>U_∞ [m/s]</th>
<th>P_∞ [Pa]</th>
<th>ρ [kg/m^3]</th>
<th>μ [kg/(m∙s)]</th>
<th>Re [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unscaled</td>
<td>0.031</td>
<td>10.76</td>
<td>101325.0</td>
<td>1.18</td>
<td>1.789×10^5</td>
</tr>
<tr>
<td>Scaled</td>
<td>0.2</td>
<td>69.44</td>
<td>15705.4</td>
<td>0.182</td>
<td>-</td>
</tr>
</tbody>
</table>

Note that the new speed and density used is physically equivalent to the ship flying at a high altitude at high speed, as illustrated in Figure 24.

Figure 24. Illustration of the new physical environment for the ship after scaling [20].

The affect on the pressure distribution due to scaling is presented in Figure 25.

Figure 25. The effects on the pressure distribution when scaling to Mach = 0.2.

A significant improvement is observed, not only in the volume but also on the surface of the ship itself. However there are still some areas in the volume that show the same behavior (although not as obvious) as in Figure 23. Furthermore the scaling showed a significant impact on the force coefficient C_X, as seen in Figure 26. The total resistance solution is lowered by ~20% when scaling the velocity. In order to determine the reliability of this solution further calculations were done in which the variables were scaled to Mach = 0.1 and 0.25. The results are compared in Figure 27.
Although the unscaled solution has a slightly smaller deviation from the mean value (illustrated by the errorbars in Figure 26), the unrealistic pressure distribution presented by them increases the trustworthiness of the scaled solution.

When lowering the speed a difference is observed. This could be because the speed is at the lower border of what is operable in Edge. When increasing the speed no significant change is observed although the calculations had however a harder time converging at high apparent wind angles ($\beta \geq 40^\circ$). Furthermore when comparing the effect that the wing has on the resistance by calculating the difference ($\Delta C_X$) between the solution with and without the wing regarding the solution between the unscaled and scaled calculations, the results show the behavior seen in Figure 28. A positive value on $\Delta C_X$ means an increase in resistance and a negative value means the opposite.
The results for the scaled solutions (dashed lines in Figure 28) show a more realistic behavior since intuitively the wing should, at small apparent wind angles, increase the resistance and start to give effect when the apparent wind angle increases. One must however keep in mind that the meshes for the winged cases are severely coarsened, the effect of which is revealed when looking at the $y^+$ distribution, see Figure 29.

![Figure 28](image1)

**Figure 28.** $\Delta C_x$ for the unscaled (solid lines) and scaled (dashed lines) at three different mounting positions; 10° (blue), 20° (red) and 30° (black) from the centerline.

5.3.3. Grid dependency

Calculations were also done on a refined mesh in order to investigate the grid dependency. This mesh is generated using the Advancing front method and has a density field surrounding the ship (as described in Section 4.2.3). Not scaling the velocity and density initially yielded a similar pressure distribution as in Figure 23. The affect on the pressure distribution when using the refined mesh and scaling to Mach = 0.2 is seen in Figure 30.

![Figure 30](image2)

**Figure 30.** The effects on the pressure distribution with the refined mesh and scaling to Mach = 0.2

Again an improvement is observed, however not as obvious, both on the surface and in the volume. Also the resulting $C_x$ is affected, as seen in Figure 31.

![Figure 29](image3)

**Figure 29.** The $y^+$ distribution for the coarsened mesh with the wing at 20° from the centerline at an apparent wind angle of 25°.

![Figure 31](image4)
A significant deviation from the previous behavior is observed at apparent wind angles >25°, especially for the solution without wing. Furthermore the flow solver had great difficulty in solving the calculations with wing using the default turbulence model and therefore the Spalart-Allmaras model was used for the refined mesh with wing using only 3 multigrid levels. The reason for this is that a finer mesh is expected to capture more dynamic details in the flow making it harder for the solver to compute. This could in some cases yield an initially steady flow unsteady. The change in turbulence model also had an effect on the solution, as shown in Figure 32. The solutions are therefore determined being grid dependant and, for the refined mesh solution, also dependant on turbulence model.

The Spalart-Allmaras model also had some difficulty in producing a satisfactory converged solution for the case without wing (red solid line in Figure 32) for apparent wind angles >30°, which can be observed through the increasingly large deviations from the mean value at these measuring points. The solution for the case without wing at an apparent wind angle of 45° yielded a ridiculous deviation (~200%) and is therefore not plotted at all.

The next step would be to again further refine the mesh to see if the solution changes any further. Preferably when increasing grid size the behavior of the solution should have an exponential behavior moving towards an asymptotic value (as illustrated in Figure 33) and when the solution seizes to deviate significantly one can be satisfied with the grid size. Whether or not the refined mesh in this case has reached this point is yet to be determined through calculations on a finer grid.
The solution for the refined mesh has however the lowest $y^+$ values as seen in the comparison in Figure 34. The figure shows the $y^+$ value for the first grid level and preferably this should be $\leq 1$ all over the ship. The refined mesh has the largest percentage that satisfies this requirement.

The spots of high $y^+$ values present in the refined mesh solution are areas where the redistribution process when generating prismatic layers has had trouble computing properly. The areas in which $y^+ > 1$ is highly connected to the velocity at the wall. This should preferably be $\sim 0$ all over the ship, however this is not the case for the refined solution with wing as seen in Figure 35.
This too is an indication that the flow solver has not converged successfully for the refined mesh.

6. RESULTS

The results presented in this chapter are mostly from those gained from the scaled calculations on the refined mesh (which had the best $y^+$ and $C_p$ distribution), with and without wing, using the Spalart-Allmaras turbulence model. The results gained from the scaled calculations on the coarsened mesh will also be presented here (even though these solutions had the worst $y^+$) since the largest amount of calculations were made on these grids (only on these grids were different mounting positions and also one other sheet angle than used otherwise calculated). The results for the coarsened winged mesh will be compared with the results for the scaled calculations on the unaltered mesh without wing, since no mesh without wing was created using the coarsened surface mesh.

Furthermore, as seen in Figure 36, apparent wind angles, $\beta$, larger than 30° only occur for wind speeds, $V_T > 5$ m/s. Therefore primary interest regarding $C_X$ and $C_Y$ results will be at angles lower than 30° since 5 m/s is the most common wind speed encountered on a worldwide route [3].
6.1. THE SHIP WITHOUT WING

In Figure 37 and Figure 38 the total force coefficients acting in the ship and side direction are presented and compared to those used in Silvanius thesis which was gained from wind tunnel (WT) testing. The results for the coarsened and refined mesh are presented in separate plots in order to stress the fact that they have been calculated using different turbulence models and should therefore not be compared to each other when looking at results.

![Figure 37](image1)

Figure 37. $C_x$ and $C_y$ for the coarsened mesh (black) compared to the values used in Silvanius thesis (blue).

![Figure 38](image2)

Figure 38. $C_x$ and $C_y$ for the refined mesh (red) compared to the values used in Silvanius thesis (blue).

A large deviation is observed when looking at the $C_x$ results for both grids. An explanation for this could be that different Reynolds numbers were used in the wind tunnel experiments than for these calculations ($7 \cdot 10^5$ for the WT tests and $\sim1.6 \cdot 10^8$ for these calculations) which makes any comparison unfair. The $C_y$ results also deviate from the WT results, yet they show a higher similarity to the behavior of the WT results. The highest $C_x$ value occurs at $\beta=15^\circ$ for both grids.

The $C_p$ distribution for four different apparent wind angles is shown in Figure 39.
The results show that the sharp edge of the bow below the semicircular surface, near the bulb, generates a low pressure area where the flow has separated. For increased $\beta$ angles, this zone is moved along the side of the hull and for $\beta>30^\circ$ the flow is completely separated from the hull. The entire sweep for the refined mesh without wing can be found in Appendix A.

6.1.1. Concerning the bow effect

Regarding the assumption of the bow effect on the flow around the naked model, which was assumed in Silvanius thesis, it is clear that the flow is far from as two dimensional as needed in order for that assumption to be correct. His assumption was based on a 2-D flow simulation which did not account for the 3-D effects that indeed takes place in the flow. This is noticed when looking at selected streamlines in the flow. Figure 40 shows the streamlines for the refined mesh without wing at three different apparent wind angles. The streamlines are generated by defining a horizontal line (shown as black lines in Figure 40) ~5m in front of the ship located ~6 m below weather deck (i.e. half way down the flat vertical surface of the bow) through which the lines should pass. Streamlines for the entire apparent wind angle sweep ($0^\circ$-$45^\circ$) for the refined mesh without wing can be found in Appendix B.
Figure 40. Streamlines for the refined mesh without wing at apparent wind angles of 0° (top left), 15° (top right) and 30° (below).

It is observed that instead of redirecting the flow to the sides of the hull, the flow prefers to go up and over the bow causing a vortex to generate on the top surface of the bow in front of the bridge. This vortex is reduced with increasing apparent wind angles and instead vorticity is instigated at the side of the ship.

6.2. Effects when adding the wing

The change in $C_X$ and $C_Y$ when adding the wing at a mounting position of 20° from the centerline for the refined mesh is presented in Figure 41 in which the resulting coefficients for the ship without wing has been subtracted from the corresponding coefficients for the ship with wing. A negative value on $\Delta C_X$ indicates that the aerial resistance is reduced.

Figure 41. $\Delta C_X$ (left) and $\Delta C_Y$ (right) for the refined mesh with a wing mounted at 20° from the centerline.
The resistance appears to be reduced in the ship direction already at $\beta=10^\circ$, however not very drastically for $\beta<30^\circ$. A dip is however present at $\beta=15^\circ$ which suggests that, for $\beta<30^\circ$ the wing is most effective when attacked from this angle. Also the wing increases the resistance when traveling at headwind. On the other hand the side force is not reduced until $\beta>20^\circ$. The corresponding percentage of the total $C_X$ and $C_Y$ for the ship without wing is presented in Figure 42.

The aerial resistance in the ships direction is reduced $\sim2\%$ at $\beta=15^\circ$ which is equivalent to $\sim0.2\%$ of the ships total resistance, since the wind resistance corresponds to $\sim10\%$ of the total resistance for the ship in question [3]. The resistance in ship direction is however increased $\sim33\%$ at headwind. The wing should therefore be subtracted when traveling at headwind.

The pressure distribution on the hull is also affected by the presence of a wing as shown in the comparison in Figure 43. More pressure distribution comparisons can be found in Appendix A.

At $\beta=15^\circ$ the low pressure zone on the flat surface of the bow is somewhat smaller when the wing is present. This suggests that the flow has not separated as much as it has without the wing.

By looking at how the flow structure is affected and also when the wing generates a force in the ships direction it is possible to determine what causes the resistance reduction.

6.2.1. Effects on the flow structure
Streamlines for the solution on the refined mesh are compared in Figure 44. Note that the streamlines for the same apparent wind angle are generated in exactly the same manner in ParaView, i.e. the line through which they should pass is located at exactly the same place and the same amount of lines is used. More streamline comparisons are presented in Appendix B.
The presence of the wing changes the character of the flow in the area where the semicircular shape ends (this is marked with a red line in Figure 44). Also the flow streamlines are more spread out when the wing is present.

In order to determine the point of separation a horizontal cut plane is generated, at a height of 19 m above the symmetry plane, showing the velocity field. The results for the refined mesh are shown in Figure 45. Velocity fields around the entire ship are shown in Appendix C.

Keep in mind that the velocity is scaled in these calculations and therefore the results display the rather high velocities as presented in Figure 45.

The point of separation is slightly pushed back when the wing is present (as indicated by the white line in Figure 45). The distance is roughly estimated to be ~2 m. Also the flow is more accelerated when the wing is present.
6.2.2. Force generated from wing

The force coefficients presented in this section are normalized using the projected area of the wing in the respective directions. Also the values presented are not the mean value, as is the case for the total $C_X$ and $C_Y$ presented elsewhere, instead the value presented are those calculated in the last iteration of the flow solver. Figure 46 shows the resulting $C_X$ and $C_Y$ for the refined mesh with a wing mounted at 20° from the centerline. Negative values on $C_X$ indicate when the force is acting in the ships direction and for $C_Y$ this indicates that the force is acting in the negative $y$-direction.

![Figure 46. $C_X$ and $C_Y$ on the wing for the refined mesh.](image)

The wing does not seem to generate any force in the ship direction for $\beta<30°$. Instead it generates a force in the negative $y$-direction, reducing the heeling moment. A conclusion drawn from this is that the reduction of the resistance at $\beta<30°$ shown in Figure 41 is more likely to be an effect of how the wing deflects the flow structure than due to force generated in the ship direction (since $C_X$ for the wing is positive at these angles).

6.3. Effects of altering mounting position and sheet angle

Keep in mind that the results presented in this section are from calculations done on the coarsened mesh with a different turbulence model.

The changes in $C_X$ and $C_Y$ for three different mounting positions, $\zeta$, are presented in Figure 47.

![Figure 47. $\Delta C_X$ (left) and $\Delta C_Y$ (right) for the coarsened mesh with a wing mounted at 10° (dashed), 20° (dashed/dotted) and 30° (dotted).](image)

The largest $C_X$ difference for $\beta<30°$ occurs at $\beta=10°$ for all three mounting positions. The solution for $\zeta=20°$ shows the largest reduction of $C_X$ whilst $\zeta=10°$ shows the largest increase. The results for $\zeta=10°$ and $20°$ are however very similar up to $\beta=20°$. The wing mounted at $30°$ shows the lowest performance.
The corresponding percentages are presented in Figure 48.

![Figure 48. Percentage of the total $C_x$ and $C_y$ of the ship without wing for the coarsened mesh.](image)

The aerial resistance is reduced $\sim 7\%$ at $\beta = 10^\circ$ which is equivalent to $\sim 0.7\%$ of the total resistance acting on the ship. However, it appears not to matter where the wing is mounted, only that the positions nearer to the centerline show a slightly better performance. One other sheet angle, $\delta$, was also investigated. For all calculations so far $\delta$ has been fixed at $13^\circ$. One grid was created using the coarsened mesh in which the sheet angle was increased to $20^\circ$ in order to see if this had any effect. The resulting $\Delta C_x$ and $\Delta C_y$ are presented in Figure 49.

![Figure 49. $\Delta C_x$ and $\Delta C_y$ for the coarsened mesh when increasing the sheet angle, $\delta$.](image)

The result show no significant difference neither for $C_x$ nor $C_y$ for $\beta < 30^\circ$. The generated force from the wing is however affected, as shown in Figure 50.

![Figure 50. $C_x$ and $C_y$ on wing for the coarsened mesh when increasing $\delta$.](image)
The results show that the wing with higher $\delta$ generates force in the ship direction earlier (and consequently more) than the previous results. However since there was no significant change in the overall resistance, this further indicates that the force generated from the wing in ship direction does not affect the resistance as much as the change in flow behavior does. At least not for the setup investigated in this thesis.

7. CONCLUSIONS

The flow around the ship proved to be very three dimensional. This suggests that the 2-D bow effect assumed in Silvanius thesis is insufficient regarding properly accounting for the three dimensional behavior of the flow.

The overall reliability of the results presented here are poor to say the least. The results are dependent on grid size, domain size and turbulence model. Therefore the results presented should be considered with care since further calculation on a finer mesh could either confirm or contradict these results. Furthermore since it has been difficult to obtain satisfactory solutions, no work has been done to optimize the shape and position of the wing.

That being said, the results do confirm that the bow wing does affect the overall resistance in a positive manner, however nowhere near as much as predicted by Silvanius. A reduction of 2% of the total aerial resistance is observed which is equivalent to a 0.2% reduction of the total ship resistance.

Furthermore the results suggest that the resistance is greater affected by the change in flow behavior than a force generated in the ship direction, since neither altering the mounting position nor the sheet angle seemed to affect the overall resistance in a significant way. Also the different mounting positions yielded fairly similar results suggesting that where the wing is placed makes no difference regarding the generation of a force in the ship direction.

The wing affected the flow by accelerating it around the bow, causing a slightly delayed separation. The length of which is roughly estimated to be ~2 m.

8. RECOMMENDATIONS

Seeing that the presence of a wing does affect the airflow around the ship in a positive manner it is of high interest to continue investigating the possibilities that wings could give. The setup in this thesis was basically to validate the results from Silvanius studies by using his recommendations concerning position and sheet angle of the wing. These recommendations were optimized to give maximum lifting force in ship direction. Instead one should investigate how to best redirect the airflow and delay the point of separation as this seemed more effective. Maybe the end result is not a set of wings, but instead a vortex generator of sorts. This is preferably done in by wind-tunnel testing as the turnaround time is much faster and it is possible to go through with far more cases than with CFD computations during the same time. What should be of primary interest to investigate is how the resistance is affected by altering the sheet angle on the wing and also consider moving it closer to the hull.

If any further CFD calculations are to be done using the grids created here I would recommend testing different boundary conditions on the symmetry plane in order to determine this dependency. Also a further investigation into why the farfield radius seems to affect the results as much as it appears to do. Lastly I would recommend rerunning all viscous calculations using time accurate calculations for better accuracy. Also the default turbulence model should be used since it is better adapted for separated flow whilst the Spalart-Allmaras is a more general model.

Preferably the wing should be able to be retractable, at least when traveling in headwind since it significantly increases the resistance of the ship.
APPENDIX A – PRESSURE DISTRIBUTION COMPARISONS

Figure A - 1. Apparent wind angles 0°-20° for the refined mesh with (right) and without (left) wing. The angles in which images are missing for the winged mesh have not been calculated.
Figure A-2. Pressure distribution at apparent wind angles 25°-45° for the refined mesh with (right) and without (left) wing. The angles in which images are missing for the winged mesh have not been calculated.
APPENDIX B – STREAMLINE COMPARISONS

Figure B - 1. Streamlines for the refined mesh with (right) and without (left) wing at apparent wind angles 0°-15°. The angles in which images are missing for the winged mesh have not been calculated.
Figure B - 2. Streamlines for the refined mesh with (right) and without (left) wing at apparent wind angles 20°-35°. The angles in which images are missing for the winged mesh have not been calculated.
Figure B - 3. Streamlines for the refined mesh without (left) wing at apparent wind angles 40°-45°. These angles have not been calculated for the refined mesh with wing.
Figure C - 1. Velocity fields for the refined mesh with (right) and without (left) wing at apparent wind angles of 0°-25°. The angles in which images are missing for the winged mesh have not been calculated.
Figure C-2: Velocity fields for the refined mesh with (right) and without (left) wing at apparent wind angles of 30°-45°. The angles in which images are missing for the winged mesh have not been calculated.
REFERENCES


5. ANSYS, Inc.: ANSYS ICEM CFD 12.1 Brochure. USA, 2009.


7. PDC, [online]: www.pdc.kth.se.


