Accuracy of transient versus steady state forces on a rudder operating in a propeller slipstream

Noggrannheten hos tidsberoende kontra stationära krafter på ett roder i en propellerström

Rolls-Royce AB
June, 2015

Erik Löfgren
erla0044@student.umu.se

Master’s thesis in Engineering Physics, 30 ECTS
Department of Physics, Umeå University

Supervisor: Urban Svennberg, Rolls-Royce AB
Examiner: Krister Wiklund, Department of Physics, Umeå University
Accuracy of transient versus steady state forces on a rudder operating in a propeller slipstream

Abstract

In computational fluid dynamics (CFD), a transient simulation is in general more costly than computing the steady state of the system, if such a state exists. The velocity field produced by the propeller blades upstream of a rudder is transient in nature, and rudder design using CFD may therefore become very time-consuming. If a steady solution could accurately predict the performance of the rudder, such an approach would be favourable. The aim of the present study was to assess the possibility to accurately predict the performance of a rudder operating in a propeller slipstream using steady state simulations, e.g. an actuator disk model (ADM). For this reason, the performance of the two-dimensional NACA 0021 rudder section submitted to a sinusoidal transverse gust, representing a transient propeller slipstream, was simulated using ANSYS Fluent. The predicted force coefficients are presented for a number of gust amplitudes, mean angles of attack and reduced frequencies of the transverse gust. The simulations have shown that the modelling error introduced when predicting the performance in a steady state is highly dependent on all these parameters of the actual transient flow, and that the steady result may be a severe over- or under-prediction of the real performance of the rudder. Heavily loaded propellers are suspected to be less suitable for ADM modelling in rudder performance prediction. The predicted unsteady lift coefficient was compared to the linear theories of Horlock and Sears, and the agreement was fair at zero mean angle of attack but poor at a mean angle of attack of 10°. It was also found that the predicted performance of the rudder was significantly altered when the chord based Reynolds number was increased by a factor of 10, which has implications on the validity of model-scale simulations. The effect of including turbulent transition modelling for some of the simulations was also investigated, and the discrepancy in predicted performance was found to be considerable. Due to the formation of a laminar separation bubble the predicted trailing edge separation and viscous stress on the rudder were significantly decreased, leading to better overall performance.
Noggrannheten hos tidsberoende kontra stationära krafter på ett roder i en propellerström

Sammanfattning

**Nomenclature**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D$</td>
<td>mm, m</td>
<td>Propeller diameter</td>
<td>-</td>
</tr>
<tr>
<td>$J$</td>
<td>-</td>
<td>Advance coefficient</td>
<td>(2.12)</td>
</tr>
<tr>
<td>$n$</td>
<td>s⁻¹</td>
<td>Propeller rate of revolutions</td>
<td>-</td>
</tr>
<tr>
<td>$V_A$</td>
<td>m s⁻¹</td>
<td>Speed of advance</td>
<td>-</td>
</tr>
<tr>
<td>$Z$</td>
<td>-</td>
<td>Number of blades</td>
<td>-</td>
</tr>
<tr>
<td>$\omega$</td>
<td>rad s⁻¹</td>
<td>Angular frequency</td>
<td>-</td>
</tr>
<tr>
<td>$\nu$</td>
<td>m² s⁻¹</td>
<td>Kinematic viscosity</td>
<td>-</td>
</tr>
<tr>
<td>$\rho$</td>
<td>kg m⁻³</td>
<td>Density</td>
<td>-</td>
</tr>
<tr>
<td>$P$</td>
<td>kg m⁻¹ s⁻²</td>
<td>Pressure</td>
<td>-</td>
</tr>
<tr>
<td>$y^+$</td>
<td>-</td>
<td>Non-dimensional wall distance</td>
<td>(2.16)</td>
</tr>
<tr>
<td>$u_r$</td>
<td>m s⁻¹</td>
<td>Friction velocity</td>
<td>(2.15)</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>kg m⁻¹ s⁻²</td>
<td>Wall shear stress</td>
<td>-</td>
</tr>
<tr>
<td>$\mu$</td>
<td>kg s⁻¹ m⁻¹</td>
<td>Dynamic viscosity</td>
<td>-</td>
</tr>
<tr>
<td>$L$</td>
<td>kg m s⁻²</td>
<td>Lift force</td>
<td>-</td>
</tr>
<tr>
<td>$D$</td>
<td>kg m s⁻²</td>
<td>Drag force</td>
<td>-</td>
</tr>
<tr>
<td>$c_l$</td>
<td>-</td>
<td>Lift coefficient per unit span</td>
<td>(2.5)</td>
</tr>
<tr>
<td>$c_d$</td>
<td>-</td>
<td>Drag coefficient per unit span</td>
<td>(2.6)</td>
</tr>
<tr>
<td>$c_c$</td>
<td>-</td>
<td>Chord force coefficient per unit span</td>
<td>(2.7)</td>
</tr>
<tr>
<td>$C$</td>
<td>-</td>
<td>Courant number</td>
<td>(2.14)</td>
</tr>
<tr>
<td>$k$</td>
<td>-</td>
<td>Reduced frequency</td>
<td>(2.10)</td>
</tr>
<tr>
<td>$\alpha_{LE}$</td>
<td>-</td>
<td>Angle of attack at leading edge</td>
<td>-</td>
</tr>
<tr>
<td>$U_m$</td>
<td>m s⁻¹</td>
<td>Mean velocity</td>
<td>-</td>
</tr>
<tr>
<td>$u_g$</td>
<td>m s⁻¹</td>
<td>Velocity amplitude of the transient gust</td>
<td>-</td>
</tr>
<tr>
<td>$Re$</td>
<td>-</td>
<td>Reynolds number</td>
<td>$Re = U_m c / \nu$</td>
</tr>
</tbody>
</table>
# Contents

1. Introduction ................................................................................................................................. 1  
   1.1 Background ......................................................................................................................... 1  
   1.2 Aim ...................................................................................................................................... 1  
   1.3 Tasks .................................................................................................................................... 1  
   1.4 Limitations .......................................................................................................................... 1  

2. Theory ......................................................................................................................................... 2  
   2.1 Governing equations ............................................................................................................ 2  
   2.2 Definitions ........................................................................................................................... 2  
      2.2.1 Force coefficients ........................................................................................................ 2  
      2.2.2 Propellers and rudders ................................................................................................. 3  
      2.2.3 Numerical parameters in turbulence modelling ........................................................... 3  
   2.3 Analytic results for an aerofoil ............................................................................................ 4  
   2.4 Rudder in a propeller slipstream ......................................................................................... 4  
   2.5 Unsteady inflow .................................................................................................................. 6  
      2.5.1 Quasi-static regime ...................................................................................................... 7  
      2.5.2 Dynamic regime .......................................................................................................... 7  
      2.5.3 Sears’ and Horlock’s models for the unsteady lift ....................................................... 7  
      2.5.4 Dynamic stall ............................................................................................................... 8  
   2.6 Effect of turbulence and boundary layers on rudder forces ................................................. 9  
   2.7 Turbulence modelling ........................................................................................................ 10  
      2.7.1 Reynolds Averaged Navier-Stokes ............................................................................ 10  
      2.7.2 The SST $k - \omega$ model ............................................................................................. 11  
      2.7.3 Transition modelling ................................................................................................. 11  
   2.8 3D effects .......................................................................................................................... 12  

3. Method ...................................................................................................................................... 13  
   3.1 Simulations ........................................................................................................................ 13  
   3.2 Turbulence model .............................................................................................................. 13  
   3.3 Fluid properties ................................................................................................................. 13  
   3.4 Geometry ........................................................................................................................... 13  
   3.5 Boundary conditions .......................................................................................................... 15  
      3.5.1 Inlet boundary condition ............................................................................................ 15  
      3.5.2 Rudder ....................................................................................................................... 16  
      3.5.3 Far field boundary conditions .................................................................................... 16  
   3.6 Mesh .................................................................................................................................. 16  
      3.6.1 Main flow .................................................................................................................. 16  
      3.6.2 Rudder ....................................................................................................................... 17  
      3.6.3 Far field boundaries ................................................................................................. 19  
   3.7 Solver settings ................................................................................................................... 19
3.8 Reference frame ................................................................................................................ 19
3.9 Time step ........................................................................................................................... 20
3.10 Test matrix and procedure ................................................................................................. 21
4 Results ....................................................................................................................................... 22
4.1 Steady simulations ............................................................................................................. 22
4.2 Transient simulations ........................................................................................................ 24
  4.2.1 Effect of the reduced frequency and gust amplitude at zero mean angle of attack ... 24
  4.2.2 Effect of the mean angle of attack on the chord force coefficient ......................... 25
  4.2.3 Effect of the mean angle of attack on the side force coefficient ......................... 29
  4.2.4 Effect of transition modelling and Reynolds number ........................................ 30
  4.2.5 Effect of waveform ................................................................................................. 32
  4.2.6 Comparison with Sears’ and Horlock’s theory ...................................................... 34
  4.2.7 Comparison with the Shilling rudder ................................................................. 35
4.3 Summary of the results ...................................................................................................... 35
4.4 Practical comments and sources of error ........................................................................... 36
5 Discussion and conclusions ....................................................................................................... 37
  5.1 Accuracy of transient versus steady state rudder forces ............................................. 37
  5.2 Suggestions for an improved steady state prediction ................................................ 38
  5.3 General conclusions and other comments ..................................................................... 39
6 Future developments ................................................................................................................. 41
7 References ......................................................................................................................................... 42
Appendix A. Table of values of Sears’ and Horlock’s functions ........................................... i
Appendix B. Coordinates of the NACA 0021 profile ................................................................. ii
Appendix C. Derivation of the inlet boundary condition ........................................................... iii
Appendix D. Figures of the dimensionless wall distance ......................................................... iv
Appendix E. Convected vortices from the leading edge ........................................................... vi
Appendix F. Streamline plots of the NACA 0021 ................................................................. vii
Appendix G. Tables of lift and drag coefficients at $\alpha m = 10^\circ$ and $15^\circ$ ............................ ix
1. Introduction

1.1 Background
Computational fluid dynamics (CFD) is today used for research purposes as well as for product design at private companies. As computational resources become more accessible the possibility of modelling complex turbulent flows increases, but the calculations may still become very time-consuming in many applications. When simulating the flows present in a propeller-rudder configuration, much of the computational effort is put into resolving the transient field produced by the propeller blades. When designing equipment that is submitted to the propeller induced flow, and when the details of the flow near the propeller is less important, a steady state representation of the propeller induced flow is beneficial.

The forces on a rudder are the result of complex interactions between free-stream flow and boundary layers, shed vortices, unsteady separation and reattachment, among other things. This makes them hard to predict accurately without numerical simulation. The propeller slipstream is also complex and transient by nature, exposing a rudder in the wake of a propeller to a very non-uniform inflow in both spanwise and chordwise directions. Modelling the propeller slipstream with a steady state representation, for instance an actuator disk model (ADM) [1], must for these reasons be validated and justified.

It is also common to include model-scale testing in the development phase of many products, rudders and propellers being no exception. The scaling may in itself lead to discrepancies between model test results and the performance of the finished product. In order to generate data which is transferrable to full scale, certain dimensionless parameters restrict the choices of model test conditions. However, it is for instance typically not achievable to reach full-scale Reynolds numbers in industrial testing facilities, making it important to know the impact of the scaling effect on the predicted performance.

1.2 Aim
The aim of this work was to determine the errors that arise in computed rudder forces due to the modelling of the transient propeller induced flow as a stationary inflow. The possibilities and prerequisites for an idealized propeller slipstream to yield acceptable errors in rudder forces were to be identified, theoretically or empirically through simulation. The effect on results and accuracy caused by the choice of turbulence model and meshing was to be evaluated.

1.3 Tasks
The conditions and parameter values for which the rudder performance may be accurately calculated using a stationary representation of the propeller induced flow were to be determined. In this purpose, simulations were to be carried out where the effect on the transient results and accuracy were evaluated as the chosen physical parameters where varied:

- The reduced frequency, $k$, which is a dimensionless parameter proportional to the number of wavelengths present on the rudder at one instant.
- The amplitude of the periodic oscillations in the inflow.
- The Reynolds number.

The dynamics of the flow around the rudder was to be physically explained in the different regions of the test matrix. In order to evaluate the effect on the results and accuracy caused by numerical settings, simulations were to be performed with and without transition modelling and on different meshes.

1.4 Limitations
The work in this thesis was limited to non-cavitating, i.e. single-phase, flows in order to let the focus be on the comparison of steady versus unsteady simulations. Also, the calculations were restricted to two-dimensional geometries in order to limit computational costs and simplify the analysis.
2 Theory

This section is intended to serve as reference for the reader, a review of the earlier work done on the topic and as support for the claims made and conclusions drawn from the results presented in this work.

2.1 Governing equations

The dynamics of an incompressible Newtonian viscous fluid are described by the momentum equations

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial}{\partial x_j} u_i = - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{1}{\rho} f_i, \quad i = 1, 2, 3
\] (2.1)

and the equation of mass conservation

\[
\frac{\partial u_i}{\partial x_i} = 0
\] (2.2)

where \( u_i \) are the velocity components in the directions of \( x, y \) and \( z \), respectively, \( \nu \) is the kinematic viscosity, \( \rho \) is the fluid density and \( f_i \) is the \( i \):th component of any external force. For repeated indices, the Einstein summation convention is used. The set of coupled equations given by (2.1) are known as the Navier-Stokes (N-S) equations [2]. The N-S equations are non-linear and closed-form analytical solutions can be found only for a limited set of simple configurations. In practical applications, the N-S equations are often either simplified, simulated or both. The field of simulating the dynamics of a fluid is known as computational fluid dynamics (CFD).

2.2 Definitions

In this section, some of the more important parameters to this work are explained.

2.2.1 Force coefficients

A body moving at a non-zero velocity relative to the surrounding fluid experiences a net force due to pressure and viscous forces. The component of the force perpendicular to the free-stream velocity is called the lift, usually denoted \( L \). One may also define the drag, \( D \), which is the net force component parallel to the free-stream velocity. From the drag and lift forces one may form the non-dimensional coefficients known as the lift coefficient

\[
C_L = \frac{L}{\frac{1}{2} \rho U_{\infty}^2 S}
\] (2.3)

and the drag coefficient

\[
C_D = \frac{D}{\frac{1}{2} \rho U_{\infty}^2 S}
\] (2.4)

where \( \rho \) is the density of the fluid, \( S \) is the planform area of the body and \( U_{\infty} \) is the free-stream velocity. When considering a 2D-model, e.g. a wing profile of infinite span, one may instead want to define the coefficients per unit length in the spanwise direction as

\[
c_l = \frac{L}{\frac{1}{2} \rho U_{\infty}^2 c}
\] (2.5)

\[
c_d = \frac{D}{\frac{1}{2} \rho U_{\infty}^2 c}
\] (2.6)

where \( c \) is the chord length of the profile, which is defined as the distance between the leading and trailing edges [3]. We may also define the chord force coefficient per unit span as
where $F_c$ is the force in the direction of the chord, with positive force being in the direction from leading to trailing edge. If the angle between the chord and mean velocity is $\alpha_m$, then it holds that

$$c_c = c_c \cos \alpha_m - c_l \sin \alpha_m.$$  

Similarly, the side force perpendicular to the rudder chord is given by

$$c_s = c_d \sin \alpha_m + c_l \cos \alpha_m.$$  

### 2.2.2 Propellers and Rudders

When considering periodic flows around a streamlined body, e.g. a rudder, it is common to use the reduced frequency to characterize the flow conditions [1]. This parameter is given by

$$\phi = \frac{\omega c}{2U_\infty}$$  

where $\omega$ is the angular frequency of the periodic flow component. For a propeller with $Z$ blades rotating at $n$ revolutions per second, this becomes

$$k = \frac{nZc}{U_\infty}.$$  

From (2.11) we interpret the reduced frequency as $\pi$ times the number of wavelengths present on the rudder at one instant. One of the most essential parameters when describing the operational conditions of a ship propeller is the advance coefficient, relating the forward velocity of the ship to the tangential velocity of the propeller tip. The advance coefficient is given by [3]

$$J = \frac{V_a}{nD}$$  

where $D$ is the diameter of the propeller and $V_a$ is the speed of advance with respect to the surrounding fluid. For a fixed geometry, i.e. where $c$ and $D$ are constants, and at a given free-stream velocity $U_\infty = V_a$ the reduced frequency is simply related to $J$ through

$$k = \frac{\pi c}{D} J^{-1}.$$  

### 2.2.3 Numerical Parameters in Turbulence Modelling

The Courant number is a numerical parameter relating the mesh size $\Delta x$, time step $dt$ and characteristic velocity $U$ of the flow. It represents the typical number of cells a fluid parcel passes through during one time step. The Courant number is defined as [1]

$$C = U \frac{dt}{\Delta x}.$$  

The friction velocity is a property of the near-wall flow with units of velocity and is defined as [4]

$$u_\tau = \sqrt{\tau_w/\rho}$$  

where $\tau_w$ is the wall shear stress and $\rho$ is the fluid density. The friction velocity is used to calculate the dimensionless wall distance, which is given by [2]

$$y^+ = \frac{u_\tau}{v} y$$  

where $y$ is the (dimensional) distance from a given point in the velocity field to the wall. The friction velocity may be calculated by estimating the wall shear stress using tabulated friction coefficients. D.J. Tritton claims that under laboratory conditions it typically holds that [2]

$$u_\tau \in (0.035 U_\infty, 0.05 U_\infty).$$  

3
From equations (2.16) and (2.17) and the height of the first mesh cell at the wall, it is possible to get a quick estimate of the $y^+$ value in the first cell. The wall $y^+$ value is an important mesh parameter in many turbulence models, and ensuring that it falls within a certain model specific range is often required to properly resolve the boundary layer flows.

2.3 Analytic results for an aerofoil

By use of theoretical models it is possible to predict the performance of a rudder or propeller blade based on its geometric properties. Possibly the two most common methods are the lifting line and lifting surface methods. In the former model, the aerofoil is represented by a line vortex of varying strength in the spanwise direction, typically passing through the aerodynamic centres of the cross-sections [3]. The lift distribution in the spanwise direction $y$ is then related to the circulation through

$$L(y) = -\rho U_\infty \Gamma(y)$$  \hfill (2.18)

where $\Gamma$ is the circulation given by

$$\Gamma = \oint u \cdot dl.$$  \hfill (2.19)

Equation (2.18) is known as the Kutta-Joukowski theorem. When the flow around the aerofoil is in a steady state, the flow leaving the trailing edge from the suction side has the same velocity as the flow leaving from the pressure side of the trailing edge. This is known as the Kutta condition [3], and it also implies that the rear stagnation point is located at the trailing edge. The so-called lifting surface models also take a chordwise distribution of vorticity into account. Both the lifting line and the lifting surface models are based on potential theory [1], i.e. the assumption that the velocity may be written

$$\bar{u} = \nabla \phi.$$  \hfill (2.20)

where $\phi$ is the velocity potential. By imposing the Kutta condition, the potential flow problem has a unique solution. The existence of a potential implies that the flow is irrotational, and the theoretical models also include an inviscid fluid assumption. Such assumptions are typically valid in the outer flow far from the boundary layer of the aerofoil. However, due to the inviscid assumption, models based on potential theory cannot directly predict flow separation or drag due to skin friction [5]. From classical thin aerofoil theory, it is known that the lift coefficient increases linearly with the angle of attack, $\alpha$, as

$$c_l = 2\pi \alpha$$  \hfill (2.21)

for small angles of attack [6]. For a real aerofoil of finite thickness the behaviour of $c_l$ is usually linear in the low $\alpha$ regime, although with a problem-dependent slope, generally different from $2\pi$. The positive slope of the lift coefficient is sustained up to the stall angle, above which the lift decreases drastically due to separation of the flow at the leading edge [2]. As stall is approached, the rear stagnation point also moves away from the trailing edge, signalling the breakdown of the Kutta condition.

2.4 Rudder in a propeller slipstream

Conventional ship rudders often have a spanwise cross-section that is well represented by a symmetric aerofoil (or hydrofoil) profile. One such profile is the NACA0021 profile, shown in Figure 1. The rudder is in its design conditions submitted to the propeller slipstream which is a transient velocity field, periodic in space and time. The axial and transversal velocity components in the slipstream may be of comparable magnitude [7], implying that the angle of attack of the inflow to the rudder may be of the same order as the stall angle of the rudder [1] [5].

The NACA0021 has a static stall angle of approximately 18° depending on the chord-based Reynolds number, turbulent intensity of the inflow [8] and possibly other parameters. For angles significantly smaller than the stall angle the behaviour is described by the linear theory, where the 2D lift coefficient is proportional to the angle of attack. In the static case at angles of attack greater than the
stall angle, the force coefficients are dramatically affected and deviate from their pre-stall behaviour. Stationary solutions may generally not be found in the post-stall regime.

The hydrodynamic performance of a rudder depends on the force that is parallel to the advance velocity of the ship and as such affects the efficiency of the system. At design conditions (rudder parallel to propeller axis) this is the force in the chordwise direction. The chord force may at a given spanwise cross-section have contributions from both the lift and drag forces, defined with respect to the angle of attack at each cross-section. For this reason, the breakdown of lift in the post-stall regime will affect the chord force, even when the rudder is at 0° incidence to the direction of advance. If at two spanwise rudder sections the angles of attack are equal but of opposite sign, the net force will be along the chord, as seen in Figure 1.

This effect is relevant in practical situations, since the velocity field in a typical propeller-rudder configuration is axisymmetric about the propeller axis when time averaged and at open water conditions. This leads to a cancellation of the side forces on the rudder at typical design conditions. However, modern rudders are commonly twisted along the spanwise direction, meaning that the chord is not aligned with the direction of advance at each cross-section. Instead, the chord line at each spanwise section is typically aligned with the expected angle of attack of the propeller slipstream at the section. This decreases the risk of rudder damage due to cavitation and may improve performance. In this case, the side force perpendicular to the chord at a given section has a contribution to the total performance of the rudder.

The propeller slipstream is inherently unsteady and the rudder experiences it as a periodic inflow, with frequency equal to the number of blades times the number of propeller revolutions per second. The characteristic features of the propeller slipstream are the pronounced helical structure, the tip vortex that is traced by the leading edge of each propeller blade and a hub vortex extending axially from the shaft line of the propeller. The flow is accelerated by the thrust of the propeller, and the slipstream contracts preserving mass balance, as presented by Molland and Turnock in [1]. The pressure in the tip vortex or on the suction side of the propeller blades may become low enough for the water to vaporize. This phenomena is known as cavitation and can be observed in the tip vortex shown in Figure 2. Measurements of the propeller slipstream and an investigation of the interaction of the tip vortex with a rudder was given by Felli et al. [9].

---

1 No hull or free surface effects.
2.5 Unsteady inflow

When the angle of attack is not constant in time the force coefficients may behave quite differently as compared to the steady case. The Kutta-Joukowski theorem given by equation (2.18) does not hold for unsteady motion [10], and therefore unsteady aerofoil models have been developed. The effects of unsteady flow on the hydrodynamic forces exerted on streamlined profiles have mainly been investigated by considering time-dependent displacements of the profile itself, rather than varying the background velocity field. A transient angle of attack may for instance be generated by forcing the aerofoil into pitching [11] and/or plunging [6] motion. However, these approaches are not dynamically equivalent to an inflow of transient angle of attack [12] nor are they fully able to represent the situation of a rudder in a propeller slipstream, where several impulses from the rotating blades may be present at the rudder at a given time. In order to properly represent even a simplified propeller slipstream, it is therefore more realistic to generate a time-dependent inflow.

To gain fundamental understanding of the response of the rudder to an angle of attack that varies with time, one can generate an inflow that is sinusoidal in space and time and investigate the effects on drag and lift coefficients due to the parameters that describe the flow. Paterson and Stern [13] investigated the response of a NACA66 to a sinusoidal transverse gust superimposed on a uniform velocity field as illustrated in Figure 3. It was found that the results depended largely on the reduced frequency, \( k \), which is defined by equation (2.10). Many other researchers have also studied the response of a system due to changes in this parameter, e.g. [5] [6] [11] [14], and the general consensus seems to be that \( k \) has a large influence on the performance of the aerofoil. Because of this, the reduced frequency will be central to the considerations in this work, and the following sections will discuss the different dynamics of the rudder in different ranges of the value of \( k \).

Figure 2: Cavitating tip vortex traced out by four-bladed propeller, illustrating the helical shape of the propeller slipstream. Picture courtesy of Rolls-Royce AB.

Figure 3: The components of velocity; mean velocity and periodic transverse gust.
2.5.1 **Quasi-static regime**

In order to explain the dynamics of the aerofoil with respect to the flow parameters it may be useful to divide the ranges of the governing parameters into several regimes. These may be categorized by the amplitude and reduced frequency of the oscillations. In 1986, Poling and Telonis showed that the Kutta condition, on which potential theory, lifting line and surface models rely, is not valid for $k$ larger than 2 [15] but has been shown to break down for an oscillating aerofoil already at $k \approx 1$ [14]. Unsteady effects in lift have even been shown for reduced frequencies as small as $k = 0.0095$ [16].

In the limit of small $k$, known as the quasi-static limit, the drag and lift coefficients at a given instant behave as their static counterparts at the current angle of attack. There is no universal definition of where the quasi-static regime ends and the dynamic one begins, but Baik et al. [6] noted quasi-static behaviour for a pitching aerofoil at $k < 0.3$. In typical full scale propeller-rudder systems, the reduced frequency might be of the order of $k = 10 - 15$, based on a blade frequency of 4 impulses per second, a rudder chord of 5 meters and a propeller slipstream velocity of 5 m/s passing the rudder. From this estimate we see that the rudder in its design conditions are submitted to highly unsteady flows, for which the lifting surface method is not valid.

2.5.2 **Dynamic regime**

In the experiments of Paterson and Stern [13] the signals in drag and lift coefficients were shown to have a strong first harmonic response, i.e. variations with the same frequency as the incident flow. For higher reduced frequency the amplitude of the periodic response in lift was smaller.

Similar results were shown in simulations by James C. Date, who in 2002 investigated the performance of the symmetric NACA0020 aerofoil submitted to a transverse periodic gust [5]. In this work it was shown that the time average of the drag coefficient varies significantly with reduced frequency. In fact, for an inflow oscillating with reduced frequency $k = 0.5$ and a gust aspect ratio of 0.25, the mean value of the drag was negative, and thus the aerofoil was producing thrust. At $k = 5$ the mean drag was positive and 4% higher than for the steady case with no periodic gust. In all simulations the mean angle of attack was zero.

The possibility of an aerofoil to produce thrust in an oscillating free stream is sometimes called the Katzmayr effect, after the director of the Vienna Aerodynamical Laboratory who first described it in 1922. This effect is important for the present study since it concerns the performance of a rudder in an unsteady inflow. In a technical memorandum of the National Advisory Committee for Aeronautics (now known as NASA) Shatswell Ober [17] gives a qualitative explanation of this phenomenon. Ober argues that if the lift on the aerofoil, as measured with respect to the instantaneous angle of incidence, has a component antiparallel to the mean flow this force may exceed the drag force and thus yield a net thrust in the direction of the mean flow. Ober also points out that even small amplitude oscillations (1° in angle of attack) may lead to significant errors in drag (on the order of 5%), which could explain experimental errors in wind tunnel measurements. However, the aforementioned work was based on the quasi-static assumption even for reduced frequencies as high as $k = 0.6$, so the numerical accuracy of the results might be questionable.

2.5.3 **Sears’ and Horlock’s models for the unsteady lift**

In 1968, J. H. Horlock gave an explicit expression for the response amplitude of the lift of an aerofoil encountering a gust of arbitrary waveform at a non-zero mean angle of attack [18]. In his work, which was based on earlier derivations for gusts purely perpendicular to the chord by Sears [19], he showed that the unsteady component of lift was given by

$$\Delta L_n = \pi c \rho U_m u_n e^{i \omega t} (S(k) - \alpha T(k) \cot(\beta_m))$$  \hspace{1cm} (2.22)

where $u_n$ is the coefficient of the $n$:th Fourier mode of the gust, defined by

$$\bar{u} = \Sigma_n u_n \sin \frac{2\pi nz}{l}$$  \hspace{1cm} (2.23)

where $\bar{u}$ is the magnitude of the gust, $z$ is the coordinate in the direction of its propagation and $l$ is the width of the gust along the $z$-axis. The angles $\alpha$ and $\beta_m$ are, respectively, the unperturbed angle
of attack and the mean angle of attack with the perturbation. Equation (2.22) contains two complex functions of the reduced frequency, $T$ and $S$, named Horlock’s and Sears’ functions respectively. Their values are typically taken from tables, as presented in appendix A or more extensively in [20].

The results presented by Horlock, as well as Sears’ theory upon which it is based, predicts a decreasing response amplitude with increasing reduced frequency, as observed by both James C. Date and Paterson and Stern. The phase lag between gust and response is also depending on the reduced frequency. Date found reasonable agreement between CFD computations on the NACA 0020 and predictions of Sears’ theory for the amplitude in $c_f$ due to low frequency gusts ($k \approx 1$), but deviations from the linear theory at higher frequency [5]. As is true for all models based on potential theory, the viscous drag is not accounted for by Sears’ or Horlock’s theories. The distortion of the gust due to the presence of the body is also neglected, although it may in practice be a significant effect.

2.5.4 Dynamic stall

When the angle of attack changes rapidly in time due to some form of relative motion of the aerofoil and the surrounding medium, and the angle of attack at some instant exceeds the static stall angle, the phenomenon called dynamic stall may occur. Dynamic stall has been widely studied by the aerospace branch of the scientific community, due to its importance for helicopter and airplane performance. Qualitatively, it is a delay in the onset of stall yielding lift at angles greater than the static stall angle [21]. The increase in lift is due to a bound vortex on the suction side which is sustained by the relative motion of the aerofoil to the fluid, according to the conjecture of Choudhry (2014) for a pitching aerofoil which is illustrated in Figure 4 [16].

The increased lift coefficient at high angles of attack during the upstroke pitching motion of an aerofoil (as seen in Figure 4) is associated with a hysteresis effect. For an aerofoil pitching around a non-zero mean angle of attack, the lift coefficient is smaller during the down stroke and the reattachment of the flow occurs at an angle smaller than the static stall angle [16]. Whether this effect also occurs for stationary aerofoils submitted to unsteady inflow is unclear, since the author is unaware of any experimental studies on this topic.

![Figure 4: Illustration of the dynamic stall process of a pitching aerofoil, courtesy of Amanullah Choudhry [16].](image-url)
There are models that take account for the unsteady phenomena of dynamic stall when calculating the time history of the force coefficients, often through a set of differential equations based on empirical coefficients and experimental data. A dynamic stall model that was originally developed for rotorcraft applications is the Beddoes-Leishman model [22]. The dynamic stall models have however often been unable to produce results that accurately agree with experiments, especially in the post-stall regime [21]. Another limitation of the Beddoes-Leishman model in specific is that the angle of attack is assumed to be uniform along the chord of the body, which is generally not the case for a rudder in a propeller slipstream.

2.6 Effect of turbulence and boundary layers on rudder forces

The N-S equations (2.1) that govern the dynamics of a fluid are a set of coupled non-linear differential equations. When the non-linear terms dominate over viscosity, meaning that the Reynolds number is large in comparison to some case dependent value, this leads to the production of flow structures over a wide span of length scales. The large scale motion creates motion at smaller scales, and they in turn give rise to even smaller structures. Eventually the coherent structures are small enough, and the gradients sharp enough, for viscosity to dominate and the so-called energy cascade terminates with dissipation at the Kolmogorov length scale. Turbulence implies motion throughout this entire spectrum, from the integral scale down to the viscous scale [23].

Even in a turbulent flow, viscosity may be dominant in the so-called boundary layer that forms close to solid walls. The boundary layer typically begins with a laminar part at the leading edge of a rudder, in which the velocity follows the well-known Blasius profile. The laminar boundary layer is unstable, and as it grows in thickness along the boundary it is typically followed by a transition region and a fully turbulent boundary layer. The turbulent boundary layer is typically associated with a larger skin friction drag coefficient [1], but may sustain larger adverse pressure gradients than a laminar boundary layer and is thereby less susceptible to separation [24].

A separated laminar boundary layer may also undergo transition into a turbulent boundary layer, at which point it may reattach to the body. Between the points of separation and reattachment, a region called a laminar separation bubble (LSB) is formed, as described by Mayle, referenced in [24]. Choudhry et al. performed simulations on the NACA 0021, investigating the LSB and its effects on the hydrodynamic performance of the section. It was found that this bubble may act as an effective camber of the body, leading to augment in both lift and drag coefficients [25].

Due to practical reasons, experimental testing during the design process is commonly done on a geometry that has been scaled down by an order of magnitude or more. Depending on the testing facility and restrictions on other dimensionless numbers, such as $J$ and the Froude number, it may not be possible to recreate full scale Reynolds numbers. For this reason, it is important to know what quantitative effect $Re$ has on the predicted performance.

The nature of the main flow and its turbulent properties are known to alter the performance of a rudder placed in it. For instance, the level of turbulent intensity in the inflow has been shown to significantly affect the static stall angle of an aerofoil. Swalwell et al. found in experiment that the stall angle of the NACA 0021 section increased from about 17.5° to 20° when the turbulent intensity was changed from 4% to 7% [8]. At similar Reynolds numbers, measurements on the NACA 0021 presented in a technical report by Sandia National Laboratories indicated a stall angle of only 13° at an unspecified level of turbulent intensity, which shows how large the changes in stall angle may be [26].

Choudhry et al. explained the improved performance of the NACA 0021 with increased Reynolds number as a result of the laminar separation bubble becoming smaller. The laminar region at the leading edge was made longer by increasing the Reynolds number, but the transition and subsequent reattachment happened earlier, leading to a shorter LSB. The lift performance of the section was thereby improved with increasing Reynolds number [25]. A short LSB may also lead to improved performance due to separation-induced transition [24]. Similarly, the stall angle and maximum lift coefficient of the NACA 0021 has been found to increase with chord based Reynolds number [27]. Simulations carried out by James C. Date on a NACA 0020 showed that predicting full-scale ($Re \approx$
4.2 \cdot 10^7) forces with simulations on model scale \( (Re \approx 2.1 \cdot 10^5) \) resulted in an under-prediction of the stall angle by 24\% [5]. Thus, it is apparent that the turbulence properties as well as scaling effects may significantly alter the result and accuracy of the performance prediction of marine equipment using CFD.

2.7 Turbulence modelling

The wide ranges of temporal and spatial scales of the turbulent eddies make it extremely difficult to resolve the entire turbulent velocity field directly. This approach, which requires a mesh resolution and time step size that today simply makes it too expensive for industrial purposes, is called direct numerical simulation (DNS). In order to make simulation times reasonable, some form of turbulence modelling is in general necessary.

2.7.1 Reynolds Averaged Navier-Stokes

The most common turbulence models in industrial use today are based on the Reynolds averaged Navier-Stokes (RANS) equations. The fundamental assumption of the RANS equations are that the velocity and pressure fields can be written as the sum of a time averaged and a fluctuating (turbulent) part, i.e.:

\[
\begin{align*}
  u_i &= \bar{u}_i + u'_i \\
  p &= \bar{p} + p'
\end{align*}
\]  

(2.24)

(2.25)

where the bar represents the time average and the variables with an apostrophe are the turbulent fluctuations. The Navier-Stokes equations are then time averaged over a suitable time scale greater than the largest time scale of the turbulent fluctuations. Using the linear property of the time averaging operator and that the time average of each fluctuation term is zero, the RANS equations become [4]

\[
\begin{align*}
  \frac{\partial \bar{u}_i}{\partial t} + \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} &= -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial (\bar{u}_i' \bar{u}_j')}{\partial x_j} \\
  \frac{\partial \bar{u}_j}{\partial x_j} &= 0.
\end{align*}
\]  

(2.26)

(2.27)

Note that the partial time derivatives of the mean velocities do not vanish, as there may be variations in the mean velocity field at a time scale larger than the turbulent time scales, for instance due to variations in an external force or boundary conditions. As can be seen from equation (2.26) the time averaging operation adds four new unknowns to the original N-S equations (nine unknowns in the three-dimensional case). These are the correlations of the form \( \bar{u}_i' \bar{u}_j' \), known as the Reynolds stresses. In order to close the problem, the Reynolds stresses are commonly assumed to be related to mean flow properties through [4]

\[
-\rho \bar{u}_i' \bar{u}_j' = \mu_t \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k' + \mu_t \frac{\partial u_k}{\partial x_k} \right) \delta_{ij}
\]  

(2.28)

where \( k' \) is the turbulent kinetic energy and \( \mu_t \) is the turbulent viscosity. This approach is known as the Boussinesq approximation, after the French physicist who proposed it in 1868 [4]. Note that the turbulent kinetic energy is conventionally denoted \( k \), but here the prime is included to distinguish \( k' \) from the reduced frequency. There are a number of different turbulence models that model the turbulent viscosity term by introducing additional variables, each governed by their own transport equation. Two popular examples of such models are the Standard \( k-\varepsilon \) model and the Shear Stress Transport (SST) \( k-\omega \) model [4].

The SST \( k-\omega \) model is known to perform better for flows with adverse pressure gradients [28] than the earlier industry standard, the Standard \( k-\varepsilon \) model. It is also insensitive to near-wall grid spacing [29], and uses a method for wall treatment that allows it to more accurately calculate hydrodynamic quantities like lift and drag. It has however been shown to predict larger separation when compared to experiment, e.g. [30], whereas the Standard \( k-\varepsilon \) model is known to under-predict the onset and
magnitude of the separation [4]. Considering the present application it was decided that the SST \( k - \omega \) was most suitable for this work.

2.7.2 The SST \( k - \omega \) model

The SST \( k - \omega \) model introduces two quantities to the RANS equations: the turbulent kinetic energy, \( k' \), and the specific rate of dissipation, \( \omega \). In ANSYS Fluent 15.0.0, the time evolution of \( k' \) and \( \omega \) is given by the two transport equations [31]

\[
\frac{\partial (\rho k')}{\partial t} + \frac{\partial (\rho u_j k')}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma_{k'} \frac{\partial k'}{\partial x_j} \right) + G_{k'} - Y_{k'} + S_{k'} \tag{2.29}
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma_{\omega} \frac{\partial \omega}{\partial x_j} \right) + G_{\omega} - Y_{\omega} + D_{\omega} + S_{\omega} \tag{2.30}
\]

where \( G_{k'} \) and \( G_{\omega} \) are the production terms and \( Y_{k'} \) and \( Y_{\omega} \) are the dissipation terms of \( k' \) and \( \omega \), respectively. \( \Gamma_{k'} \) and \( \Gamma_{\omega} \) are the effective coefficients of diffusivity. The explicit computations of all these terms are shown in [31]. The terms \( S_{k'} \) and \( S_{\omega} \) are user defined source terms. From the local values of \( k' \), \( \omega \) and the mean flow field, the so-called turbulent viscosity is calculated as

\[
\mu_t = \frac{\rho k'}{\omega} \frac{1}{\max \left( \frac{1}{\alpha'}, \frac{S F_2}{\alpha_1 \omega} \right)} \tag{2.31}
\]

where \( S \) is the rate of strain magnitude, \( \alpha' \) is a damping function, \( F_2 \) is a blending function and \( \alpha_1 \) is a model constant. With the SST \( k - \omega \) model it is possible to resolve the entire boundary layer flow, while other models may have to use so-called wall functions that extrapolate the boundary layer from main flow properties. The SST \( k - \omega \) automatically switches between behaving as the standard \( k - \varepsilon \) model near the wall, to behaving as a modified \( k - \varepsilon \) model in the free-stream [31].

2.7.3 Transition modelling

Recently the SST \( k - \omega \) model has begun to be used together with transition modelling, taking into account the possibility of the boundary layer to transition from laminar to turbulent, or the reverse, along the wall. The percentage of time that the flow at a given point is turbulent is called the intermittency, commonly denoted \( \gamma \). In 2011 Delafin et al. showed that the SST \( k - \omega \) model may perform quite differently with and without transition modelling for an aerofoil at moderate Reynolds number and small static angles of attack [28]. A comparison was made of the two approaches for a pitching aerofoil at reduced frequency \( k = 0.18 \) and \( Re = 7.5 \cdot 10^5 \). It was observed that coupling the SST \( k - \omega \) with the two-equation \( \gamma - Re_\vartheta \) transition model produced numerical data in good agreement with experiment at small angles of attack. The fully turbulent SST \( k - \omega \) model failed to predict the unsteady vortex generation and shedding and produced different results than with the transition model for small angles of attack. When the angle of attack was slightly larger, and the boundary layer on the suction side was almost fully turbulent, the two approaches were in agreement. Finally, as the laminar separation bubble grew, the results from the two models differed again. Also, Delafin et al. noticed a smaller hysteresis in the force coefficients for the SST \( k - \omega \) model.

The benefits of using transition modelling comes at the expense of additional computational cost. Langtry estimated that including the two equation \( \gamma - Re_\vartheta \) model requires an additional CPU time of approximately 18% on the same grid. However, he also points out that is also necessary to have \( \gamma^+ \) values no larger than one at the wall and also a high enough streamwise grid resolution to resolve the transition [24]. ANSYS Fluent 15.0.0 supports transition modelling with a one-equation transition model, where the additional equation governs the intermittency. This equation is given by

\[
\frac{\partial (\rho \gamma)}{\partial t} + \frac{\partial (\rho u_j \gamma)}{\partial x_j} = P_\gamma - E_\gamma + \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu_t}{\sigma_\gamma} \right) \frac{\partial \gamma}{\partial x_j} \right] \tag{2.32}
\]
where $P_\gamma$ and $E_\gamma$ are production and destruction terms, respectively, defined in [31]. The intermittency is then included in the calculation of the production and dissipation terms of turbulent kinetic energy in equation (2.29).

### 2.8 3D effects

This study is limited to two-dimensional flows around a rudder section. This means that no spanwise cross-flows are taken into consideration, although such flows are quite pronounced in a real rudder application, due to the helical nature of the propeller slipstream [32]. Another source of cross-flow is the finite length of a real rudder. The 2D section corresponds to a rudder of infinite aspect ratio, but in a real situation the rudder span and mean chord are of similar dimensions. In this finite rudder case a tip vortex will be generated at the deep end of the rudder due to the pressure difference between the starboard and port sides, causing the fluid to accelerate across the rudder tip. This tip vortex induces spanwise flows along the rudder. Another result of the tip vortex is an induced velocity component called downwash which is added to the mean flow, changing the angle of attack as perceived by the rudder to create a smaller effective angle of attack, which may delay stall [1]. However, due to this effect the lift force on the rudder, being perpendicular to the effective inflow velocity, will have a component in the direction of the mean flow. This force, called induced drag, adds to the total drag of the 3D rudder [3].

All of the aforementioned effects are ignored when predicting the rudder performance in 2D, which inevitably means that fully three-dimensional performance prediction will incorporate effects that are lost in this study. The results that are presented in this work should therefore be considered with due caution when conclusions are extrapolated to 3D.
3 Method

With the aim to answer the questions posed in this work a literature survey was performed, yielding large contributions to the previous sections and valuable insights in preparation for the work that was to follow. Simulations were carried out using commercial CFD software and compared to table values and earlier research on the topic. This chapter describes the simulation process.

3.1 Simulations

The steady and unsteady performance of the rudder, represented in 2D by a NACA 0021 profile, was predicted using the CFD software ANSYS Fluent 15.0.0. Both steady and transient simulations were performed, with the aim to be able to quantify the unsteady effects on performance. In the steady simulations, the lift, drag and chord force coefficients were calculated over a range of angles of attack, with and without transition modelling. The transient simulations included studies with respect to a larger set of parameters. The same force coefficients were now computed while independently varying the reduced frequency, the amplitude of the transverse component of velocity, the Reynolds number and the mean angle of attack. During the simulations much effort was put into generating a model and mesh that properly resolved the transverse periodic gust. A large number of different meshes and geometries were evaluated before one was chosen.

3.2 Turbulence model

The SST $k – \omega$ model was chosen for the turbulence modelling, due to its favourable performance for adverse pressure gradients and its ability to resolve the wall flows at the rudder without the use of wall functions. This method was expected to yield more accurate predictions of hydrodynamic quantities than, for instance, the Standard $k – \varepsilon$ method with wall functions. The latter model has also been known to predict separation poorly [5], which was of large importance in this study. For some of the simulations to be presented, the one-equation intermittency transition model was activated for comparison.

3.3 Fluid properties

In all simulations the physical properties of the fluid were taken from the standard values for liquid water in the material library of ANSYS Fluent 15.0.0. The density was $\rho = 998.2 \text{ kg m}^{-3}$ and viscosity $\mu = 1.003 \cdot 10^{-3} \text{ kg m}^{-1} \text{s}^{-1}$.

3.4 Geometry

The 2D simulations were performed with the rudder geometry given by the NACA 0021 profile, as seen in Figure 5. The coordinates of this shape are given in appendix B [33]. This section has a maximum thickness to chord length ratio of 0.21 and no camber. The symmetric NACA 0021 can be considered representative of a conventional ship rudder. The chord length of the rudder was 0.15 m, representing a typical model-scale rudder chord length.

In order to deduce whether the results were highly dependent on the rudder geometry, a subset of the transient simulations were also carried out for the Shilling rudder section which can be seen in Figure 6. This is in line with the work of James C. Date [5] and thereby increases the possibility to validate the results. Similar settings were used when generating the mesh for both rudders, although the free-stream mesh size was finer in the case of the Shilling rudder. The settings for the NACA 0021 are presented in subsection 3.5.
The boundaries of the rectangular computational domain were placed at a distance of 7 chord lengths downstream of the rudder, and 6.7 chords in the transverse direction to the mean flow as seen in Figure 7. From the data presented in Table 1, one might argue that the solution is not entirely independent of the transverse placement of the boundaries, but the error was considered acceptable when taking into account the heavily amplified computational cost of increasing the domain size in the transverse direction. It was also expected that any influence on the rudder forces from the nearby wall would have no significant impact in the qualitative behaviour of the rudder, with respect to the control parameters. Therefore, this placement of the walls was deemed sufficiently distant from the rudder to have acceptable influence on the results. An increase in the downstream direction was less costly due to the coarser mesh size in the far wake region, as described in section 3.6.1, and from Table 1 the solution appears to be insensitive to the placement of the downstream boundary.
3.5 Boundary conditions

This section describes the boundary conditions that were applied in ANSYS Fluent to the geometry previously described.

3.5.1 Inlet boundary condition

It was desired to apply a velocity inlet boundary condition which generated a mean flow component and a transverse periodic gust. Generalizing the flows applied by Date [1], as well as by Paterson and Stern [13] to a mean flow of arbitrary angle of incidence, such a flow field is described by

\[ u = U_m \cos \alpha_m + u_g \sin \alpha_m \sin \left( \frac{\omega}{U_m} x \cos \alpha_m + \frac{\omega}{U_m} y \sin \alpha_m - \omega t \right) \]  
\[ v = U_m \sin \alpha_m - u_g \cos \alpha_m \sin \left( \frac{\omega}{U_m} x \cos \alpha_m + \frac{\omega}{U_m} y \sin \alpha_m - \omega t \right) \]  

\[ 3.1 \]
\[ 3.2 \]
where $\alpha_m$ is an arbitrary angle of incidence of the mean flow, $U_m$ is the mean flow velocity and $u_g$ is the magnitude of the transverse gust. The angular frequency $\omega$ is related to the reduced frequency $k$ through equation (2.10). A full derivation of this inlet boundary condition is given in appendix C.

However, for the transient simulations performed in the present study, a non-zero mean angle of attack was created by redefining the geometry and mesh with the rudder at an angle, rather than through adjusting $\alpha_m$ in the boundary condition. In other words, the angle $\alpha_m$ was given by the mesh, and the boundary condition that was used for the transient simulation simplified to

\[
\begin{align*}
    u &= U_m \\
    v &= u_g \sin(\omega t). 
\end{align*}
\]

(3.3) 
(3.4)

By inclining the rudder in the geometry, a mesh was produced that was denser in the direction of the wake at the trailing edge of the rudder, which was considered beneficial. The boundary condition in equations (3.1)-(3.2) does however still describe the field as it was perceived by the rudder, and is therefore given as reference. It was also used for some of the steady simulations, while setting $u_g = 0$, $\omega = 0$, $\alpha_m \neq 0$. This is further explained in section 3.10.

The default settings for the turbulent properties were used at the inlet. The turbulent intensity was $1\%$ and the turbulent viscosity ratio was $10$. These settings represent typical outer flow values, and no other reference data restricted the choice.

3.5.2 Rudder

A no-slip boundary condition ($\bar{u} = 0$) was applied to the rudder. The default roughness height of 0 m and roughness constant 0.5 were used. These values represent a hydrodynamically smooth wall, i.e. a surface at which any roughness is entirely contained within the laminar sublayer [1].

3.5.3 Far field boundary conditions

A pressure outlet boundary condition ($p = 0$ bar) was applied to the downstream boundary of the computational domain. The default settings for backflow direction and turbulent properties were used, since no backflow was expected during the simulation.

A no-slip boundary condition was implemented on the domain walls parallel to the mean flow. Although this type of boundary condition does not correspond to a true open-water simulation, it has the benefit of fast convergence. Any boundary effects were instead avoided by placing the boundaries sufficiently far from the rudder, as described in section 3.4.

3.6 Mesh

A few different meshes were used in the simulations, one for each mean angle of attack during the transient simulations. All meshes were produced with ANSYS Meshing 15.0.0. The same mesh settings were applied to all cases, except for some fine tuning to produce meshes of high quality. Due to large artificial diffusion of momentum associated with the gradients of the periodic flow, a significantly higher overall mesh resolution was required than what is typically necessary for 2D simulations of this kind. The grid independence study presented in this section corresponds to the NACA 0021. For the Shilling rudder similar settings were used, although a finer free stream mesh size was applied for practical reasons.

3.6.1 Main flow

The main flow region was split into two parts; one domain containing the rudder and another domain for the far wake as shown in Figure 7. The latter region was allowed to contain cells of larger size, while the mesh in the former region was refined using a Face Sizing condition. Both regions were meshed using the Uniform setting of the MultiZone method in ANSYS Meshing, yielding a mesh of mostly quadrilateral cells. The cell size was chosen such that the number of cells per wavelength of the periodic gust always was at least 10 to 50 in order to control the numerical diffusion. This effect was very strong for the largest reduced frequencies even at this free stream resolution. This approach
did however lead to a relatively high cell count when compared to typical aerofoil simulations in a steady inflow.

The chosen mesh had to have a balance between numerical diffusion and cell count. An excerpt from the process of selecting mesh properties at a given set of flow conditions is shown in Table 2. Comparing the two first entries, it can be seen that a 60% increase in cell count only led to a 0.9% change in the computed mean drag coefficient. From this it was concluded that the increased cost of further mesh refinement outweighed the benefit in gained accuracy, and so the overall element size of 0.005 m was chosen.

Table 2: Mean drag coefficient at \( \alpha_m = 0 \), \( dt = 2 \cdot 10^{-4} \), \( k = 0.785 \) and \( u_g = 1.25 \text{ m/s} \) for different levels of mesh refinement.

<table>
<thead>
<tr>
<th>Mesh properties</th>
<th>\langle c_d \rangle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cell count: 131 000</td>
<td>-0.0224</td>
</tr>
<tr>
<td>Element size 0.005 m</td>
<td></td>
</tr>
<tr>
<td>Number of inflation layers: 20</td>
<td></td>
</tr>
<tr>
<td>First layer height: 6 ( \cdot 10^{-6} )m</td>
<td></td>
</tr>
<tr>
<td>Estimated ( y^+ ): 1.5</td>
<td></td>
</tr>
<tr>
<td>Cell count: 210 000</td>
<td>-0.0222</td>
</tr>
<tr>
<td>Element size 0.004 m</td>
<td></td>
</tr>
<tr>
<td>Number of inflation layers: 20</td>
<td></td>
</tr>
<tr>
<td>First layer height: 4 ( \cdot 10^{-6} )m</td>
<td></td>
</tr>
<tr>
<td>Estimated ( y^+ ): 1.0</td>
<td></td>
</tr>
<tr>
<td>Cell count: 132 000</td>
<td>-0.0220</td>
</tr>
<tr>
<td>Element size 0.005 m</td>
<td></td>
</tr>
<tr>
<td>Number of inflation layers: 25</td>
<td></td>
</tr>
<tr>
<td>First layer height: 6 ( \cdot 10^{-6} )m</td>
<td></td>
</tr>
<tr>
<td>Estimated ( y^+ ): 1.5</td>
<td></td>
</tr>
<tr>
<td>Cell count: 161 000</td>
<td>-0.0205</td>
</tr>
<tr>
<td>Element size 0.005 m</td>
<td></td>
</tr>
<tr>
<td>Number of inflation layers: 45</td>
<td></td>
</tr>
<tr>
<td>First layer height: 1 ( \cdot 10^{-6} )m</td>
<td></td>
</tr>
<tr>
<td>Estimated ( y^+ ): 0.25</td>
<td></td>
</tr>
<tr>
<td>Cell count: 164 000</td>
<td>-0.0207</td>
</tr>
<tr>
<td>Element size 0.005 m</td>
<td></td>
</tr>
<tr>
<td>Number of inflation layers: 48</td>
<td></td>
</tr>
<tr>
<td>First layer height: 2 ( \cdot 10^{-7} )m</td>
<td></td>
</tr>
<tr>
<td>Estimated ( y^+ ): 0.05</td>
<td></td>
</tr>
</tbody>
</table>

3.6.2 Rudder

The accuracy of the hydrodynamic forces on the rudder was of highest priority in the present study. For this reason the choice was made to resolve the entire boundary layer with prism cells, generated using the Inflation option in ANSYS Meshing. The height of the first layer was controlled to yield a dimensionless wall distance smaller than \( y^+ = 1 \) in the first cell. This was done to ensure that the viscous sublayer was resolved. Using equations (2.16) and (2.17), it was estimated that \( y = 4 \cdot 10^{-6} \) m should yield \( y^+ = 1 \), and so the height of the first layer was set as \( y = 2 \cdot 10^{-7} \) m to be safe. The growth rate of the prism layers was set to 1.2, which is the default value of the growth rate in ANSYS Meshing.

The dependency of the rudder forces on the near-wall grid resolution was evaluated by gradually increasing the number of inflation layers and decreasing the first layer height. From the last three entries in Table 2 it can be concluded that the solution was fairly insensitive to further refinement of
the mesh in the boundary layer. Therefore the last entry in the table was accepted, although the mean drag was still slightly changing with the parameters. In order to limit the aspect ratio of the prismatic cells and increase the spatial resolution near the leading and trailing edges of the rudder, the Bias option of the Inflation function was used on both sides of the rudder. The bias factor was set to 10, meaning that the element sizes near the trailing and leading edges were 10 times smaller than at the mid-chord. This made it possible to achieve better resolution in the wake region, where the vorticity was expected to be strong, without generating needless cells in other parts of the near-body region.

Special care had to be taken when generating the mesh in order to have a smooth behaviour at the trailing edge of the rudder, where the sharp corners easily lead to inflation layer cells of very poor quality. The mesh in Figure 8 shows an acceptable behaviour at the trailing edge.

The meshing method for the main flow automatically generated transitional cells between the inflation layers and the main flow as shown in Figure 9. This created an effective body of influence, centred at the rudder, which helped to resolve the wake in the near-rudder region. The resolution of the wake is important for the accuracy of the computed drag coefficient [34], and the combination of the transitional cells and the high overall resolution of the main flow was considered sufficient to resolve the wake behind the rudder. In order to keep the cell count down, no further refinements of the wake region were made.

Figure 8: The mesh around trailing edge of rudder.

Figure 9: The higher cell density in the near body region between inflation layers and main flow.
Far field boundaries

Neither the flow at the far field boundaries nor the forces on the domain walls were in themselves interesting for the present study. Therefore no inflation layers were added at these boundaries, and the $y^+$ values ranged from 350 to 600, which is well outside the transition from the viscous sublayer to the mixing layer. This allowed the SST $k-\omega$ turbulence model to treat the wall flows with wall functions instead of resolving the entire boundary layer. The unconventionally large $y^+$ was justified since the details of the flow at the domain walls were of no interest in the present study, as long as it did not affect the flow around the rudder. The mesh at the solid walls is shown in Figure 10.

![Figure 10: The mesh at the domain boundary.](image)

Solver settings

The default solution methods in ANSYS Fluent 15.0.0 for the SST $k-\omega$ model were used. These included a coupled scheme for pressure-velocity coupling, a least squares cell based scheme for the handling of gradients and a first order implicit scheme for the transient formulation. The momentum terms were discretized with a second order upwind scheme, while the turbulence equations were solved with a first order upwind scheme. The convergences of all residuals were monitored and a fixed number of 25 iterations per time step was chosen based on their behaviour.

Reference frame

The lift and drag on the rudder are defined as the forces perpendicular and parallel, respectively, to the free-stream velocity field. In this study, the force coefficients were calculated with respect to the mean flow rather than the instantaneous direction of the inflow. The most relevant force coefficient for evaluating the performance of a rudder when the rudder is at zero incidence to the direction of advance is the chord force coefficient. The net chord force is in this case the only contribution to the useful thrust force and the cause of any energy savings or losses. For design purposes of twisted rudders, where the rudder chord is not always aligned with the direction of advance of the ship, the force coefficient perpendicular to the chord is also of importance, as seen in Figure 11. The side force coefficient will also be presented in the results.

![Figure 11: The relationships between chord force, side force and useful thrust at a twist angle $\beta$.](image)
3.9 Time step

For the transient simulations a time step size of $dt = 0.0002 \, s$ was used. At this time step size and with the chosen mesh, the Courant number was on the order of $C = 0.2$ in the main flow region. By systematically decreasing the time step size and performing the same simulations, a convergence study with respect to $dt$ was performed. The results shown in Table 3 seem to imply that the solution was not yet fully converged with respect to the time step size at the chosen $dt$, since the mean drag coefficient was still decreasing almost linearly with decreasing time step size. However, since a doubling of the time step size resulted in a change in the results of about 3% and the scheme for the transient formulation was first order in time\(^2\), the error could be estimated to be on the order of 1-2%. The chosen $dt$ was therefore taken as a compromise between accuracy and computing cost.

Furthermore, since the temporal resolution of the transient gusts was at least 72 time steps per period for all simulations, any temporal under-resolution ought to be of the wake, which should be equally pronounced for all simulations. Thus the modelling error due to an under-converged time step should not have resulted in any significant biasing of the results. Figure 12 shows the results of the simulation versus the number of iterations per period. From this graph it is clear that any gain in accuracy comes at an ever-increasing cost as the number of iterations per period increases.

Table 3: Refinement analysis with respect to time step for $\alpha_m = 0, \, k = 0.785$ and $u_g = 1.25 \, m/s$.

<table>
<thead>
<tr>
<th>Time step</th>
<th>Free stream $C$</th>
<th>$\langle c_d \rangle$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$1.2 \cdot 10^{-3} s$</td>
<td>1.2</td>
<td>-0.0155</td>
</tr>
<tr>
<td>$4 \cdot 10^{-4} s$</td>
<td>0.4</td>
<td>-0.0196</td>
</tr>
<tr>
<td>$2 \cdot 10^{-4} s$</td>
<td>0.2</td>
<td>-0.0207</td>
</tr>
<tr>
<td>$1.5 \cdot 10^{-4} s$</td>
<td>0.15</td>
<td>-0.0210</td>
</tr>
<tr>
<td>$1 \cdot 10^{-4} s$</td>
<td>0.1</td>
<td>-0.0213</td>
</tr>
</tbody>
</table>

*Figure 12: The mean drag coefficient versus the number of iterations per period.*

---

\(^2\) The discretization error of a first order scheme is to leading order proportional to the step size.
3.10 Test matrix and procedure

Three different sets of steady simulations were carried out. For the first case the steady performance of the NACA 0021 was evaluated by computing $c_l$, $c_d$ and $c_v$, where the rudder was inclined at the angles $\alpha_m = 0^\circ, 10^\circ$ and $15^\circ$ in the mesh. The results of this data series were used for comparison with the transient simulations. In a second case, two more data series were calculated for angles up to and slightly beyond the static stall angle using the same mesh as the transient simulations at zero angle of attack. The angle was for practical reasons instead generated by changing $\alpha_m$ in the inlet boundary conditions (3.1)-(3.2). These two series were used to compare the results with and without transition modelling. All computations, both transient and steady, were performed with an inlet velocity of $U_m = 5.0 \text{ m/s}$, yielding a chord-based Reynolds number of $Re = 7.5 \cdot 10^5$.

For the transient simulations, a larger number of control parameters were considered. For the mean angles of attack $\alpha_m = 0^\circ, 10^\circ$ and $15^\circ$, and at $Re = 7.5 \cdot 10^5$, the effects of varying the gust amplitude and reduced frequency were independently investigated. Simulations at $15^\circ$ were performed with the SST $k-\omega$ model alone, as well as with the $\gamma$ transition model activated. At the angles $\alpha_m = 0^\circ$ and $15^\circ$, additional simulations were carried out at $Re = 7.5 \cdot 10^6$ for comparison. The increased Reynolds number was achieved by decreasing the viscosity with a factor of 10.

One simulation was carried out with the periodic gust being composed of its fundamental frequency, $k$, and the second harmonic, $2k$. The time history of the angle of attack thereby resembled a more realistic signal in a propeller slipstream. This was done to shed some light on the importance of resolving the full waveform of the transient inflow. For the Shilling rudder, independent parametric studies with respect to $k$ and $\alpha_m$ were carried out at $\alpha_m = 0^\circ$.

All transient simulations suffered from some degree of numerical damping, due to the transient gust being spatially under-resolved. In order to limit the biasing effect this had on the results, the gust amplitude was calculated at a point along the line tangent to the leading edge and perpendicular to the mean velocity. For all results presented in this report, $u_g$ was thus the gust amplitude computed at the leading edge of the rudder, as seen in Figure 13. The actual value of $u_g$ varied slightly between consecutive data points in a data series, but was no further than 5% from the stated value of the corresponding data series. It should however be noted that the shear stresses in the sinusoidal field are proportional to $\omega^2 u_g/U_m \sim k^2 u_g$. Therefore even a physically accurate solution will be biased due to viscosity, yielding smaller amplitudes at the trailing edge for gusts of higher frequency.

![Figure 13: The geometric definitions of $u_g$ and $\alpha_m$.](image-url)
4 Results

The results of the simulations are presented in two sections; steady and transient simulations. Experimental data and results from simulations of other authors have been included for comparison in some of the presented figures. The overall agreement with previous results was good, but direct comparison is difficult due to the large number of parameters that affect the numerical values of the results. The effect and importance of some of these parameters are evaluated in the following sections. The values of $y^+$ in the first cells at the rudder never exceeded a certain value. Surface plots of $y^+$ for a few different simulations are shown in appendix D.

4.1 Steady simulations

The computed steady lift coefficient of the NACA 0021 is shown versus angle of attack in Figure 14. The first case, denoted Case 1 in the legend in the figure, corresponds to the data points that will be used for comparison with the transient results. For these results the angle was generated directly in the geometry. The second case compares the results with and without transition modelling, where the angle was generated by changing $\alpha_m$ in the boundary condition. Experimental data by Gregorek et al. in 1989, reproduced in [35], is inserted for comparison. This experimental data comes from wind tunnel experiments at $Re = 1.5 \cdot 10^6$. It is clear that the simulation with transition modelling predicted a larger stall angle and maximum lift coefficient than the SST $k-\omega$ model did, even with identical turbulence properties at the inlet. For angles larger than 18 degrees, the latter model did not converge towards a steady value of the force coefficients, which indicated that the deep stall regime had been reached.

The SST $k-\omega$ model predicted a stall angle of around 16 degrees at the conditions $Re = 7.5 \cdot 10^5$ and a turbulent intensity of 5% at the inlet. This is slightly below the stall angle that was experimentally found by Swalwell et al. for the NACA 0021 at 4% turbulent intensity and $Re = 3.5 \cdot 10^5$ [8]. The stall angle is according to Swalwell expected to increase with Reynolds number and turbulent intensity. The smaller values of $c_l$ and the earlier onset of stall found in the steady simulations are therefore most likely due to the dissipation of turbulence between the inlet and rudder. The lift curve shows good agreement with Gregorek et al. and the computed values lie between the measurements made by Swalwell et al. and those of Sheldahl and Klimas [26], not shown here.

![Figure 14: Steady lift coefficient versus angle of attack for the NACA 0021.](image-url)
The steady drag coefficient is similarly shown versus angle of attack in Figure 15, and the chord force computed from the lift and drag coefficients is shown in Figure 16. Agreement with Gregorek et al. is good, except for at 12 degrees angle of attack. This is however believed to be due to differences in turbulent intensity or error in the reference data, as the simulation data agrees well with the trend of Swalwell et al. The agreement between experiment and the steady case data increases the trustworthiness of the results from both the steady and transient simulations.

Figure 15: Steady drag coefficient versus angle of attack for the NACA 0021.

Figure 16: Steady chord force coefficient versus angle of attack for the NACA 0021.
4.2 Transient simulations

The results presented in this section were produced with the SST $k - \omega$ model without transition modelling, except for where it is explicitly stated otherwise. Comparison and discussion of data sets are carried out with respect to the flow parameters and the use of the transition model. Any steady data referenced in this section corresponds to Case 1 in the results presented in the previous section.

4.2.1 Effect of the reduced frequency and gust amplitude at zero mean angle of attack

The force coefficients on the NACA 0021 exhibited periodic response to the unsteady gust. Figure 17 shows the computed mean chord force coefficient versus reduced frequency for a few different gust amplitudes. All data points were gathered with the rudder at zero mean angle of attack, which implies that the chord force coefficient is also equivalent to the drag coefficient. Although not shown here, the power spectral density of the signals in force coefficients indicated strong first order response, i.e. at the frequency of the gust. The highest frequency data points were produced with a rudder that was scaled up by a factor of 2, as indicated by the legend in the figure. Due to this, the Reynolds number was twice as large for these simulations, which is expected to increase the maximum lift coefficient and delay stall. This explains the sudden shift in computed chord force towards a more favourable, lower value.

It can be noted that the mean chord force coefficient is increasing monotonically with increasing $k$. A significant thrust force is generated by the rudder in the case of large amplitude, low frequency oscillations. This agrees well with the earlier results presented by James C. Date [5], which are included in the figure for comparison, and the simulations of Paterson and Stern [13]. The reference values by Date were computed with a mean velocity of $10 \text{ m/s}$ and a periodic gust amplitude of $2.5 \text{ m/s}$, making the angle of attack history comparable to the data set of $u_g = 1.05 \text{ m/s}$, although the former simulations were carried out at a significantly larger Reynolds number. It is also noteworthy that the mean drag coefficient for the small amplitude oscillations approaches the value of the steady drag coefficient, and appears to exceed it towards a possibly stable, slightly larger value. Date found the same behaviour for small amplitude oscillations in the limit of high frequency and $\alpha_m = 0$.

The large amplitude gust data series intersects that of the smaller amplitude gusts. It is from this data not clear whether the chord force coefficient approaches a limiting value in the limit of high frequency, but it is the belief of the author that this would be the case if such simulations were carried out. It can however be stated that the favourable negative chord force coefficients that are associated with the large amplitude gusts at low frequency are not found at high frequency. Instead, the mean drag appears to increase with gust amplitude at high frequency, in the case of zero mean angle of attack.

The results indicate a trend of decreasing chord force coefficient towards the limit of $k \rightarrow 0$, i.e. the quasi-steady regime. It is expected by the author that all curves should approach respective values that correspond to the mean values of the force coefficient, as calculated from the steady lift and drag at each instantaneous angle of attack. That is, by calculating the time history of the angle of attack and finding the corresponding steady force coefficients at each instant, one should find the limiting value of the data series shown in Figure 17, in the limit of $k \rightarrow 0$. However, in the cases where stall is likely to occur, i.e. for very large gust amplitudes, the trend would probably look different in the limit of small $k$. This is nevertheless well outside the operating range of a rudder in a propeller slipstream and of little consequence for this work.
4.2.2 Effect of the mean angle of attack on the chord force coefficient

The chord force coefficient data from simulations at a mean angle of attack of $10^\circ$ is shown in Figure 18. It is evident that the point of intersection of the large and small amplitude data series has moved towards the low frequency range, in comparison to the same data sets in the case of $\alpha_m = 0^\circ$. The point of intersection between the data series for $u_g = 0.35 \text{ m/s}$ and $u_g = 1.05 \text{ m/s}$ lied within $k \in (7, 8)$ at $\alpha_m = 0^\circ$ and within $k \in (3.5, 5)$ at $10^\circ$. The mean chord force coefficient appears to be fairly insensitive to $k$ within the considered range for the small amplitude oscillations. For the gusts of larger amplitude, the mean chord force coefficient shows clear dependency with increasing frequency, and has not yet converged towards a steady value within the presented range. Computations at higher $k$ would be necessary to determine the limiting behaviour.

For both the large and small amplitude oscillations, the rudder is producing significant thrust in the presented range of frequencies. This is also predicted by the steady simulation, which is included in Figure 18. The thrust force is attributed to the increase in mean lift coefficient that is gained by placing the rudder at $10^\circ$ with respect to the mean inflow. In the practical application of a rudder in a propeller slipstream, this implies that the spanwise cross sections subjected to a non-zero mean angle of attack may produce a larger thrust force than those where the rudder is aligned parallel to the inflow. When comparing the transient and steady results, it is evident that modelling the fully transient performance with a steady simulation would yield an under- or over-prediction, depending on the flow parameters of the real transient case. For $k$ larger than 2 and a gust amplitude of $u_g = 0.35 \text{ m/s}$ or smaller, the agreement between the transient and steady predictions is very good, differing only within 1%. However, for $u_g = 1.05 \text{ m/s}$ the steady result is a 3% over-prediction of the real transient thrust, and the disagreement appears to be growing with $k$. For all amplitudes, it holds that $\partial \langle c_l \rangle / \partial k > 0$ throughout the investigated range of frequencies. It seems plausible, if one extrapolates from the presented data that the disagreement may be converging towards a limiting value with increasing $k$. If so, the error introduced when predicting the performance in a steady state would likely be significant (greater than 3%) for a slipstream of a typical propeller operating at $k \approx 10$. 

**Figure 17**: Mean chord force coefficient versus reduced frequency at $\alpha_m = 0$. Data sets with an asterisk were produced with a rudder of double thickness and chord length.
When the rudder was placed at $\alpha_m = 15^\circ$, a series of vortices was observed to be shed from the leading edge and convected along the cord by the mean flow. The vortices, which were shed at blade frequency were generated at the leading edge as the angle of attack reached its maximum, and released as the angle of attack was decreasing. Their orientation agreed with the instantaneous change in rudder lift. That is to say, when the section lift was decreasing, a vortex of negative circulation was shed in agreement with the Kelvin theorem of circulation [2]. This series of starting vortices carried with them low pressure zones which in turn influenced the flow near the rudder, as can be seen from the figures in appendix E. This behaviour was less pronounced for low $k$, and not observed at $\alpha_m = 0^\circ$. It is unknown to the author how large influence these vortices had on the total force acting on the rudder.

The predicted performance at $\alpha_m = 15^\circ$ can be seen in Figure 19, where data series from simulations using transition modelling is also included for future reference. Again we find that the predicted thrust of the rudder decreases with increasing $k$ for the large amplitude gusts. In this case the dependency on reduced frequency seemed to be stronger than in the $10^\circ$ case, as the mean chord force coefficient varied more over the presented range. The performance of the rudder submitted to large amplitude gusts varied with $26\%$ between $k = 1$ and $k = 5$. The corresponding variation was only $16\%$ when the rudder was at $10^\circ$.

The performance depended non-monotonically on $k$ for the gust amplitude $u_g = 0.35 \text{ m/s}$. For the large amplitude oscillations, the chord force was monotonically increasing with $k$. This behaviour is similar to what was observed for the large amplitude oscillations in the $10^\circ$ case, although in the present case the curvature is not negative for all values of $k$. There exists an inflexion point somewhere between $k = 2$ and $k = 3.5$ for $u_g = 1.05 \text{ m/s}$, which may indicate that several dynamical effects affecting the performance are $k$-dependent. These effects may for instance be a combination of increased pressure drag as a result of separation, and possibly a stronger interaction between the boundary layer and the shed vorticity from the leading edge. In the cases where the transient prediction is poorer than the steady result, it is most likely due to the transient angle of attack exceeding the stall angle of the rudder, leading to a loss of lift and thereby loss of thrust in the chordwise direction. A significantly larger separated region was observed at $\alpha_m = 15^\circ$ than at $\alpha_m = 10^\circ$ for the large amplitude gusts at $k = 5$, which strengthens this hypothesis. Streamlines of the two cases are shown in Figure F1 and Figure F2 in appendix F.
The results presented here seem to imply that the gain in chord force by placing the rudder section at an angle to the propeller slipstream depends on the reduced frequency at which the propeller operates, as well as the amplitude of the variations in velocity angle it produces. This effect would of course also have to be weighted with the respective misalignment of the chord to the direction of advance before an optimal rudder design could be found. In other words, the increase in thrust by twisting the rudder to change the angle of attack may be outweighed by the misalignment of the rudder chord to the heading of the ship, leading to a net loss of useful thrust.

Predicting the rudder performance with a steady state simulation will lead to different results than if a fully transient simulation is used, which can be seen from Figure 19. It can also be seen that if the result of a steady simulation at $\alpha_m = 15^\circ$ is used to model the transient performance, this will be an under- or over-prediction of the fully transient result, depending on the reduced frequency in a similar way as at $\alpha_m = 10^\circ$. The steady simulation over-predicts the performance with 20% compared to the fully transient calculations at $u_g = 1.05 \text{ m/s}$ and $k = 5$. The steady result is instead a 12% under-prediction if used to model the transient results at $u_g = 1.05 \text{ m/s}$ and $k = 1$. Table 4 shows the relative over- or under-prediction of the rudder performance for two values of the gust amplitude. From this data, it is clear that a steady simulation is more likely to over-predict the performance when the mean angle of attack and/or the gust amplitude is large. Similar to the 10° case, the limiting behaviour with increasing $k$ cannot be determined from the presented data, but the disagreement is increasing towards the high-$k$ end of the range. The computed mean lift and drag coefficients that were used to calculate the chord force coefficients for $\alpha_m = 0^\circ$, 10° and 15° are given in table form in appendix G.

Table 4: Relative disagreement of the performance predicted by the steady simulations compared to the results of transient simulations with $k = 5$ and $u_g = 1.05 \text{ m/s}$. The discrepancy is given by $E = -(c_{c,\text{steady}} - c_{c,\text{transient}})/c_{c,\text{transient}}$.

<table>
<thead>
<tr>
<th>$\alpha_m$</th>
<th>$u_g = 0.35 \text{ m/s}$</th>
<th>$u_g = 1.05 \text{ m/s}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>-1%</td>
<td>-6%</td>
</tr>
<tr>
<td>10°</td>
<td>0.4%</td>
<td>3%</td>
</tr>
<tr>
<td>15°</td>
<td>2%</td>
<td>20%</td>
</tr>
</tbody>
</table>

Figure 19: Mean chord force coefficient versus reduced frequency at $\alpha_m = 15^\circ$. 

1

<table>
<thead>
<tr>
<th>$u_g = 0.10 \text{ m/s}$, SST k-ω</th>
<th>$u_g = 0.35 \text{ m/s}$, SST k-ω</th>
<th>$u_g = 1.05 \text{ m/s}$, SST k-ω</th>
<th>$u_g = 0.35 \text{ m/s}$, TM</th>
<th>$u_g = 1.05 \text{ m/s}$, TM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steady $c_c$ at 15°</td>
<td>Steady $c_c$ at 15°</td>
<td>Steady $c_c$ at 15°</td>
<td>Steady $c_c$ at 15°</td>
<td>Steady $c_c$ at 15°</td>
</tr>
</tbody>
</table>
The time histories of the chord force coefficient in the case of $\alpha_m = 15^\circ$ are shown for small and large amplitude oscillations in Figure 20 and Figure 21, respectively. From these histories, which are normalized along the time axis to simplify comparison, the $k$-dependency of the mean chord force coefficients can be further explained. It is evident that increasing the reduced frequency dampens the amplitude of the responses in force coefficients, while the mean value of the chord force coefficient (the offset of the periodic signal) varies non-monotonically.

Although not shown in the figure, the signal of the chord force coefficient is shifted nearly 180 degrees in phase with the angle of attack at the leading edge. That is, for the largest angles of attack, the smallest (most negative) chord force coefficients are found. This is understood from the large region of negative slope found in the steady chord force coefficient plot (see Figure 16). Thus the kink in the $k = 1$ data series in Figure 21 occurs at the smallest angle of attack, corresponding to approximately $2^\circ$ in the free-stream. In other words, this curious effect is not due to the rudder being close to stalled conditions, but more likely due to sudden reattachment of the separated layer or relocation of the suction side stagnation point.

Figure 20: Chord force coefficient versus normalized time. $u_g = 0.35 \text{ m/s, } \alpha_m = 15^\circ$.

Figure 21: Chord force coefficient versus normalized time. $u_g = 1.05 \text{ m/s, } \alpha_m = 15^\circ$. 
4.2.3 Effect of the mean angle of attack on the side force coefficient

The side force coefficients at $\alpha_m = 10^\circ$ and $15^\circ$, shown in Figure 22 and Figure 23 respectively, exhibited more erratic behaviour compared to the chord force coefficients at the same angles of attack. The data is far from converged with respect to $k$ at the upper end of the presented range. However, the relative variations at $10^\circ$ are significantly smaller for $c_s$ than for $c_c$. The former had variations of no more than 4% over the presented range, which is small compared to the 16% variations in $c_c$. The steady result is a fair estimate of the side force at $10^\circ$ but less so at $15^\circ$, where considerably larger discrepancies between steady and transient results were found.

Still, it should be noted that the main contribution to the rudder performance is from the chord force, unless the twist is very large. As an example, consider a rudder submitted to an inflow at $\alpha_m = 15^\circ$ with respect to the direction of advance. Assume that the twist is $5^\circ$, making the relative angle of attack $10^\circ$. In this case, the force coefficient in the direction of advance is made up of the two components $c_c|_{\alpha_m=15^\circ} \cos 5^\circ$ and $c_s|_{\alpha_m=15^\circ} \sin 5^\circ$. Let the reduced frequency be $k = 5$. From Figure 22 and Figure 18 we find that the first term is roughly 50% larger in magnitude than the second, implying that the accuracy of the chord force coefficient is more important for the total performance prediction.

Figure 22: The side force coefficient at $\alpha_m = 10^\circ$. 
4.2.4 Effect of transition modelling and Reynolds number

The results of the simulations for one of the gust amplitudes at $\alpha_m = 0^\circ$ and at two Reynolds numbers are included in Figure 24, with and without transition modelling. All other model parameters and flow conditions were kept identical at the two Reynolds numbers to show the isolated impacts of either using the transition model or changing the Reynolds number. The predicted performance was improved by including transition modelling. A laminar separation bubble was observed, centred near the mid-chord. The LSB largely improved the performance of the rudder by decreasing viscous stress, which was roughly 40\% lower with the transition model activated. This corresponded to approximately 35\% of the total decrease in chord force coefficient, the rest being due to reduced pressure drag. It is interesting to note that the disagreement between the two approaches is almost negligible at higher Reynolds number, as seen in Figure 24. This is understandable since the transition model is expected to be active to a lesser extent (due to earlier transition) at higher Reynolds number, and the contribution of viscous stress to the total drag is expected to be smaller. Also, the LSB is expected to reduce in size with increasing Reynolds number [25]. Thus the benefit of including the additional equation is lost at the larger Reynolds number, where the fully turbulent model becomes a good approximation.

The results of the simulations at $\alpha_m = 15^\circ$, with and without the transition model are shown in Figure 19. The predicted mean chord force coefficients for both small and large amplitude gusts are significantly decreased by activating the transition model, as they were at $\alpha_m = 0^\circ$, but in this case the exact shapes of the curves are not preserved. This indicates that the transition model resolves or emulates additional physics that are not taken into account by the SST $k-\omega$ model alone. Looking back at the steady results shown in Figure 14, we may understand that the larger predicted thrust was due to the overall larger lift coefficients and the later onset of stall that were predicted by the transition model, as compared to the SST $k-\omega$ model. The LSB had now moved towards the leading edge of the suction side, where it acted as an effective camber that may have improved the performance of the rudder [25].

There was also a significant decrease in wall shear stress due to parts of the boundary layer being laminar. The viscous stress in the direction of the chord was decreased by approximately 35\%, which technically contributed to the improved performance, but at this mean angle of attack the pressure forces dominated over wall shear stress by several orders of magnitude. Therefore the main effect of the LSB on the results was through a favourable thrust due to delayed separation. Figures F3 and F4...
in appendix F show the extent of the separation with and without transition modelling. The laminar separation bubble can be seen in Figure 25 as a thin region of flow reversal and low velocities near the wall. Note the sudden increase in wall shear stress downstream of the transition to turbulent boundary layer. The moderate values of wall stress within the bubble correspond to forces acting in the opposite direction of the free-stream due to the reversed flow in the LSB. The transition of the pressure side boundary layer occurred at the trailing edge, as can be seen from the intermittency plot in Figure 26. This was considered quite late since the influence from the fully turbulent free-stream flow potentially could trigger bypass transition earlier along the rudder chord [24], but the favourable pressure gradient is likely to have stabilized the boundary layer and delayed transition. When \( u_g \) was increased the transition occurred earlier, which shows that the free-stream oscillations indeed could influence the location of the transition.

From Figure 24 it can also be discerned that the performance of the rudder is favourably affected by the increase in Reynolds number, when computed with the SST \( k-\omega \) model. By increasing the Reynolds number with a factor of 10, the chord force coefficient decreased by around 40 – 50% depending on \( k \). This was most likely due to an increased lift coefficient [8] [25] and decreased pressure drag on the rudder [27]. This also explains the discontinuity at the first data point where the rudder of double size was used in the data series in Figure 17. It is less obvious why the transition model predicted slightly poorer rudder performance at higher Reynolds number, although the difference might not be significant.

![Figure 24: Mean chord force coefficient versus reduced frequency at \( \alpha_m = 0 \) and \( u_g = 0.35 \text{ m/s} \) at two different chord based Reynolds numbers.](image)

![Figure 25: Greyscale plot of the wall shear stress magnitude at \( \alpha_m = 15^\circ \). The laminar separation bubble at the leading edge is visible in the velocity vector plot.](image)
The mean chord force coefficient computed for the high Reynolds number case at $\alpha_m = 15^\circ$ is shown in Figure 27. As before, the predicted performance of the rudder was greatly improved compared to the model-scale Reynolds number case, which is shown in Figure 19. The 10-fold reduction of the viscosity led to an increase of approximately 30% in rudder thrust in the chordwise direction. This highlights the importance of having the correct chord-based Reynolds number when accurate predictions of lift and drag coefficients are desired. Also, the steady performance prediction seemed to be more consistently an over-prediction of the real transient rudder performance at the higher Reynolds number.

4.2.5 Effect of waveform
The effect of including a second harmonic signal in the angle of attack, so as to more accurately emulate a realistic propeller slipstream, can be seen in Figure 28 and Figure 29. The chord force coefficients are plotted alongside $\alpha_{\ell,k}$, which is the angle of attack calculated at a point in a plane perpendicular to the direction of the mean flow, at a distance of roughly 2.7 chord lengths from the rudder. Figure 28 shows the results with only a single frequency in the angle of attack signal, and
Figure 29 shows the corresponding data when the second harmonic was included. The computed mean chord force coefficient in the first case was $\langle c_c \rangle = -0.2002$, and including the second harmonic only yielded a reduction of 0.4% in the mean chord force coefficient. This can be understood by noting from the waveform in Figure 29 that, although the maximum value of $\alpha_m$ is larger when the second harmonic is included, it is compensated by a more narrow pulse width and a larger amount of time with smaller angle of attack. It would therefore seem that resolving the slipstream harmonics of higher frequency than blade frequency is less important for the total rudder performance.

Figure 28: History of angle of attack and lift coefficient response with only fundamental frequency.

$\alpha_m = 15^\circ$, $U_m = 5.0 \, m/s$, $k_1 = 2.5$, $u_{g_1} = 0.45 \, m/s$.

Figure 29: History of angle of attack and lift coefficient response with fundamental frequency and second harmonic. $\alpha_m = 15^\circ$, $U_m = 5.0 \, m/s$, $k_1 = 2.51$, $u_{g_1} = 0.45 \, m/s$, $k_2 = 5.02$, $u_{g_2} = 0.45 \, m/s$. 
4.2.6 **Comparison with Sears’ and Horlock’s theory**

The amplitude of the oscillations in the lift coefficient at $\alpha_m = 0^\circ$ is shown versus reduced frequency in Figure 30. The analytic solution of Sears is included for comparison, as well as the analytic solution rescaled by the slope of the steady lift coefficient, found through linear fitting to the steady data in Figure 14. The CFD results agree well with the analytic theory at this mean angle of attack and gust amplitude. However, at $\alpha_m = 10^\circ$, with the same gust amplitude and over the same range of frequencies, the agreement between linear theory and CFD was much poorer, as seen in Figure 31. Although omitted in this report, calculations for gusts of larger amplitude showed similar behaviour, indicating that the linear theory is best suited for cases when the mean angle of attack is small. This is explained by the large region of separation that is created behind the inclined rudder, and the non-linear effects associated with it. Adjusting the coefficient in the analytic expression to the lift slope data from the steady simulations improved the prediction in the $10^\circ$ case, while yielding a poorer prediction when the rudder was at zero mean angle of attack.

![Figure 30: Unsteady lift coefficient amplitude versus reduced frequency at $\alpha_m = 0^\circ, u_g = 0.35 \text{ m/s.}$](image1)

![Figure 31: Unsteady lift coefficient amplitude versus reduced frequency at $\alpha_m = 10^\circ, u_g = 0.35 \text{ m/s.}$](image2)
4.2.7 Comparison with the Shilling rudder

The simulations with the Shilling rudder at $\alpha_m = 0^\circ$ showed the same qualitative behaviour as for the NACA 0021. The results are presented in Figure 32, and as for the NACA 0021 the mean chord force coefficient was increasing monotonically with increasing $k$. For the Shilling rudder, it does however appear as if the coefficients were converging towards a different value than that of the steady simulation. This was most likely due to the fact that the flow behind a Shilling rudder is largely separated even at $\alpha_m = 0^\circ$, making the flow intrinsically unsteady and causing a disagreement between the steady solution and mean value of a transient simulation. The dependency of the mean chord force coefficient on $k$ appeared to be weaker for this geometry than for the NACA 0021 but the trend was similar. Convergence could not be reached for transient simulations at larger $\alpha_m$.

![Figure 32: Mean chord force coefficient versus reduced frequency at $\alpha_m = 0^\circ$ for the Shilling rudder.](image)

4.3 Summary of the results

The performance of the rudder has been investigated with respect to the control parameters $k$, $u_g$, $\alpha_m$ and $Re$, independently. It was found that the effect of varying one of the parameters cannot be determined without also knowing the others. At $\alpha_m = 0^\circ$ and $10^\circ$, it was found that $\partial \langle c_c \rangle / \partial k > 0$ for all of the considered values of $u_g$, whereas this monotonic behaviour was not found for all gust amplitudes at $\alpha_m = 15^\circ$. The steady simulations appeared to over-predict the performance compared to the transient simulations for large values of $k$ and $u_g$. At low values of the reduced frequency ($k \lesssim 2$), the steady prediction was on the other hand a more conservative estimate of the thrust.

The effect of varying $Re$ was tested at $\alpha_m = 0^\circ$ and $15^\circ$, where it was found that the predicted performance was significantly better at $Re = 7.5 \cdot 10^6$ than at $Re = 7.5 \cdot 10^5$. The steady prediction was also more favourably affected than the result of the transient simulation. The difference between the prediction of the SST $k - \omega$ with and without transition modelling was large at model-scale Reynolds number, where the predicted viscous stress on the rudder was decreased due to the formation of a laminar separation bubble. At higher Reynolds number, the two approaches produced almost identical results at $\alpha_m = 0^\circ$ and no LSB was observed.

When the amplitude of the response in lift coefficient to the unsteady inflow was compared to analytic theory, the agreement was good at $\alpha_m = 0^\circ$, but quite poor at $\alpha_m = 10^\circ$. The result of including a
second harmonic of comparable magnitude in the gust signal indicated that the response to the fundamental frequency dominates that of the overtone.

The behaviour of the Shilling rudder was qualitatively similar to that of the NACA 0021 at $\alpha_m = 0^\circ$, although with a weaker dependency on $k$. This suggests that the results found in this study have applicability to considerations of other rudder geometries.

4.4 Practical comments and sources of error

Because of limitations in computational cost, the data presented in this report was generated by simulations that are more or less distorted representations of the desired physical situation. Mainly, the practically unavoidable spatial under-resolution of the periodic gust at high $k$ has led to numerical diffusion in the streamwise direction and an under-prediction of the amplitude of the unsteadiness at the trailing edge of the rudder. The amplitude decreased with around 20% per wavelength, meaning that the amplitude at the trailing edge was $k$-dependent. This inevitably lead to some amount of biasing, and thereby an over-prediction of the $k$-dependency in the force coefficients. The perfectly harmonic perturbation of moderate aspect ratio is nevertheless inherently sensitive to the Kelvin-Helmholtz instability even in theory, and the viscous stresses are proportional to $\omega^2 u_g / U_m^2 \sim k^2 u_g$. Thus it seems implausible to generate such a velocity field entirely free from diffusion of momentum.

Judging by the grid independence study described in chapter 3, it is also likely that the simulations did not quite correspond to true open water calculations without boundary effects. The exact numerical values of the force coefficients should as such be regarded with some caution, but the trends and the behaviour of the rudder with respect to control parameters should be reproducible. It is possible that the rudder wake was also under-resolved. Date et al. have found that the grid density in the wake of a Shilling rudder at zero angle of incidence had a clear impact on the computed drag coefficient [34].

The convergence of the force coefficients towards a purely time-periodic behaviour was found to be notably slower than the convergence of the angle of attack history, computed at the rudder. This could be due to some initial physical effect, e.g. transient boundary layer development over the rudder, or the result of modelling error in the SST $k - \omega$ model. In either case, a fair amount of gust periods (on the order of 10 – 20) were allowed to pass the rudder before the time averaging would begin.
5 Discussion and conclusions

The agreement of the results of the simulations with previous simulation data or experiment gives credibility to the results presented in this work. The stall angle and lift coefficient slope in the steady case agreed very well with the previous results of Swalwell et al. [8] and Gregorek et al., referenced in [35], and the transient simulations reproduced the behaviour found by James Date [5].

5.1 Accuracy of transient versus steady state rudder forces

It should be kept in mind when interpreting the data presented in this report, that a steady simulation corresponds to the case where \( \omega = 0 \), and not \( \omega \to 0 \). The latter case is the quasi-steady limit, in which the predicted performance may still be very different from a steady simulation. Thus, in order to estimate the accuracy of modelling the transient case with a steady simulation, one should locate the transient design point (the \( k, \alpha_m \) and \( u_g \) that best correspond to the real case) and consider the trend as \( u_g \to 0 \). In this study it has been found that such a steady simulation may lead to severe over- or under-prediction of the rudder performance.

It has been found that the accuracy of modelling the rudder performance with a steady simulation is likely poor when the actual transient inflow at the rudder consists of large amplitude gusts at high reduced frequency and large mean angles of attack. A steady solution, i.e. the result of an ADM simulation, would likely over-predict the performance of the rudder at spanwise sections where the rudder in reality is submitted to such unsteady inflow and a mean angle of attack close to the static stall angle. As seen from e.g. Figure 19, these sections could perform considerably worse in experiment or fully transient simulations than the steady model would indicate, if the 2D analysis holds for the fully three-dimensional case. On the other hand, for rudder sections where the mean value of the transient angle of attack is smaller, e.g. further outside the projected area of the propeller disk, the discrepancy between steady prediction and real performance could be acceptable. For smaller frequencies, e.g. \( k < 3 \), the accuracy of a steady solution is much more difficult to estimate. It may be an over- or under-prediction depending on the level of unsteadiness, i.e. gust amplitude, and the mean angle of attack at the considered rudder cross-section.

A quantitative steady performance prediction of the rudder as a whole seems to be intrinsically uncertain due to these complications. However, it could be concluded that the larger the amplitude of the unsteady oscillations and the greater the mean angles of attack are, the less likely a steady simulation is to predict rudder performance that agrees with experiment. For practical purposes, this could mean that a steady ADM simulation is less suitable for rudder performance prediction if the propeller is heavily loaded, which typically increases the generated angles of attack [1]. In such a situation, it is the belief of the author that the steady simulation would over-predict the performance of the rudder. Further studies in 3D and/or for larger \( k \) are recommended to validate or falsify this statement. As previously stated, the accuracy of a steady solution is even harder to predict for a propeller that operates at exceptionally low \( k \), which likely makes steady analysis invalid for this type of application.

The back-of-the-envelope calculations previously shown in the results, comparing the relative contributions of the chord force and side force to the total rudder performance, indicated that the accuracy of the chord force coefficient is of greatest importance for reasonably twisted rudders. Nevertheless, the contribution of the side force to the performance increases with the twist angle, and little attention has been given to \( c_s \) in this study. For highly twisted rudders (e.g. twist angle greater than \( 5^\circ \)), a more detailed analysis of the side force coefficient is also necessary to determine the accuracy of a steady prediction.

The main goal of this study was to determine whether it is possible to accurately predict the forces acting on a rudder using a steady state model of the real, fully transient case. At first glance, the data in Figure 17 seems to indicate that if the actual propeller-rudder system being simulated operates at a sufficiently large \( k \), the true performance (experimentally measured or simulated fully transient) could become independent of \( k \) and only weakly dependent on \( u_g \). In this limit the steady results
might be close (within 10%, for instance) to the results of a transient simulation. However, Figure 17 only shows the data for $\alpha_m = 0^\circ$, and so this conclusion cannot be drawn in general.

The dependency of the mean chord force coefficient on the mean angle of attack is strong, which makes the accuracy of the predicted performance of a rudder section dependent on the inflow at the specific spanwise location. In fact, the predicted performance of the rudder has not exhibited limiting behaviour with increasing $k$ for all gust amplitudes in the two cases where $\alpha_m = 10^\circ$ and $15^\circ$, which are fairly typical angles of attack in a propeller slipstream. Therefore, simple conclusions that hold for any propeller slipstream or any given rudder are hard to draw, solely based on the results presented in this report. Nor can it be deduced from the shown data what values of $k$, if any, are large enough for experiment or a fully transient model to agree with a steady simulation.

In conclusion, the parameter values for which accurate steady state modelling of the transient rudder performance is possible could not be quantified. Based on the results presented in this study, such a task seems more complex than the first impression would tell. It is unclear whether or not the error is predictable for model parameters smaller or larger than some critical values, and if so, it is also unclear how much the critical values would depend on the geometry of the specific case. A very rough estimate of the modelling error due to predicting the rudder performance in a steady state is an over-prediction on the order of 10 – 50%, provided that the reduced frequency of the propeller-rudder configuration is in the range $3 < k < 5$. For smaller $k$, the error is too dependent on the amplitude of the unsteadiness to be easily predicted, and the behaviour at larger $k$ has not been thoroughly investigated. Further simulations, as suggested in chapter 6, might give a more definite answer to this question. Instead it appears as if transient modelling, or some form of correction of the steady data, e.g. using previous knowledge of the slipstream and empirical correlations is necessary for accurate performance prediction.

5.2 Suggestions for an improved steady state prediction

With some knowledge of the real wake behind the propeller being modelled by the ADM, e.g. the history of axial and tangential velocity at a number of points on the radius, it might be feasible to estimate whether the predicted performance from the steady simulation is better or worse than experiment would show. A possible work flow for utilizing prior knowledge of the propeller slipstream together with 2D simulations in the rudder design process is presented in Figure 33.

If analytic models (e.g. linear potential theory and dynamic stall models) are implemented for the solution of the force coefficients, their respective modelling errors are also introduced into the result. Instead it is suggested that a database of predicted performance from a large number of 2D test cases is generated and used for reference in the design process. This database could for instance be a large higher-order tensor where the performance of the 2D section is stored for specific values of $k, u_g, Re, \alpha_m$, etc. If the angle of attack history is known at a number of points, these parameters could be easily estimated using an FFT routine. A very large number of simulations would be necessary before a sufficiently extensive database is generated, but such simulations may be scripted and run in parallel when computer time is available, as different sections are solved for independently. The idea of reusability also helps to make the approach valid for industrial purposes. With this information, it could be estimated whether the performance predicted by an ADM model will be conservative or optimistic, based on the relative 2D performance of a few cross-sections.

The restriction of the flow to horizontal motion (i.e. no cross-flow) and the discretization error associated with the finite number of spanwise sections would of course lead to losses in accuracy. Still, it is possible that fast considerations of this kind may at an early stage provide the rudder designer with an indication of the accuracy of an ADM simulation and an estimate of the effects on the performance due to design changes.

---

3 Fast Fourier Transform.
The amplitude of the response in lift coefficient to the transient gust agreed well with the analytic theory of Sears at $\alpha_m = 0^\circ$. At this angle of incidence, however, the lift coefficient has no contribution to the chord force coefficient. At $\alpha_m = 10^\circ$, where the lift coefficient contributes largely to the total chord force coefficient as given by (2.7), the agreement with the linear theory of Horlock was poor. For this reason, the linear theories of Horlock or Sears are probably not optimal for rudder design.

5.3 General conclusions and other comments

Another aim of this work was to determine the importance of the choice of numerical model. For the simulations presented in this report, which were conducted at model scale, the effect of including transition modelling was substantial, as can be seen in Figure 19 and Figure 24. The one-equation transition model predicted considerably better performance (40−50% reduction in drag) at $\alpha_m = 0^\circ$ for all $k$, and 30% better performance at $\alpha_m = 15^\circ$. The main contribution to the favourable prediction of the transition model was lower pressure drag, but viscous stress was also reduced. Both effects were related to the formation of a laminar separation bubble. The lack of LSB at higher Reynolds number for the case of $\alpha_m = 0^\circ$ made the transition model redundant for performance prediction.

Although the discrepancy in chord force coefficient at $\alpha_m = 0^\circ$ was basically a vertical translation of the data points, the curve took another shape when transition modelling was applied at $15^\circ$. This may imply that the validity of a stationary performance prediction is affected by whether or not transition modelling is included, although from the presented data no conclusion could be drawn either way. It is also hard to deduce from the results of this study which approach is more accurate for the present application, as no experimental data have been found that could be used to validate the simulations with transition modelling. Further studies would be necessary, both with respect to the dependency on the flow parameters and through comparison with experiment to determine which model is best suited for rudder prediction.

The Reynolds number had a similar impact on the results, where a 10-fold reduction of the viscosity yielded a reduction in drag of around 40−50% at $\alpha_m = 0^\circ$ and around 30% more thrust at $15^\circ$. It is the belief of the author that the similarity between the numerical value of this effect and that of including transition modelling is merely a coincidence. The discrepancy is a clear indication that rudder simulations and tests at model-scale may suffer from considerable losses in accuracy when used to predict the performance of a full-scale rudder. The effect is probably particularly pronounced when there is strong separation at the rudder, e.g. for rudder twist distributions that give rise to large angles of attack. Thus, an estimation of the scaling effect, e.g. by simulation at two or more values of $Re$, is highly recommended for the validation of model-scale simulations. It is also interesting to note that the steady performance prediction was more favourably affected by an increased Reynolds number at $\alpha_m = 15^\circ$ than the transient prediction.

It has been found that the accuracy of the results can be very sensitive to the amplitude of the transient gust at the rudder. This means that properly resolving the slipstream and preventing artificial damping of the oscillations is of very large importance for the accuracy of transient simulations. This in turn
implies high demands on the mesh density, even in the free-stream and possibly beyond what is common practice. On the other hand, the simulation with a realistic waveform hinted that resolving the higher order harmonics of the propeller slipstream may have negligible effect on the performance of the rudder. If this observation is a general property of rudder performance, it would imply that some artificial damping of the slipstream is acceptable for accurate performance prediction.
6 Future developments

Two important obstacles that need to be overcome before definite conclusions can be drawn regarding the validity of steady rudder performance prediction are simulations at higher $k$, which are perhaps more representative of a real propeller, and simulations in 3D. The author suggests comparisons between an ADM and e.g. a sliding mesh simulation or experimental data for a given propeller over a range of advance coefficients. This could help to determine whether heavily loaded propellers indeed are less suitable for steady rudder performance prediction, as was concluded for 2D in this study.

It is recommended for future studies of this kind to attempt to generate a finer grid where necessary (rudder wake, and possibly near-body free-stream), and perhaps to reduce the extent of the computational domain, while modifying the boundary conditions to avoid boundary effects. A symmetry or slip boundary condition might be appropriate on the walls parallel to the free-stream.

Due to its clear effect on the predicted rudder performance, it is suggested that transition modelling is included in further simulations, provided that the additional boundary conditions are such that a realistic inflow reaches the rudder.
7 References


Appendix A. Table of values of Sears’ and Horlock’s functions

*Table A1: Values of Sears’ \( S \) and Horlock’s \( T \) function versus reduced frequency. \( i = \sqrt{-1} \). [18]*

<table>
<thead>
<tr>
<th>( k )</th>
<th>( S(k) )</th>
<th>( T(k) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.821 – 0.164( i )</td>
<td>1.819 – 0.114( i )</td>
</tr>
<tr>
<td>0.1</td>
<td>0.702 – 0.160( i )</td>
<td>1.692 – 0.060( i )</td>
</tr>
<tr>
<td>0.2</td>
<td>0.624 – 0.126( i )</td>
<td>1.602 + 0.022( i )</td>
</tr>
<tr>
<td>0.3</td>
<td>0.568 – 0.085( i )</td>
<td>1.528 + 0.111( i )</td>
</tr>
<tr>
<td>0.4</td>
<td>0.525 – 0.044( i )</td>
<td>1.463 + 0.198( i )</td>
</tr>
<tr>
<td>0.5</td>
<td>0.456 + 0.032( i )</td>
<td>1.337 + 0.361( i )</td>
</tr>
<tr>
<td>0.7</td>
<td>0.369 + 0.126( i )</td>
<td>1.134 + 0.566( i )</td>
</tr>
<tr>
<td>1.0</td>
<td>0.082 + 0.268( i )</td>
<td>0.306 + 0.845( i )</td>
</tr>
<tr>
<td>2.0</td>
<td>−0.048 + 0.247( i )</td>
<td>−0.096 + 0.744( i )</td>
</tr>
<tr>
<td>2.5</td>
<td>−0.145 + 0.178( i )</td>
<td>−0.525 + 0.315( i )</td>
</tr>
<tr>
<td>3.5</td>
<td>−0.081 – 0.159( i )</td>
<td>−0.259 – 0.487( i )</td>
</tr>
</tbody>
</table>
Appendix B. Coordinates of the NACA 0021 profile

*Table B1: Thickness distribution of the rudder profile used in the simulations [33]. T is the maximum thickness and c is the chord length.*

<table>
<thead>
<tr>
<th>$x/c$</th>
<th>$t/T$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0000</td>
<td>0.02104</td>
</tr>
<tr>
<td>0.9500</td>
<td>0.13442</td>
</tr>
<tr>
<td>0.9000</td>
<td>0.24124</td>
</tr>
<tr>
<td>0.8000</td>
<td>0.43707</td>
</tr>
<tr>
<td>0.7000</td>
<td>0.61043</td>
</tr>
<tr>
<td>0.6000</td>
<td>0.76028</td>
</tr>
<tr>
<td>0.5000</td>
<td>0.88204</td>
</tr>
<tr>
<td>0.4000</td>
<td>0.96687</td>
</tr>
<tr>
<td>0.3000</td>
<td>1.00000</td>
</tr>
<tr>
<td>0.2500</td>
<td>0.98981</td>
</tr>
<tr>
<td>0.2000</td>
<td>0.95583</td>
</tr>
<tr>
<td>0.1500</td>
<td>0.89052</td>
</tr>
<tr>
<td>0.1000</td>
<td>0.78018</td>
</tr>
<tr>
<td>0.0750</td>
<td>0.69973</td>
</tr>
<tr>
<td>0.0500</td>
<td>0.59225</td>
</tr>
<tr>
<td>0.0250</td>
<td>0.43564</td>
</tr>
<tr>
<td>0.0125</td>
<td>0.31559</td>
</tr>
<tr>
<td>0.0050</td>
<td>0.20419</td>
</tr>
<tr>
<td>0.0000</td>
<td>0.00000</td>
</tr>
</tbody>
</table>
Appendix C. Derivation of the inlet boundary condition

We want to describe a velocity field that consists of a mean flow component and a transverse periodic gust. We may write this as

$$\mathbf{u} = \mathbf{U}_0 + \mathbf{\tilde{u}} \sin(\mathbf{k} \cdot \mathbf{x} - \omega t)$$  \hfill (C.1)

where $\mathbf{U}_0 = (U_{0x}, U_{0y})$ and $\mathbf{\tilde{u}} = (\tilde{u}_x, \tilde{u}_y)$ are constant vectors, $\mathbf{u} = (u, v)$ is the total velocity field, $\mathbf{k} = (k_x, k_y)$ is the wave vector and $\omega$ is the angular frequency of the periodic gust. The velocity field should obey the continuity equation

$$\nabla \cdot \mathbf{u} = 0$$  \hfill (C.2)

as well as the transverse wave property, which may be written

$$\mathbf{k} \cdot \mathbf{\tilde{u}} = 0.$$  \hfill (C.3)

We also want the periodic gust to be perpendicular to the mean flow, which implies

$$\mathbf{U}_0 \cdot \mathbf{\tilde{u}} = 0.$$  \hfill (C.4)

We have that

$$\nabla \cdot \mathbf{u} = (k_x \tilde{u}_x + k_y \tilde{u}_y) \cos(\mathbf{k} \cdot \mathbf{x} - \omega t) = \mathbf{k} \cdot \mathbf{\tilde{u}} \cos(\mathbf{k} \cdot \mathbf{x} - \omega t)$$  \hfill (C.5)

which implies that any velocity field that simultaneously obeys equations (C.1) and (C.3) will also obey the continuity equation. Furthermore, the equations (C.3) and (C.4) imply that the wave vector $\mathbf{k}$ should be aligned with the mean flow $\mathbf{U}_0$. Let the mean flow be given by

$$\mathbf{U}_0 = (U_{0x}, U_{0y}) = (U_m \cos \alpha_m, U_m \sin \alpha_m).$$  \hfill (C.6)

where $\alpha_m$ is the angle of attack of the mean flow and $U_m$ is a constant magnitude. This means that

$$\mathbf{k} = (k_x, k_y) = \left( \frac{2\pi}{\lambda} \cos \alpha_m, \frac{2\pi}{\lambda} \sin \alpha_m \right)$$  \hfill (C.7)

where $\lambda$ is the wave length of the oscillations. Since the oscillations are transported by the mean flow and forced at the angular frequency $\omega$, the wave length is given by

$$\lambda = 2\pi U_m / \omega.$$  \hfill (C.8)

This yields

$$\mathbf{k} = \left( \frac{\omega}{U_m} \cos \alpha_m, \frac{\omega}{U_m} \sin \alpha_m \right).$$  \hfill (C.9)

We may now choose

$$\mathbf{\tilde{u}} = (\tilde{u}_x, \tilde{u}_y) = (u_g \sin \alpha_m, -u_g \cos \alpha_m)$$  \hfill (C.10)

which fulfills equations (C.3) and (C.4) for any choice of the constant gust amplitude $u_g$. Finally we have that the incompressible velocity field representing the mean flow and the transverse periodic gust is given by

$$u = U_m \cos \alpha_m + u_g \sin \alpha_m \sin \left( \frac{\omega}{U_m} x \cos \alpha_m + \frac{\omega}{U_m} y \sin \alpha_m - \omega t \right)$$  \hfill (C.11)

$$v = U_m \sin \alpha_m - u_g \cos \alpha_m \sin \left( \frac{\omega}{U_m} x \cos \alpha_m + \frac{\omega}{U_m} y \sin \alpha_m - \omega t \right).$$  \hfill (C.12)
Appendix D. Figures of the dimensionless wall distance

Figures D1 through D4 show the $y^+$ values in the first cells at the rudder surface at a fixed instant in the transient simulations. All figures were produced with $\alpha_m = 0^\circ$.

Figure D1: Instantaneous greyscale plot of $y^+$ for $Re = 7.5 \cdot 10^5$ with the SST $k - \omega$ model.

Figure D2: Instantaneous greyscale plot of $y^+$ for $Re = 7.5 \cdot 10^5$ with the transition model.
Figure D3: Instantaneous greyscale plot of $y^+$ for $Re = 7.5 \cdot 10^6$ with the SST $k-\omega$ model.

Figure D4: Instantaneous greyscale plot of $y^+$ for $Re = 7.5 \cdot 10^6$ with the transition model.
Appendix E. Convected vortices from the leading edge

Figure E1: The convected vortices shed from the leading edge at blade frequency. The greyscale represents pressure on the rudder surface. $\alpha_m = 15^\circ$, $k = 5$ and $u_g = 1.05 \text{ m/s}$. Picture a) is taken 20 time steps before picture b).
Appendix F. Streamline plots of the NACA 0021

Figure F1: Streamlines around the NACA 0021 at $k = 5, u_g = 1.05 \text{ m/s}, \alpha_m = 10^\circ$.

Figure F2: Streamlines around the NACA 0021 at $k = 5, u_g = 1.05 \text{ m/s}, \alpha_m = 15^\circ$. 
Figure F3: Streamlines showing the trailing edge separation without transition modelling at $\alpha_m = 15^\circ, k = 5, u_g = 0.35 \text{ m/s}$.

Figure F4: Streamlines showing the trailing edge separation with transition modelling at $\alpha_m = 15^\circ, k = 5, u_g = 0.35 \text{ m/s}$.
Appendix G. Tables of lift and drag coefficients at $\alpha_m = 10^\circ$ and 15$^\circ$.

Table G1: The mean drag coefficient at $\alpha_m = 10^\circ$.

<table>
<thead>
<tr>
<th>$k$</th>
<th>$u_g = 0.10 \text{ m/s}$</th>
<th>$u_g = 0.35 \text{ m/s}$</th>
<th>$u_g = 1.05 \text{ m/s}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0331</td>
<td>0.0312</td>
<td>0.0147</td>
</tr>
<tr>
<td>2</td>
<td>0.0333</td>
<td>0.0325</td>
<td>0.0270</td>
</tr>
<tr>
<td>3.5</td>
<td>0.0333</td>
<td>0.0334</td>
<td>0.0332</td>
</tr>
<tr>
<td>5</td>
<td>0.0334</td>
<td>0.0337</td>
<td>0.0337</td>
</tr>
</tbody>
</table>

Table G2: The mean lift coefficient at $\alpha_m = 10^\circ$.

<table>
<thead>
<tr>
<th>$k$</th>
<th>$u_g = 0.10 \text{ m/s}$</th>
<th>$u_g = 0.35 \text{ m/s}$</th>
<th>$u_g = 1.05 \text{ m/s}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-0.7943</td>
<td>-0.7921</td>
<td>-0.7804</td>
</tr>
<tr>
<td>2</td>
<td>-0.7943</td>
<td>-0.7917</td>
<td>-0.7990</td>
</tr>
<tr>
<td>3.5</td>
<td>-0.7944</td>
<td>-0.7940</td>
<td>-0.8020</td>
</tr>
<tr>
<td>5</td>
<td>-0.7947</td>
<td>-0.7957</td>
<td>-0.7800</td>
</tr>
</tbody>
</table>

Table G3: The mean drag coefficient at $\alpha_m = 15^\circ$.

<table>
<thead>
<tr>
<th>$k$</th>
<th>$u_g = 0.10 \text{ m/s}$</th>
<th>$u_g = 0.35 \text{ m/s}$</th>
<th>$u_g = 1.05 \text{ m/s}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0684</td>
<td>0.0674</td>
<td>0.0698</td>
</tr>
<tr>
<td>2</td>
<td>0.0687</td>
<td>0.0730</td>
<td>0.0897</td>
</tr>
<tr>
<td>3.5</td>
<td>0.0698</td>
<td>0.0781</td>
<td>0.0916</td>
</tr>
<tr>
<td>5</td>
<td>0.0700</td>
<td>0.0744</td>
<td>0.0825</td>
</tr>
</tbody>
</table>

Table G4: The mean lift coefficient at $\alpha_m = 15^\circ$.

<table>
<thead>
<tr>
<th>$k$</th>
<th>$u_g = 0.10 \text{ m/s}$</th>
<th>$u_g = 0.35 \text{ m/s}$</th>
<th>$u_g = 1.05 \text{ m/s}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-0.9892</td>
<td>-0.9867</td>
<td>-1.0936</td>
</tr>
<tr>
<td>2</td>
<td>-0.9909</td>
<td>-1.0412</td>
<td>-1.1504</td>
</tr>
<tr>
<td>3.5</td>
<td>-1.0048</td>
<td>-1.0739</td>
<td>-0.9747</td>
</tr>
<tr>
<td>5</td>
<td>-1.0037</td>
<td>-0.9965</td>
<td>-0.9269</td>
</tr>
</tbody>
</table>