

Design and Analysis of Swirl Recovery Vanes for an Isolated and a Wing Mounted Tractor Propeller

Tom Caton Arnaud Stokkermans

Flight Performance and Propulsion



DESIGN AND ANALYSIS OF SWIRL RECOVERY VANES FOR AN ISOLATED AND A WING MOUNTED TRACTOR PROPELLER

by

Tom Caton Arnaud Stokkermans

in partial fulfillment of the requirements for the degree of

Master of Science
in Aerospace Engineering

at the Delft University of Technology,
to be defended publicly on Thursday 27th of August, 2015 at 14:00.

Supervisors:	Prof. dr. ir. L.L.M. Veldhuis Prof. dr. -ing. G. Eitelberg
Thesis committee:	Dr. ir. C.J. Simao Ferreira Dr. H.H. Brouwer

An electronic version of this thesis is available at <http://repository.tudelft.nl/>.
Thesis registration number: 040#15#MT#FPP

ABSTRACT

In light of the energy crisis of the early 1970's, NASA and industry gained a renewed interest in high-speed propellers for improved propulsive efficiency and explored the idea of swirl recovery vanes (SRV) to generate a net thrust from the residual swirl in the propeller slipstream. After this first effort on the aerial application of SRV, only recently research is resumed. When a wing is introduced in the slipstream of a propeller, for instance for a wing-mounted tractor-propeller, conclusions drawn on SRV in isolated condition may not hold.

The objective of this research is to gain an improved understanding of the aerodynamic interaction between the propeller and swirl recovery vanes in an isolated configuration and wing-mounted tractor arrangement in the cruise condition and in a high-thrust condition. This study is realized by performing a series of transient Reynolds-averaged Navier-Stokes CFD simulations of a propeller with and without SRV in an isolated and installed configuration.

Throughout this research the 6-bladed propeller of the European APIAN project is used. Available experimental propeller performance, blade pressure and slipstream measurements are used to validate the isolated propeller CFD model. Within the limitations of fully turbulent modelling of the boundary layer by means of automatic wall functions, good agreement is found with the experimental data, including the existence of a conical separation vortex at low advance ratios.

Simulated performance and slipstream results are presented of the APIAN propeller with SRV designed for the APIAN-INF test program in the DNW-LLE. PIV measurements in a plane spanned by the radial and rotation axis provide a comparison of the slipstream velocity components and vorticity. This simulation combined with the PIV measurements enables an extensive description of the structure of root and tip vortices induced by the propeller blades and swirl recovery vanes. It is found that the propulsive efficiency increase by the addition of SRV is only $\Delta\eta_p = 0.57\%$ which is much lower than the design prediction of $\Delta\eta_p = 1.8\%$. Therefore this design is not used in the remainder of the research and new SRV designs are proposed.

An SRV analysis tool based on lifting-line theory modified for non-uniform inflow is presented. In combination with an optimisation routine, this tool allows for the design of SRV for an isolated propeller. From a simplified analysis of an elliptical vane in a uniform swirl flow, it is concluded that optimisation for maximum SRV thrust is preferred over complete swirl recovery to reach the highest gain in propulsive efficiency. Four designs are presented: Design 1 is optimised for the cruise condition with a constraint on stall for the high-thrust condition. Design 2 is optimised for the high-thrust condition with a constraint on the cruise condition for zero or positive efficiency benefit. These are designs where the SRV have a fixed pitch in flight. Also two variable pitch designs are proposed. The effect of cropping and the number of vanes on the propulsive efficiency is investigated as well for the objective of design 1.

Design 1 and 2 are used in CFD simulations behind the isolated propeller to validate the predictions from the SRV analysis tool. In general the simulation results show that SRV lead to an increase in propulsive efficiency by increasing the system thrust over a wide range of advance ratios, with minor effect on the system power. Gains in propulsive efficiency of 0.39 % and 0.20 % are found in the cruise condition and 2.62 % and 3.07 % in the high-thrust condition for design 1 and 2 respectively. For high advance ratios the prediction is very accurate, while towards lower advance ratios the tool overpredicts the propulsive efficiency gain. The difference is within the limits that can be explained by the set assumptions. Design 1 proves that it is possible to increase the propulsive efficiency of an operating point close to the point of maximum propeller propulsive efficiency. Design 2 shows that if a larger increase in propulsive efficiency at low advance ratios is desired, the design can be changed at the cost of propulsive efficiency benefit at higher advance ratios, for a fixed SRV pitch design. Downstream of the SRV, somewhat less than half of the swirl is recovered on average. An expansion of the slipstream boundary is present, which is the result of the interaction of propeller blade and vane tip vortices.

In the last part the wing of a Fokker 50 is introduced behind the propeller and SRV design 1. The loading on the wing induces an upwash upstream of the wing, resulting in a deviation from the SRV design inflow that is different for each vane by such a degree that flow separation degrades the SRV performance to a large extent. Therefore a change in the SRV design is made by turning each vane over an angle to obtain the time- and radial-average design inflow in the cruise condition. For future research it is recommended to find a different design for each vane. Since the effect of the wing upwash on the SRV inflow field varies with advance

ratio and with wing loading and thus varies in flight, a variable pitch SRV design is recommended where the pitch of each vane is adjusted individually. For the cruise condition the increase in propulsive efficiency by the addition of SRV without considering differences in wing drag is found to be $\Delta\eta_p = 0.93\%$, which is considerably higher than without wing, mainly due to the increased propeller propulsive efficiency, but partly by increased SRV thrust as well. $\Delta\eta_p = 2.14\%$ for a medium-thrust condition, which is very similar to the value without wing. For a wing-mounted tractor-propeller conclusions on SRV performance can only be drawn from the complete force balance of thrust and lift of the propeller, SRV, wing and nacelle. Considering the drag of all components, the net increase in propulsive efficiency by the addition of SRV is found to be $\Delta\eta_{p_{\text{net}}} = -0.14\%$ for the cruise and $\Delta\eta_{p_{\text{net}}} = 1.00\%$ for the medium-thrust condition with a net increase in lift of 0.35% and net decrease in lift of 0.55% respectively. Careful optimisation of SRV taking the wing into account as well as the lift as a constraint will most likely result in a performance benefit, since already with this non-optimised design an increase in thrust or lift can be found depending on the advance ratio. The propeller slipstream greatly affects the wing lift and drag distribution by its increased axial velocity and introduced swirl. It is concluded that SRV reduce some of the effects of the propeller on the wing lift and drag distribution by a reduction of the swirl, resulting in a smaller deviation from the wing loading without propeller. A design procedure for SRV should include the wing for instance by an additional lifting line and optimise for combined SRV and wing maximum thrust with a constraint on the net lift. This may lead to SRV designs more focussed on providing the optimal inflow for the wing in order to reduce the wing drag.

CONTENTS

Abstract	iii
Preface	vii
Nomenclature	ix
1 Introduction	1
1.1 Propeller Performance and the Swirl Loss	2
1.2 Swirl Recovery Vanes State of Art	3
1.3 Opportunities and Challenges of SRV for a Wing-Mounted Tractor-Propeller	7
1.4 Research Objective	9
1.5 Thesis Outline	9
2 Propeller CFD Model	11
2.1 Geometry	11
2.1.1 Propeller Geometry	11
2.1.2 Domain Geometry	12
2.2 Mesh	12
2.2.1 General Mesh Properties	12
2.2.2 Mesh Dependency Study	13
2.3 Solver	13
2.3.1 Governing Equations and their Discretization	14
2.3.2 Turbulence Model	14
2.3.3 Boundary Conditions and Interfaces	15
2.3.4 Solving Strategy	16
2.4 Results	16
2.4.1 Mesh Dependency Study	16
2.4.2 Influence of Inlet Turbulence and Blade Turbulent Modelling	25
2.4.3 Influence of Advance Ratio	25
3 APIAN-INF CFD Model	29
3.1 Geometry	29
3.1.1 SRV Geometry	29
3.1.2 Domain Geometry	30
3.2 Mesh	30
3.3 Solver	31
3.4 Results	32
3.4.1 SRV Wall Mesh Dependency Study	33
3.4.2 Effect of Profile Transformation	33
3.4.3 360 ° Domain Performance Results and Comparison with the Experiment	35
3.4.4 Description of the Slipstream and Comparison with the Experiment	36
4 SRV Design Procedure	45
4.1 Optimise For Maximum Thrust or Minimum Downstream Swirl?	45
4.2 SRV Analysis Tool	48
4.2.1 SRV Airfoil Properties	48
4.2.2 SRV Planform Geometry	49
4.2.3 Adapted Lifting Line Theory	50
4.2.4 Assumptions	53

4.3	Design Optimisation Routine	54
4.4	Design Results	55
4.4.1	Design for Cruise with Constraint Condition	55
4.4.2	Design for High-Thrust with Constraint Condition	59
4.4.3	Design for Cruise without Constraint Condition	61
4.4.4	Design for High-Thrust without Constraint Condition	61
5	Propeller-SRV CFD Model	63
5.1	Geometry	63
5.2	Mesh	63
5.3	Results	64
5.3.1	Effect of SRV on Propeller Performance	64
5.3.2	SRV Performance and Loading	66
5.3.3	Description of the Slipstream	76
6	Propeller-SRV-Wing CFD Model	83
6.1	Geometry	83
6.1.1	Propeller-SRV-Wing Geometry	83
6.1.2	Domain Geometry	84
6.2	Mesh	85
6.3	Results	86
6.3.1	Mesh Dependency Study	86
6.3.2	Effect of Wing on SRV Design	87
6.3.3	Effect of Wing on Propeller and SRV Performance	89
6.3.4	Effect of SRV on Thrust and Lift Force Balance	91
6.3.5	Description of the Slipstream	95
7	Conclusions and Recommendations	101
7.1	Conclusions	101
7.2	Recommendations	104
A	Newton's Method and Linearized Initial Estimate	107
B	Additional Results of Propeller-SRV-Wing CFD Model	109
	Bibliography	119

PREFACE

'I owned the world that hour as I rode over it. Free of the earth, free of the mountains, free of the clouds, but how inseparably I was bound to them.' [1] Charles Lindbergh once wrote this on flying over the Rocky Mountains in his single-engine propeller monoplane: the Spirit of St. Louis. This quote is relevant in multiple ways. First of all, the duration of powered flight is bound by its energy source: It is an ever continuing quest to increase the fuel efficiency of airplanes, in order to save fuel or to extend flight. This thesis is part of that quest for fuel efficiency. Second, this thesis is only relevant in light of a resurgence of propeller or open-rotor powered airplanes. Last but not least, this quote describes my passion for flying, aerospace engineering and the beauty of nature.

The achievement of this thesis would not have been possible without the support of others. I would like to thank my supervisors Leo and Georg for their input as well as for the interesting discussions during the weekly propeller research group meetings. I hope these meetings continue for future students since they provide a means to learn from the work of others and to see the relevance of your own work in an informal setting. I would also like to thank Nando for discussing and proofreading this work and the peers of room 6.01 for their company during the endless hours of preparing, running and processing the simulations. I would like to thank Elisabeth for reintroducing me to flying, this time in the form of gliding, which provided a welcome but relevant distraction from this work. Last but not least I would like to thank my parents who gave me the opportunity to reach this milestone.

*Tom Stokkermans
Delft, August 2015*

NOMENCLATURE

Latin Symbols

$\mathcal{R}$	Aspect ratio
$\frac{cm_{\max}}{c}$	Airfoil maximum camber over chord ratio
$\frac{t}{c}$	Airfoil thickness over chord ratio
$\frac{x_{cm_{\max}}}{c}$	Airfoil chordwise location of maximum camber
$d\mathbf{l}_k$	Panel k spanwise length vector
$\mathbf{A}_c$	Chord shape function coefficient vector
$\mathbf{A}_\theta$	Pitch shape function coefficient vector
$\mathbf{F}$	Force vector
$\mathbf{R}$	Residual vector
$\mathbf{r}_{j_A}$	Vector from corner point j_A to the control point at panel k
$\mathbf{r}_{j_B}$	Vector from corner point j_B to the control point at panel k
$\mathbf{u}_t$	Unit vector in the direction of the trailing vortices of a horseshoe vortex
$\mathbf{u}_{a_k}$	Unit vector tangent to chord of panel k
$\mathbf{u}_{n_k}$	Unit vector normal to chord of panel k
$\mathbf{V}$	Velocity
$\mathbf{V}_{in_k}$	Non-uniform inflow velocity at panel k
$\mathbf{v}_{in_k}$	Inflow velocity unit vector of panel k
b	Parabolic section drag polar constant
b	Wing span
C_D	Drag coefficient
c_j	Chord of panel j
C_L	Lift coefficient
c_m	Mean chord
C_P	Power coefficient ($\equiv \frac{P_s}{\rho_\infty n^3 D_p^5}$)
C_p	Pressure coefficient
c_r	Root chord
C_T	Thrust coefficient ($\equiv \frac{T_p}{\rho_\infty n^2 D_p^4}$)
c_{75}	Propeller blade chord at $0.75R_p$
C_{d_0}	Section zero-lift drag coefficient
C_{l_0}	Section lift coefficient at zero angle of attack
$C_{l_{\alpha k}}$	Section lift curve slope of panel k
C_{l_α}	Section lift curve slope
$C_{l_{\max}}$	Section maximum lift coefficient
$C_{T_{SRV}}$	SRV thrust coefficient
D_p	Propeller diameter
G_k	Dimensionless strength of the horseshoe vortex at panel k
J	Advance ratio ($\equiv \frac{V_\infty}{nD_p}$)
k	Turbulence kinetic energy
l_n	Length of the nacelle including spinner and hub
L_v	Vane lift

L_{l_v}	Vane lateral force
$L_{w_{\text{without}}}$	Wing lift for the configuration without SRV
m	Number of shape function coefficients
M_∞	Undisturbed Mach number
n	Propeller rotational speed
$N_{1,\theta}$	Class function variable determining root pitch slope
$N_{1,c}$	Class function variable determining root chord slope
$N_{2,\theta}$	Class function variable determining tip pitch slope
$N_{2,c}$	Class function variable determining tip chord slope
n_{cr}	Critical amplification factor
N_{hv}	Number of horseshoe vortices per swirl recovery vane
P_p	Propulsive power
P_s	Shaft power
p_t	Total pressure
p_∞	Undisturbed static pressure
r	Radial position
R_n	Nacelle radius
R_p	Propeller radius
R_{SRV}	Swirl recovery vane radius
Re_c	Reynolds number based on chord
s	SRV span
S_{SRV}	SRV planform area
T	Thrust
T_b	Propeller blade thrust
T_p	Propeller thrust
T_u	Turbulence level
T_v	Vane thrust
T_{net}	Net thrust
t_{bp}	Propeller blade passage period
$T_{p_{\text{without}}}$	Propeller thrust for the configuration without SRV
T_{SRV}	SRV thrust
T_{t_∞}	Undisturbed total temperature
U	Mean velocity
u'	Root mean square of turbulent velocity fluctuations
V_a	Axial velocity component
V_r	Radial velocity component
V_t	Tangential velocity component
V_∞	Undisturbed air speed
z_{SRV}	SRV quarter chord downstream distance from propeller blade rotation axis

Greek Symbols

α_i	Induced angle of attack
α_k	Local angle of attack at panel k
α_{0k}	Section zero-lift angle of attack of panel k
α_0	Section zero-lift angle of attack
$\alpha_{C_{l_{\text{max}}}}$	Angle of attack of section maximum lift coefficient
β_{75}	Propeller blade pitch at $0.75R_p$
β_{SRV}	SRV angle

$\Delta\eta_p$	Increase in propulsive efficiency by the addition of SRV
$\Delta\eta_{p_{\text{net}}}$	Net increase in propulsive efficiency by the addition of SRV for the configuration with wing
$\Delta\eta_{p_p}$	Increase in propeller propulsive efficiency by the addition of SRV
$\Delta\mathbf{G}$	Dimensionless vortex strength correction vector
ΔC_P	Increase in power coefficient by the addition of SRV
ΔC_{T_p}	Increase in propeller thrust coefficient by the addition of SRV
ΔC_T	Increase in overall (propeller and SRV thrust combined) thrust coefficient by the addition of SRV
$\Delta p_{t_{\text{inlet}}}$	Inlet total pressure jump
η_p	Propulsive efficiency ($\equiv \frac{TV_\infty}{P_s} = \frac{C_T J}{C_P}$)
$\frac{\mu_t}{\mu}$	Eddy viscosity ratio
Γ_j	Strength of the horseshoe vortex at panel j
λ	Tip-to-root chord taper ratio
$\Lambda_{c/4}$	Quarter chord sweep angle
ω	Vorticity
ζ_k	Dimensionless spanwise length vector of panel k
ω_t	Tangential vorticity component
ϕ	Swirl angle
ϕ_d	Swirl angle downstream of the SRV
ρ	Air density
ρ_∞	Undisturbed air density
Θ	Twist angle
θ	Pitch angle
θ_r	Root pitch angle

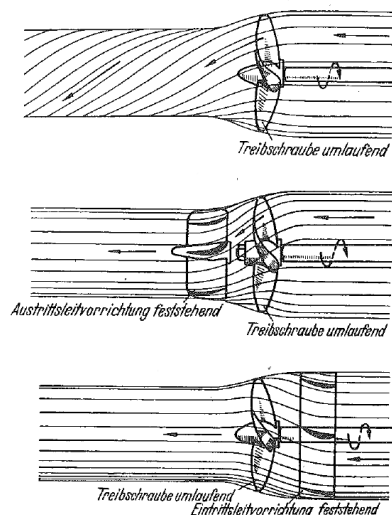
1

INTRODUCTION

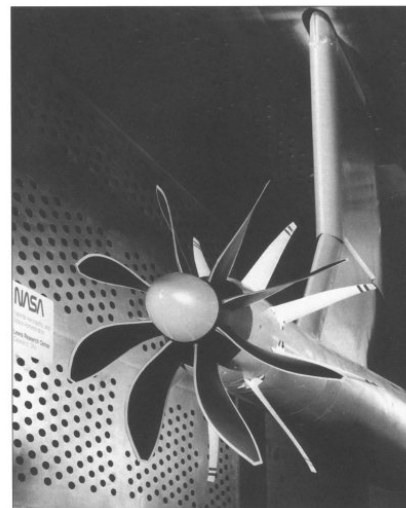
The torque applied to turn a propeller results in rotation of the fluid, the so-called swirl. A part of the shaft power is going into this angular momentum of the fluid and does not result in useful propulsive power. As early as 1903 a US patent was filed by Curtis [2] with the idea of marine pre-swirl vanes:

The object I have in view is to improve the efficiency of marine propellers by giving the column of water a spiral or rotary motion before it strikes the blades of the propellers, so as to present it to the blades at the most effective angle. I find that this can be accomplished by the employment of a number of stationary blades radiating from the propeller-shaft and curved or inclined to give the column of water the desired spiral or rotary motion opposite to the rotary motion produced by the propeller itself.

The idea is that the pre-swirl vanes create a swirl, which is counteracted by the propeller swirl. In this way the propeller shaft power that is normally put into angular momentum of the fluid is not lost but put into momentum in the axial direction, improving the propulsive efficiency. The propeller and vanes can also be interchanged such that the vanes recover the swirl of the propeller: The principle of swirl recovery vanes (SRV) is to generate a net thrust from the residual swirl in the propeller slipstream. Both layouts are depicted in Figure 1.1 (a) in a sketch of Weinig [3] from 1940.



(a) Sketch of an isolated marine propeller (top), marine propeller with swirl recovery vanes (middle) and with pre-swirl vanes (bottom) from Weinig [3].



(b) An advanced propeller swirl recovery model in the NASA Lewis Research Center 8 x 6 foot supersonic wind tunnel [4].

Figure 1.1: Early sketch of pre-swirlers and swirl recovery vanes and later implementation of SRV.

In the light of the energy crisis of the early 1970's, NASA and industry gained a renewed interest in high-speed propellers for improved propulsive efficiency. This resulted in NASA Lewis Research Center's Advanced Turboprop Project (1976-1987) [4]. A part of this project explored the idea of SRV, shown in Figure 1.1 (b). The concept of the contra-rotating open rotor (CROR), where a second rotating stage recovers the swirl of the upstream rotor, also gained increased attention. It is generally thought that the CROR recovers swirl losses better, at the cost of increased complexity, with increases in cruise propulsive efficiency in the order of 8 % according to Strack et al. [5]. Research on the concept and integration of CROR has continued since, but after this first research effort on the aerial application of SRV, only recently research is resumed by Wang et al. [6] and Sinnige et al. [7].

1.1. PROPELLER PERFORMANCE AND THE SWIRL LOSS

A simple representation of a propeller is given by the classical propeller momentum theory. This theory is based on the assumption of a streamtube, extending from infinitely far upstream to infinitely far downstream, exactly enclosing the propeller disk. The flow is assumed to be inviscid and incompressible, rotation of the fluid is neglected and the axial velocity and pressure at each cross section of the streamtube is assumed to be uniform.

In general, the propulsive power P_p of a propeller is defined as the propeller thrust T_p multiplied by the undisturbed air speed V_∞ . It is the rate at which useful work is done. The propulsive efficiency η_p is defined as the ratio of the propulsive power to the shaft power P_s , the power required to turn the propeller. Three often used quantities describing propeller performance are the thrust coefficient C_T , power coefficient C_P and the advance ratio J defined as:

$$C_T \equiv \frac{T_p}{\rho_\infty n^2 D_p^4} \quad (1.1)$$

$$C_P \equiv \frac{P_s}{\rho_\infty n^3 D_p^5} \quad (1.2)$$

$$J \equiv \frac{V_\infty}{n D_p} \quad (1.3)$$

where ρ_∞ is the undisturbed air density, n the propeller rotational speed and D_p the propeller diameter. For the classical propeller momentum theory it can be derived that the propulsive efficiency, often called the ideal efficiency in case of the assumption of inviscid, incompressible flow, is equal to [8]:

$$\eta_p \equiv \frac{T_p V_\infty}{P_s} = \frac{C_T J}{C_P} = \left[\frac{1}{2} + \sqrt{\frac{1}{4} + \frac{2C_T}{\pi J^2}} \right]^{-1} \quad (1.4)$$

The torque applied to turn a propeller results in rotation of the fluid, the so-called swirl. The assumption of neglecting rotation of the fluid does not have to be made in order to reach a closed form solution for the propulsive efficiency. When a part of the shaft power is going into angular momentum of the fluid, the estimated propulsive efficiency for the same condition (same C_T and J) will be less. When assuming a uniform tangential velocity and removing the restriction of a uniform pressure distribution at each cross section of the streamtube, the following equation can be derived [8]:

$$\eta_p = \left[\frac{1}{2} + \sqrt{\frac{1}{4} + \frac{\pi^2}{4J^2} \left(1 - \sqrt{1 - \frac{16C_T}{\pi^3}} \right)} \right]^{-1} - \frac{J^2}{\pi^2} \left[\sqrt{1 + \frac{\pi^2}{J^2} \left(1 - \sqrt{1 - \frac{16C_T}{\pi^3}} \right)} - 1 \right] \quad (1.5)$$

Figure 1.2 shows the propulsive efficiency as predicted by Equation (1.4) and (1.5) versus the advance ratio for two thrust coefficients. Neglecting rotation of the fluid leads to an overprediction of the propulsive efficiency, especially for larger thrust coefficients. So, if this loss of shaft power that is put into angular momentum of the fluid could be recovered by means of SRV, a substantial increase of propulsive efficiency can be reached, especially for high thrust conditions.

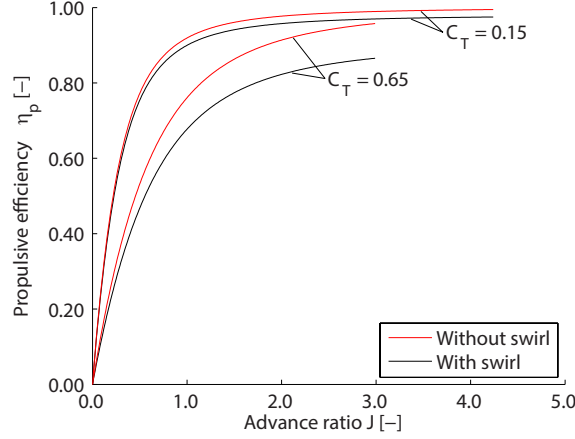


Figure 1.2: Propulsive efficiency as predicted by Equation (1.4) without the effect of swirl and by Equation (1.5) with the effect of swirl versus the advance ratio for two thrust coefficients.

1.2. SWIRL RECOVERY VANES STATE OF ART

There is only limited research on the aerial application of SRV: First, there is literature related to NASA Lewis Research Center's Advanced Turboprop Project consisting of experimental performance results by Gazzaniga and Rose [9], experimental noise results by Dittmar and Hall [10] and numerical Euler predictions by Yamamoto [11] and Miller [12]. Second, there is recent literature on SRV consisting of a computational fluid dynamics (CFD) analysis by Wang et al. [6] and an experimental investigation into the effect of SRV on the propeller performance, slipstream and noise by Sinnige et al. [7]. There is also research on the marine application of SRV, for instance a CFD and lifting line analysis by Çelik and Güner [13] and an experimental investigation on propeller stator interaction by Farnsworth et al. [14]. The main conclusions of this research will be discussed in this section.

The main goal of SRV is to increase the propulsive efficiency by generation of an additional thrust force on top of the propeller thrust. One can define the the propulsive efficiency increase by the addition of SRV $\Delta\eta_p$ as:

$$\Delta\eta_p = J \left(\left(\frac{C_T}{C_P} \right)_{\text{with}} - \left(\frac{C_T}{C_P} \right)_{\text{without}} \right) \quad (1.6)$$

where the subscripts *with* and *without* denote the results with SRV and without SRV respectively and when SRV are present C_T is defined as:

$$(C_T)_{\text{with}} = \frac{T_p + T_{SRV}}{\rho_\infty n^2 D_p^4} \quad (1.7)$$

where T_{SRV} is the SRV thrust force. Gazzaniga and Rose [9] present the only experimental performance results of a high speed propeller with SRV, conducted in the NASA Lewis supersonic wind tunnel. For its design cruise condition a 1.7% increase in propulsive efficiency by the addition of SRV is found. Miller [12] and Yamamoto [11] present numerical predictions of the same configuration by means of three dimensional Euler code. For the design condition an efficiency gain of 3.5% and 5.2% is found. They both ascribe the overprediction with respect to the wind tunnel test to the inviscid flow assumption in the Euler code. Figure 1.3 of Yamamoto [11] provides a comparison of measured and numerical swirl angle downstream of the propeller and downstream of the SRV. On average, about half of the swirl angle is recovered by the SRV, more for smaller radii and less more outward.

Contrary to the previous results, Wang et al. [6] is less positive on the performance gain of SRV. Although C_T increases by the addition of the SRV over the whole range of advance ratios, the required C_P increases even more, thereby lowering the overall propulsive efficiency for most of the range of advance ratios including the design point.

Although the density and Reynolds number differ from aerial applications, conclusions drawn from the marine application of SRV may still be applicable. Çelik and Güner [13] describes a procedure to design SRV for marine propellers. Previous research has shown that SRV can gain up to 5 – 7% in propulsive efficiency when the propeller loading is moderate to high. An advanced lifting line procedure is presented, and is val-

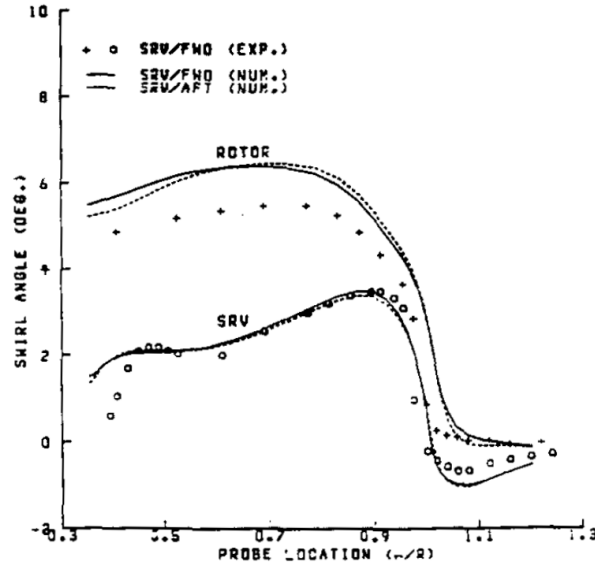


Figure 1.3: Numerical and experimental swirl angle versus radial position $0.25D_p$ behind the propeller and $0.59D_p$ behind the SRV from Yamamoto [11].

idated by means of transient CFD simulations. For two different designs, the lifting line procedure shows efficiency gains of 4.6% and 5.9% which are confirmed by the CFD results with very similar efficiencies of 4.4% and 5.6%. A general trend is found that the propulsive efficiency increases with increasing number of stator vanes up to nine. This information should be used with care since water blockage, especially in the hub region, may be caused by a large number of blades. According to Farnsworth et al. [14] full scale field tests have shown an efficiency improvement of 5% by pre-swirl vanes.

The effect on the propulsive efficiency of the axial distance between the propeller and SRV is not clear. While in Gazzaniga and Rose [9] no effect can be noticed, Miller [12] predicts a small decrease in efficiency gain with increasing axial distance and Yamamoto [11] predicts a small increase. A gain in propulsive efficiency is found for increasing axial distance in Çelik and Güner [13].

The effect of SRV on the propeller performance is very small. According to Gazzaniga and Rose [9], the addition of SRV results in a slightly lower rotor loading. Dittmar and Hall [10] investigated the interaction noise that might be generated by the propeller-SRV combination. Noise generated by advanced propellers is investigated because of the importance of low cabin noise during cruise. No increase in noise is found. The slight unloading of the propeller even results in a small noise reduction. In Sinnige et al. [7] also no measurable changes in propeller performance were found, concluding that the upstream effect of SRV is negligible. An increase in total sound pressure level of 2 to 6 dB was measured however, most probably resulting from the periodic impingement of the rotor blade wakes on the SRV.

The wind tunnel test of Gazzaniga and Rose [9] includes a propeller blade angle β_{75} study showing the propulsive efficiency of the propeller alone, the propulsive efficiency of the propeller-SRV combination and the power coefficient versus the advance ratio for a range of Mach numbers for the design SRV angle setting of $\beta_{SRV} = 86.1^\circ$. This data can be used to explain for which conditions SRV are beneficial by means of a propeller chart: Figure 1.4 presents a $C_P - J$, $\Delta\eta_p - J$ and $\eta_p - J$ plot. In the $C_P - J$ plot the experimental data is shown in black with different symbols for each Mach number. Clearly four different lines for the four propeller blade angles can be noticed, with a certain spread due to Mach number differences. For each measurement point the thrust coefficient for the propeller alone and the propeller-SRV combination is calculated using Equations (1.1) and (1.7) respectively. The C_T values for each data set of a specific (M, β_{75}) combination are linearly interpolated to find the (C_P, J) values corresponding to a range of constant C_T values. These are shown in blue and red symbols corresponding to the C_T of the propeller-SRV combination and propeller alone case respectively. Lines of constant C_T are then drawn by second order polynomial fits in blue and red respectively. Note that normally a propeller chart is plotted for a certain Mach number, Reynolds number Re (based on a characteristic length) combination according to Ruijgrok [15], but since the C_T isolines fit the data points for different (M, Re) values well, plotting for all available Mach numbers is preferred since it increases the range of the plot and makes the results and conclusions more generic.

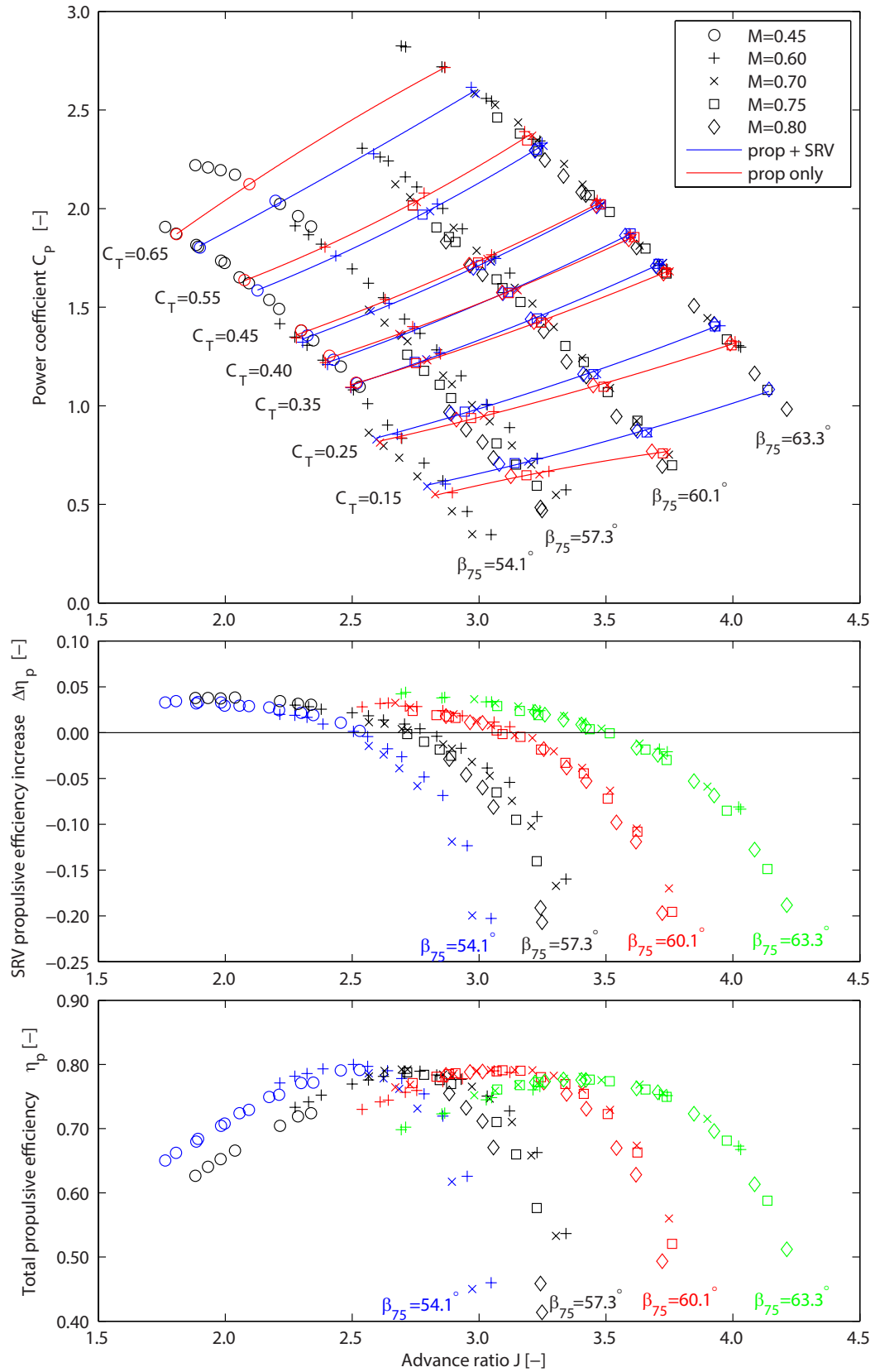


Figure 1.4: Propeller chart with and without SRV, SRV propulsive efficiency increase and propeller-SRV combination propulsive efficiency versus advance ratio for $\beta_{SRV} = 86.1^\circ$ derived from experimental data of Gazzaniga and Rose [9].

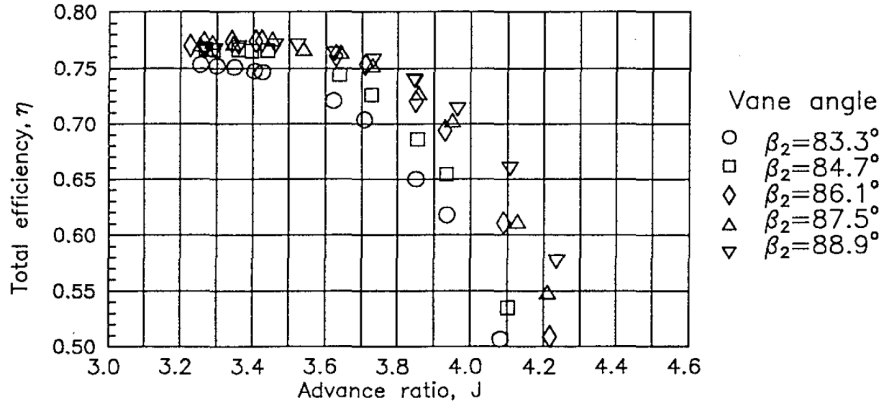


Figure 1.5: Propulsive efficiency of propeller-SRV combination versus advance ratio with propeller blade angle $\beta_{75} = 63.3^\circ$ and $M = 0.80$ for a range of SRV angles ($\beta_{SRV} = \beta_2$) from Gazzaniga and Rose [9].

Clearly a trend in the C_T isolines can be noticed: For relatively low values of C_P the available shaft power results in a lower C_T for the propeller-SRV combination than for the propeller alone case; a lower C_T at the same operating condition (M, Re, β_{75}, J) is equivalent to a lower thrust. At high values of C_P the reverse is true. A low shaft power leads to a weak swirl, and therefore the thrust of the SRV is less than its drag, while the reverse is true for a high shaft power. In general, the $C_P - J$ plot can be split up in two parts, roughly divided by the isoline of $C_T = 0.40$, where for conditions above this isoline, the thrust generated by the SRV is higher than its drag and where for conditions below this isoline, the drag of the SRV is higher than its thrust. Since the C_T isolines fit the data of the whole range of Mach numbers well, the statement that there is roughly a certain fixed threshold of thrust coefficient above which SRV are beneficial, is in general true irrespective of Mach number. The design condition ($C_P = 2.19, J = 3.26, \beta_{75} = 63.3^\circ, M = 0.80$) corresponds to a $C_T = 0.52$ which is in the area where SRV are beneficial.

The previous results are reflected in the $\Delta\eta_p - J$ plot, where $\Delta\eta_p$ is the propulsive efficiency increase by the addition of SRV. The area of $\Delta\eta_p > 0$ corresponds to the points of $C_T > 0.40$, the line $\Delta\eta_p = 0$ corresponds roughly to $C_T = 0.40$ and the area of $\Delta\eta_p < 0$ corresponds to $C_T < 0.40$. Another way of looking at it is that for each propeller blade angle β_{75} , the largest gains in propulsive efficiency can be found towards lower J , i.e. higher propeller speeds for a certain Mach number, only limited by the maximum propeller speed to avoid sonic tip speeds. A maximum of $\Delta\eta_p = 4.4\%$ can be found at $\beta_{75} = 63.3^\circ$ and $M = 0.60$ while also for $M = 0.45$ almost similar gains are reached at smaller blade angles. This means that especially during a high thrust flight condition, more thrust can be gained with a propeller equipped with SRV for the same power requirement.

The $\eta_p - J$ plot, where η_p is the propulsive efficiency of the propeller-SRV combination, puts the previous results in perspective. When comparing the $\eta_p - J$ plot with the $\Delta\eta_p - J$ plot, the observation can be made that the highest increases in propulsive efficiency by the SRV do not correspond to the highest values of the propulsive efficiency of the propeller-SRV combination. The maxima of the $\eta_p - J$ curves correspond to slightly positive $\Delta\eta_p$ values. The effect of the SRV on the $\eta_p - J$ curve is a decrease of slope to the left of the maxima and an increase of slope to the right of the maxima. From this it can be concluded that SRV have a positive effect on the propulsive efficiency as long as a propeller blade angle β_{75} is selected such that the current advance ratio is the same or lower than the advance ratio corresponding to the $\eta_p - J$ curve's maximum of that β_{75} .

Figure 1.5 presents $\eta_p - J$ curves, where η_p is the propulsive efficiency for the propeller-SRV combination, for a range of SRV angles β_{SRV} for β_{75} and M corresponding to the design condition. This plot is part of the vane angle study. The previous results correspond to the design SRV angle of $\beta_{SRV} = 86.1^\circ$. This figure shows the importance of a correct SRV angle. A decrease in SRV angle to $\beta_{SRV} = 83.3^\circ$ results in a propulsive efficiency decrease over almost the whole range of advance ratios that is larger than the gain in propulsive efficiency by the SRV. In other words, a decrease in β_{SRV} of less than 3° results in SRV that almost only produce net drag.

1.3. OPPORTUNITIES AND CHALLENGES OF SRV FOR A WING-MOUNTED TRACTOR-PROPELLER

Up to date, the focus of aerial research on swirl recovery vanes has been on enhancing the propulsive efficiency of an isolated propeller and nacelle. The conclusions of such research are directly applicable if the propulsor is located away from the wing, for instance if it is fuselage mounted. When a wing is introduced in the slipstream of the propulsor, which is the case for a wing-mounted tractor-propeller, conclusions drawn on SRV in isolated condition may not hold. Installation effects of propellers are well described in literature. A short overview is given in this section, giving insight in possible opportunities and challenges of integration of SRV behind a wing-mounted tractor-propeller. Two main effects will be described, the effect of the wing induced field on the propeller and the effect of the propeller slipstream on the wing.

When a loading is present on the wing, a wing-mounted tractor-propeller operates in the induced velocity field of the wing. A positive lift distribution on the wing results in an upwash, so the propeller inflow is under an angle with respect to the propeller axis of rotation. The inflow conditions are almost equivalent to an isolated propeller under an angle of attack, except for the local deviation in the induced velocity field because of the local deviation of wing loading by the propeller slipstream according to Veldhuis [16]. This latter small effect is for instance visible in the work of Stuermer [17] which provides isolated and installed propeller loading for the tractor-propeller wing configuration of Samuelsson [18]. This specific wing is symmetric and results in a sinusoidal propeller thrust at zero wing angle of attack.

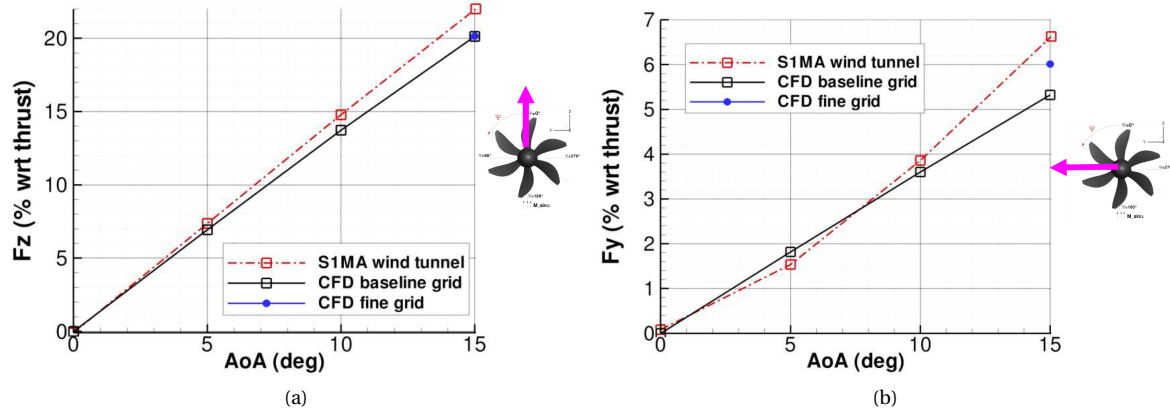
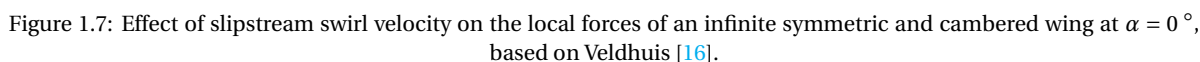


Figure 1.6: Experimental and numerical comparison of isolated APIAN propeller (a) vertical load with angle of attack and (b) lateral load with angle of attack from Beaumier [19].

The main effect can be described by looking at an isolated propeller under an angle of attack. Figure 1.6 shows the variation of the in-plane load with angle of attack for the APIAN propeller from Beaumier [19]. The vertical load increases to about 15% of the thrust at $\alpha = 10^\circ$ and the lateral load to about 4%. This is the result of variation in blade loading with azimuthal position due to varying angle of attack. For the propeller as a whole, this results in in-plane forces and out-of-plane moments. The total pressure rise will be larger on the down-going blade side than on the up-going blade side. The slipstream will also be altered globally by the angle of the undisturbed flow. SRV experience this altered, circumferentially non-uniform slipstream. The question can be raised whether the SRV performance is affected by such a circumferentially non-uniform flow field.

Second, a wing-mounted propeller in tractor configuration has an effect on the wing by its slipstream. The focus is on the situation that the slipstream impinges on a part of the wing at zero incidence. From the simulations of Stuermer [17] of the configuration of Samuelsson [18] it can be concluded that the lift distribution is mostly influenced directly behind the propeller, where the slipstream impinges on the wing. As a result of this changed loading and flow pattern, the lift distribution outside of the slipstream is also affected. The first main effect on the wing lift and drag distribution is by the locally increased axial velocity in the slipstream. This leads to an increase in resultant forces on that part of the wing. The second main effect is that of the tangential velocity component in the slipstream: For the symmetric airfoil of this wing a minimum in lift occurs on the propeller downward side due to the induced negative angle of attack by the tangential velocity in the slipstream. A maximum occurs on the upward side. Figure 1.7 shows a sketch of



The earlier described swirl recovery effect of the wing results in an effective drag reduction when the propeller is on. When considering a wing-mounted tractor-propeller, the inboard-up rotating propeller is better for a lower drag than the outboard-up rotating propeller for two reasons: First, because the loading on the inboard side is higher than on the outboard side when the propeller is off, the largest reduction in drag is reached when on the higher loaded side the angle of attack increases and on the lower loaded part the angle of attack decreases, so when the propeller is rotating inboard-up. This can be concluded from Figure 1.7 for the cambered airfoil which represent the case that loading is present, since the increased loading on the already higher loaded side leads to an increased forward drag component and the decreased loading on the lower loaded side leads to a decreased downstream drag component. Second, the induced drag is lower because of the more elliptic lift distribution [16]. Still, the spanwise loading with an inboard- or outboard-up rotating propeller is far from ideal. The wing shape parameters could be adapted to optimize it for minimum drag at a given design lift coefficient. In this optimization not only induced but also viscous drag should be considered. From Veldhuis [16] it can be concluded that a small performance improvement can be obtained from such an optimization.

The difference in drag between an inboard-up and outboard-up turning propeller is especially important for airplane with co-rotating propellers on the left and right wing-half. Often co-rotating propellers are applied because of reduction of cost of production and maintenance. An asymmetry in drag results in additional trimming drag. Also the asymmetry in lift distribution with its corresponding rolling moment results in trimming drag.

The slipstream of a propeller with SRV has a reduced or almost completely removed swirl velocity component. Any of these asymmetric effects will be reduced or removed. Also the wing swirl recovery will be reduced or removed, and therefore the wing drag will be higher than for the propeller without SRV. The dynamic pressure of the slipstream will likely be a little lower, because of a small total pressure loss, but the axial velocity may be increased, leading to higher local lift and drag forces. Kroo [20] points out that the performance advantages of a CROR may be less for well-integrated wing-propeller designs than in isolated systems. This same conclusion may be drawn when the CROR is replaced by a propeller with SRV.

1.4. RESEARCH OBJECTIVE

The potential of swirl recovery vanes to increase the propulsive efficiency, the limited research on SRV in general and the opportunities and challenges of SRV for a wing-mounted tractor-propeller lead to the following objective of this MSc thesis:

The objective of this research is to gain an improved understanding of the aerodynamic interaction between the propeller and swirl recovery vanes in an isolated configuration and wing-mounted tractor arrangement in the cruise condition and in a high-thrust condition.

This study is realised by performing a series of transient Reynolds-averaged Navier-Stokes (RANS) CFD simulations of a propeller with and without swirl recovery vanes in an isolated and installed configuration. A number of goals are set to reach this objective:

- Perform isolated propeller simulations to validate the propeller performance and slipstream.
- Perform isolated propeller-SRV simulations to investigate the performance benefit of SRV, the aerodynamic interaction between the propeller and SRV and the effect of SRV on the slipstream.
- Perform propeller-SRV-wing simulations to investigate the effect of SRV on the overall performance of the system and investigate the aerodynamic interaction of the propeller, SRV and wing.

The original idea was to use an existing SRV design for all simulations, the APIAN-INF SRV designed by van Kuijk [21] for the APIAN propeller and experimentally tested in Sinnige et al. [7]. However, from isolated propeller-SRV simulations it is found that its propulsive efficiency benefit is much lower than predicted by its design tool. This has led to two additional goals:

- Construct an SRV design procedure for isolated propeller-SRV configurations.
- Validate the SRV design procedure by means of isolated propeller-SRV simulations.

1.5. THESIS OUTLINE

The body of this thesis consists of this introduction and six more chapters. Chapter 2 presents the propeller CFD model. For this model of the isolated propeller the majority of the mesh and solver settings are chosen and the resulting performance and slipstream are validated by means of a comparison of these in a mesh dependency study with experimental data. Chapter 3 provides a discussion on the APIAN-INF CFD model. This model consists of an isolated propeller and SRV as tested in the APIAN-INF wind tunnel test described in Sinnige et al. [7]. A mesh dependency study for the SRV and rotor-stator specific solver settings are discussed in this chapter and the propeller performance and slipstream are compared to the experimental results. Due to the inadequate SRV performance, in Chapter 4 an SRV design procedure is introduced and a number of SRV designs are proposed, designed for the propeller slipstream results of Chapter 2. For two of these designs in Chapter 5 a propeller-SRV CFD model is constructed and their performance is discussed and compared to the design performance. For one of the designs, optimised for cruise, the slipstream is discussed as well. This latter design is used for the propeller-SRV-wing CFD model in Chapter 6 for which the effect of SRV on the overall performance of the system is investigated and the aerodynamic interaction of the propeller, SRV and wing is described. Finally, the conclusions drawn throughout the work are stated in Chapter 7, including a number of recommendations for future research.

2

PROPELLER CFD MODEL

Throughout this research a single propeller CFD model is used for all simulations. The purpose of this chapter is to explain the characteristics of this model and discuss the choice of a suitable mesh. Experimental data is used for validation. Furthermore, the resulting wake characteristics are determined in order to be used in following chapters. First, the geometry of the propeller model is presented in Section 2.1. The characteristics of the corresponding mesh are given in Section 2.2. The choice of solver and corresponding settings are explained in Section 2.3 and the validation and wake results are discussed in Section 2.4.

2.1. GEOMETRY

The geometry of the propeller model can be split in the propeller geometry and fluid domain geometry from which this propeller geometry is subtracted.

2.1.1. PROPELLER GEOMETRY

Throughout this research, the 6-bladed propeller of the European APIAN project is used. The APIAN project dealt with the investigation of the acoustic and aerodynamic installation effects of this propeller. As part of this program, wind tunnel measurements in the DNW-HST transonic wind tunnel were performed on the isolated 1/8 scaled APIAN propeller for a range of Mach numbers between 0.20 and 0.78 with varying incidence angle and tunnel pressure as described by Custers and Elsenaar [22]. This geometry is chosen because of known blade pressure and slipstream data from this experiment, and because of previous research on swirl recovery vanes with this propeller by van Kuijk [21]. Figure 2.1 shows an isometric view of this propeller and Table 2.1 summarizes the key propeller properties.

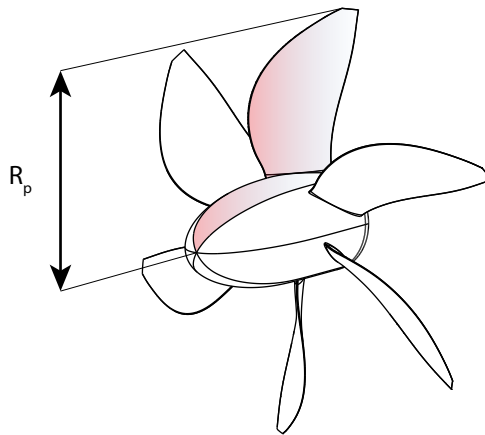


Figure 2.1: APIAN propeller, spinner and hub at $\beta_{75} = 40.4^\circ$ with the wedge of the propeller CFD model coloured in red.

Table 2.1: Overview of APIAN propeller properties.

Variable	Value
Scale	1/8
Propeller radius R_p	250 mm
Propeller blade pitch β_{75} at $0.75R_p$	40.4 °
Propeller blade chord c_{75} at $0.75R_p$	94.2 mm

2.1.2. DOMAIN GEOMETRY

Since the wake of a propeller with axis-symmetric nacelle is cyclic with the number of the blades, only a single blade is modelled with appropriate boundary conditions. For this propeller a 60 ° wedge suffices as is shown in red in Figure 2.1. The domain for the propeller model is also a wedge of the same angle and consists of three regions: an outer region, rotating region and wake region, see Figure 2.2 for an overview of these regions. The dimensions of the outer region are chosen to be sufficiently large with respect to the propeller radius R_p in order to minimize influencing the flow properties near the propeller by the boundary conditions. The upstream dimension and radial dimension are similar to the domain used in Ortun et al. [23] for the APIAN propeller, while the downstream dimension is larger to get a less disturbed flow field at the outlet. The rotating region encapsulates the propeller, spinner and hub wedge and is used for refinement of the mesh and for simulation of the propeller motion by various means. This region is of special interest because it remains unchanged for all subsequent simulations. The wake region is defined for mesh refinement to capture the wake flow properties with a high resolution.

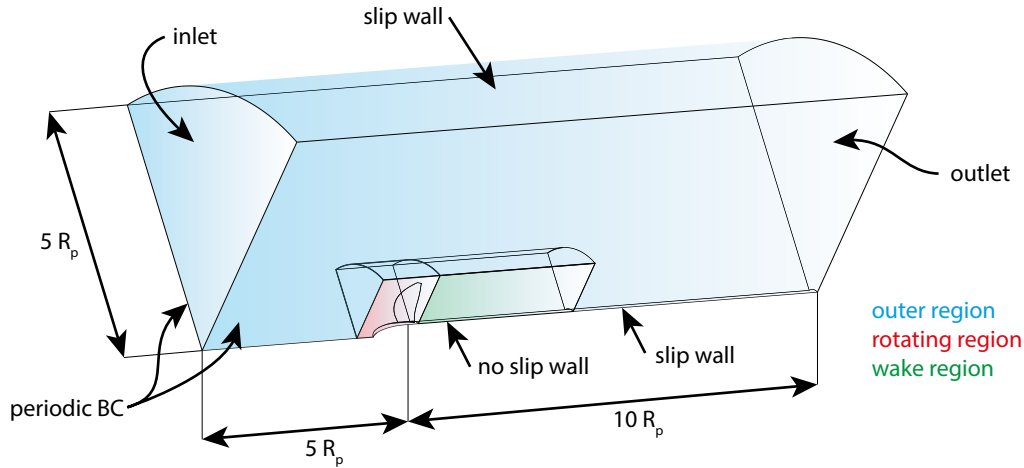


Figure 2.2: Domain, regions and corresponding boundary conditions of the propeller CFD model.

2.2. MESH

The mesh of the various propeller domain regions is constructed by means of ANSYS® Meshing [24]. This tool is used throughout this research.

2.2.1. GENERAL MESH PROPERTIES

An overview of the general mesh properties can be found in Table 2.2. The ANSYS meshing settings that deviate from the default settings are also given. The unstructured mesh is made up of a triangular wall mesh, 12 layers of semi-structured prismatic elements adjacent to all no-slip walls and tetrahedral elements in the remainder of the domain. Wall refinement of all no-slip walls, volume refinement of the rotating and wake region, a first layer thickness of the inflation layer, a growth rate of the inflation layer and a growth rate of the remainder of the mesh control the mesh density in the whole domain. The first layer thickness is tuned to comply with the y^+ requirement of the turbulence model and will be discussed in Section 2.3.2. The number of inflation layers is tuned to encapsulate the boundary layer on the blades and nacelle. The meshes on the periodic boundaries are conformal to ensure that no interpolation of flow quantities is required. The rotating

Table 2.2: Overview of propeller CFD model general mesh settings.

Variable	Value
Mesh classification	Unstructured
Element type	
Volume	Tetrahedral
Wall	Triangular
Growth rate	1.2
Inflation layer	
Element type	Prismatic
Nr. of layers	12
First layer thickness	0.10 <i>mm</i>
Growth rate	1.2
Advanced Size Function	Proximity and Curvature
Relevance Center	Fine

region and the outer and wake region are adjacent to each other such that a sliding-mesh technique can be used for motion of the rotating region. Two independent refinements of the walls are applied: refinement of the leading and trailing edge and tip of the blade and refinement of the remainder of the no-slip walls including the remainder of the blade. These refinements and the volume refinement of the rotating and wake region are varied in the mesh dependency study.

2.2.2. MESH DEPENDENCY STUDY

A mesh dependency study is performed for the propeller CFD model to investigate the solution dependency on the mesh and to find a compromise between mesh size and accuracy. Table 2.3 gives an overview of the mesh size and refinement dimensions of the candidates. Refinement of all no-slip walls and refinement of the rotating and wake region volume is considered separately. Six different meshes are created with various refinement. Three volume refinements are applied together with a coarse wall refinement to investigate solely the effect of volume refinement. Also three wall refinements are applied together with a coarse volume refinement to investigate solely the effect of wall refinement. A fine wall, fine volume refinement mesh is created to investigate any cross coupling of wall and volume refinement. To put the number of nodes in perspective, a 6,327,540 node mesh was created in Roosenboom et al. [25] for a complete eight bladed propeller without inflation layer. No inflation layer was created since the propeller was modelled with slip walls. The focus of Roosenboom et al. [25] is on the slipstream and interaction with the wing, comparable to the focus of this work. Taking the different number of blades into account, this is very comparable to the number of nodes of the rotating region with coarse wall mesh and fine volume mesh.

Table 2.3: Overview of mesh size and refinement dimensions for the mesh dependency study of the propeller CFD model.

wall refinement	volume refinement	# nodes total	# nodes rotating region	volume size [mm]	wall size [mm]	i.e. & t.e. size [mm]
Coarse	Coarse	1,013,031	705,986	10	3.0	0.25
Coarse	Fine	1,278,062	763,904	7	3.0	0.25
Coarse	Extra Fine	1,960,195	927,949	5	3.0	0.25
Fine	Coarse	1,443,191	973,772	10	2.0	0.20
Fine	Fine	1,709,646	1,032,574	7	2.0	0.20
Extra Fine	Coarse	2,148,782	1,434,573	10	1.5	0.15

2.3. SOLVER

A limited number of the CFD solvers are available because of lack of licences and restrictions by organizations or governments. The solver chosen for this research is the commercial solver ANSYS® CFX. Initially the commercial solver ANSYS® Fluent was considered as well. Both are general purpose CFD software with many similarities, but using CFX is advantageous for these simulations because of the enhanced capabilities for modelling turbo-machinery. Open-source solver SU2 was also considered initially, but does not yet have a sliding-mesh or Chimera capability as described in Francois et al. [26] necessary to model the rotating do-

main. One of the main arguments for using CFX is the capability of connecting two dissimilar pitch domains by means of transformation of the flow quantities. This is for instance necessary when a single blade and vane of a propeller and SRV are simulated when the total number of blades and vanes are dissimilar. Section 3.3 discusses this problem in more detail. Information on the governing equations, discretization schemes, turbulence model, boundary conditions and interfaces in the sections below can be found in [27], unless referenced otherwise.

2.3.1. GOVERNING EQUATIONS AND THEIR DISCRETIZATION

Most flows are turbulent. As long as the Reynolds number is sufficiently large, a flow over a body will become turbulent. A turbulent flow can be considered as a mean flow plus unsteady fluctuations. A turbulent flow is a solution of the Navier-Stokes equations, and can be simulated by solving these equations numerically, the so called direct numerical simulation (DNS). A turbulent flow consists of fluctuations with a very large range of length and time scales. In order to grasp a turbulent flow in a numerical simulation, the spatial discretization should be sufficiently small to capture the smallest length scale and the domain should be sufficiently large to capture the largest length scale. This same principle also holds for the temporal discretization. For flows with relatively large Reynolds number, the required spatial and temporal resolution become too large to realistically solve for the foreseeable future. In order to reduce the computational cost, in a large eddy simulation (LES), the mean flow and the unsteady large-scale and intermediate-scale motions are directly calculated, while the effect of small-scale fluctuations on the mean is modelled. This results in a modelling error. One step further down are the simulations based on the Reynolds-averaged Navier-Stokes (RANS) equations, where the effect of fluctuations of all scales on the mean flow is modelled.

Many flows can be simulated well with LES which RANS simulations cannot, i.e. flows with large separation, bluff-body flows and transition, although this comes at the expense of computational resources according to Davidson [28]. In LES there is especially a high mesh requirement near the wall which also limits the required time step according to Fujii [29]. LES is still in a stage of fundamental research application, for instance to simulate the axis-symmetric and statistically steady turbulent flow between a rotating and a fixed disk as a representative of the flow in unshrouded rotor-stator configurations in Andersson and Lygren [30]. The advancement of computer power does cause a shift to hybrid LES-RANS simulations, where the RANS formulation is applied near the solid surface to reduce the dense mesh requirement and the LES formulation is applied to the outer regions to capture the larger turbulent structures, i.e. regions of separated flow [29]. Representative publications on propeller CFD simulations of the last decade are almost all based on RANS simulations. In particular, Roosenboom et al. [25] regards DNS and LES as too computationally expensive and restricts itself to a RANS simulation although more computational resources are available than for this work. This research will also be restricted to RANS simulations.

In CFX the RANS equations are discretized with an element-based finite volume method. CFX is a node centred solver, meaning that the solution variables are stored in the mesh node instead of the cell centroid as in Fluent. Therefore the mesh node count correlates to the computational time and this node will be specified for each different mesh. A control volume is constructed around each mesh node and is used to conserve mass, momentum and energy. For the discretization of the advection term, the high resolution scheme is chosen, an upwind scheme which is as close to 2nd order accurate by means of the Barth-Jespersen boundedness principle, avoiding non-physical oscillations in regions of rapid variation of the solution. All diffusion derivative and pressure gradient terms are evaluated using shape functions at the integration point. CFX is a pressure-velocity coupled solver, using a fully implicit discretization of the equations at each time step. In a steady state analysis, the time step is used to physically guide the solution to a steady state. In a transient analysis the solution is calculated at each time step. The chosen scheme for the transient term uses the dual time-stepping method and is second order accurate.

The flow is considered compressible for all simulations. The equation of state is modelled as an ideal gas to calculate the local variations in density.

2.3.2. TURBULENCE MODEL

For simplicity, a fully turbulent turbulence model is chosen and thus transition will be neglected. In recent literature, propeller CFD simulations are often but not exclusively conducted with one-equation Spalart-Allmaras type turbulence models like Francois et al. [26], Roosenboom et al. [25] and Stuermer [17], resolving the whole boundary layer including viscous sub-layer. This requires a very dense grid near walls ($y^+ < 1$). In order to lower the computational cost, in Roosenboom et al. [25] and Stuermer [17] the propeller is modelled with slip walls to avoid the dense grid requirement and the boundary layer of the other aerodynamic

surfaces is fully resolved. In order to lower the computational cost and still model a boundary layer on the propeller blades, part of the boundary layer can be modelling with wall functions. For this research the $\kappa - \omega$ turbulence model with shear stress transport (SST) is chosen with automatic near wall treatment. This type of model is for instance used in Ortun et al. [23] for the simulation of the APIAN propeller with non-zero inflow angles. In earlier simulations by the author of the APIAN propeller in ANSYS Fluent, both $\kappa - \omega$ SST and Spalart-Almaras turbulence models with wall functions were used and very similar performance results were obtained. The automatic near wall treatment automatically switches between resolving the viscous sub-layer and using scalable wall functions, depending on the y^+ value at the wall. For this research, the mesh is constructed such that the average y^+ value is in the order of 25 and the maximum in the order of 50, and thus wall functions are automatically used.

2.3.3. BOUNDARY CONDITIONS AND INTERFACES

An overview of the specified boundary conditions can be found in Figure 2.2. The following boundary conditions are specified:

Inlet At the domain inlet a total pressure jump $\Delta p_{t_{inlet}}$ with respect to the undisturbed static pressure p_∞ is set to reach the undisturbed air speed V_∞ . Furthermore, the undisturbed total temperature T_{t_∞} is specified. These quantities vary slightly with advance ratio and are chosen to match the wind tunnel experimental results, see Table 2.4. The inlet turbulence is set to the default low intensity setting, resulting in a turbulence level of $T_u = 1\%$ and eddy viscosity ratio of $\frac{\mu_t}{\mu} = 1$. Contrary to the name, the low intensity setting may still be considered as quite high compared to the turbulence of undisturbed flow in wind tunnels. Values of the turbulence level $< 0.1\%$ can be found in low turbulence wind tunnels according to Bearman and Morel [31], for instance $< 0.02\%$ for Sinnige et al. [7]. Values of Eddy viscosity ratio $\frac{\mu_t}{\mu} = 0.1 - 1$ are common for external flow while values of $\frac{\mu_t}{\mu} = 1 - 10$ are found for external flows in wind tunnel according to Saxena [32]. In order to check the influence of these turbulence quantities on the propeller performance, they are changed to $T_u = 0.05\%$ and $\frac{\mu_t}{\mu} = 0.1$ for one simulation.

Outlet At the domain outlet, the static pressure is prescribed to be equal to the undisturbed static pressure on average.

Slip wall The top of the domain and the rear end of the nacelle wall are specified as slip walls. The rear end of the nacelle is adjacent to the outlet and a boundary layer on that surface is not desirable in order to reach outflow that is as uniform as possible.

No slip wall The propeller blade, spinner, hub and the front end of the nacelle wall are set to no slip walls. In order to investigate the effect of fully turbulent modelling of the propeller blade, spinner and hub, these surfaces are changed to slip walls for one simulation.

Periodic On the sides of the domain a conformal periodic boundary condition is specified. This is possible because of the conformal mesh on these boundaries and desirable to avoid interpolation errors.

The outer and wake region are part of the same mesh, forming a stationary region together. The rotating region is connected to the stationary region by means of interfaces. These interfaces are general grid interfaces (GGI), meaning that the mesh on both sides of the interface do not match and the flow quantities are interpolated. The blade motion in the rotating region is either modelled by a reference frame transformation with frozen rotor interfaces, or by rotation of the mesh with transient rotor-stator interfaces. In case of the latter, both sides of an interface do not necessarily overlap and thus flow quantities of the non-overlapping regions are rotated around the rotation axis before interpolation.

Table 2.4: Overview of undisturbed flow quantities and inlet total pressure jump.

J [-]	Experimental data	V_∞ [m/s]	p_∞ [Pa]	ρ_∞ [kg/m ³]	T_{t_∞} [K]	$\Delta p_{t_{inlet}}$ [Pa]
0.95	yes	67.92	99461.2	1.2084	289.06	2784.91
1.00	yes	68.00	99302.2	1.2026	289.00	2780.46
1.30	yes	68.04	99307.3	1.2066	289.04	2791.74
1.60	no	68.00	99300.0	1.2026	289.00	2780.40
1.80	no	68.00	99300.0	1.2026	289.00	2780.40

2.3.4. SOLVING STRATEGY

The solving process is split in two parts: First a good initial condition is created by means of a simulation with frozen rotor interfaces. In this steady state simulation the time advances with large time steps. This solution is only appropriate if the propeller and nacelle are modelled with axis-symmetric inflow and without wing according to Veldhuis and Luursema [33]. It deviates considerably from the correct solution when SRV and the wing are added in following simulations, but suffices as initial condition. This initial condition is then used for a transient simulation with transient rotor-stator interfaces. The time-steps used throughout this research correspond to a 1° or 2° propeller rotation. This depends on the number of inner loop iterations for each time-step. If the number of inner loop iterations becomes too large (about > 10), the time-step can be reduced to reach better accuracy in possibly lower simulation time. 1° equivalent time-step is common in literature, for instance Roosenboom et al. [25], Stuermer [17] and Ortun et al. [23]. Depending on the downstream extend of the aerodynamic surfaces in the remainder of the domain, their upstream effect and the mean flow velocity, between 2 and 4 propeller rotations are required to reach periodic behaviour in the flow quantities.

2.4. RESULTS

The results of the propeller CFD model consists of results related to the mesh dependency study for one advance ratio, results for different inlet turbulence quantities and blade turbulent modelling for one advance ratio for the chosen mesh and results for different advance ratios for the chosen mesh. For three advance ratios including that of the mesh dependency study as indicated in Table 2.4, experimental data from the DNW-HST wind tunnel test of Custers and Elsenaar [22] is presented alongside.

2.4.1. MESH DEPENDENCY STUDY

Table 2.5 presents propeller performance quantities for the different mesh refinements, including deviations from the experimental data for an advance ratio of $J = 1.00$ at $M_\infty = 0.200$. The effect of wall refinement and volume refinement are independently measured. The effect of volume refinement on all performance quantities is very small with a maximum variation of 0.2% between the coarse and extra fine volume mesh. For any volume refinement and the coarse wall refinement, the thrust coefficient (including spinner and hub and corrected for base area pressure) C_T is under-predicted, while the power coefficient C_P is over-predicted. This results in an under-prediction of the propulsive efficiency η_p . With increasing wall refinement the deviation of C_T decreases, the deviation of C_P increases, and η_p stays almost constant. It is likely that with increasing wall refinement the prediction of C_T goes asymptotically to a small under-prediction with respect to the experimental value, while the C_P goes asymptotically to a constant over-prediction of 5 – 6%. The under-prediction of the C_T may be explained by the decambering effect by the thicker displacement thickness due to the fully turbulent modelling of the boundary layer. To put the under-prediction of the thrust in perspective, in Stuermer [17], a 5.5% over-prediction of the thrust is found when modelling the blades of an isolated propeller as inviscid walls; due to the lack of a boundary layer, the blade profiles are not de-cambered and thus the thrust is higher. The over-prediction of the C_P may be explained by the larger viscous drag and thus larger shaft power by the fully turbulent modelling of the boundary layer.

Table 2.5: Propeller performance quantities of mesh dependency study for the propeller CFD model including deviations from the experimental data at $J = 1.00$ and $M_\infty = 0.200$.

Wall refinement	Volume refinement	C_T [-] (deviation [%])	C_P [-] (deviation [%])	η_p [-] (deviation [%])
Coarse	Coarse	0.5318 (–3.57)	1.015 (+2.95)	0.5232 (–6.32)
Coarse	Fine	0.5317 (–3.59)	1.015 (+2.95)	0.5230 (–6.36)
Coarse	Extra Fine	0.5310 (–3.72)	1.013 (+2.75)	0.5231 (–6.34)
Fine	Coarse	0.5375 (–2.54)	1.025 (+3.96)	0.5237 (–6.23)
Fine	Fine	0.5372 (–2.59)	1.024 (+3.86)	0.5234 (–6.29)
Extra Fine	Coarse	0.5414 (–1.83)	1.031 (+4.57)	0.5241 (–6.16)

Figure 2.3 through 2.5 present the pressure coefficient distribution at 0.65 , 0.75 and $0.85R_p$ for the different mesh refinements, and also includes experimental data. For convenience, the wall refinement is indicated by different coloured curves while volume refinement is indicated by different curve styles. Again, the effect of volume refinement is very small; the dashed and dashed-dotted curves coincide with the solid curves. The effect of wall refinement is especially noticeable at $0.85R_p$ but also present at the lower radii, where a second

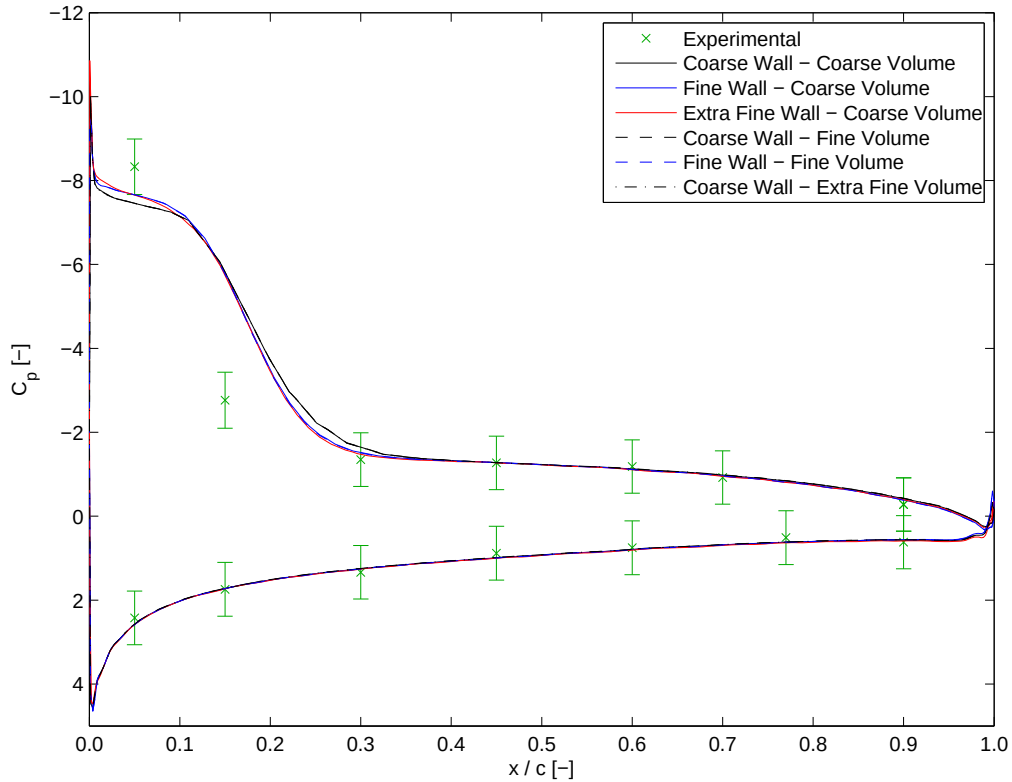


Figure 2.3: Distribution of pressure coefficient $C_p = \frac{p - p_\infty}{\frac{1}{2} \rho_\infty V_\infty^2}$ defined with undisturbed conditions at $0.65R_p$ on the blade for the various wall and volume refinement of the mesh dependency study including experimental data with indication of the variation over time.

suction peak increases with increasing mesh refinement. This can directly be related to the increase in C_T with wall refinement. The numerical data compares reasonably well to the experimental data, although the latter shows quite large variations in time, indicated by the vertical bars. The region of low pressure aft of the leading edge pressure peak is smaller in the experimental data at 0.65 and $0.75R_p$. The magnitude of the second suction peak at $0.85R_p$ is slightly under-predicted compared to the experimental data. This second suction peak is a result of the formation of what Schüle et al. [34] describes as a 3-D conical separation vortex. This vortex structure is analogous to the delta-wing vortices and is a result of a high angle of attack and large leading edge sweep angle. Figure 2.7 shows six different streamline images of the blade. The top left image shows the 3D velocity streamlines around the blade in a reference frame fixed to the blade going through a plane cutting the blade at $0.95R_p$. The 3-D conical separation vortex structure can be observed, originating from the point where the leading edge sweep turns sign and leaving the blade surface near the blade tip. This structure moves to higher blade radii with increasing wall refinement. The top middle image shows the wall shear streamlines on the blade. Clearly the region of the separation vortex stands out from the attached flow at lower radii and at chordwise locations more downstream of the vortex structure. The top right image and all bottom images show the velocity surface streamlines in a reference frame attached to the blade on surfaces 0.5 , 1.0 , 3.0 and 6.0 mm from the blade. At 0.5 mm from the blade, the velocity streamlines form a pattern very similar to the wall shear streamlines as expected. Moving away from the surface, the region of the separation vortex becomes smaller and moves to larger radii: The thickness of the vortex increases with increasing blade radius and is indeed conical in shape. At 6.0 mm from the blade, the streamlines are almost parallel to the blade sections and encapsulate the vortex structure completely. The magnitude of the velocity close to the surface in the vortex structure and also above the vortex structure is very high. This corresponds to the second suction peak which is especially visible in Figure 2.5. Since the experimental data also shows this suction region, it is likely that such a separation vortex was also present in the wind tunnel test.

The DNW-HST windtunnel test also consists of time-average wake measurements. The rake was positioned in a plane downstream of the propeller as sketched in Figure 2.6. From this plane, flow quantities are

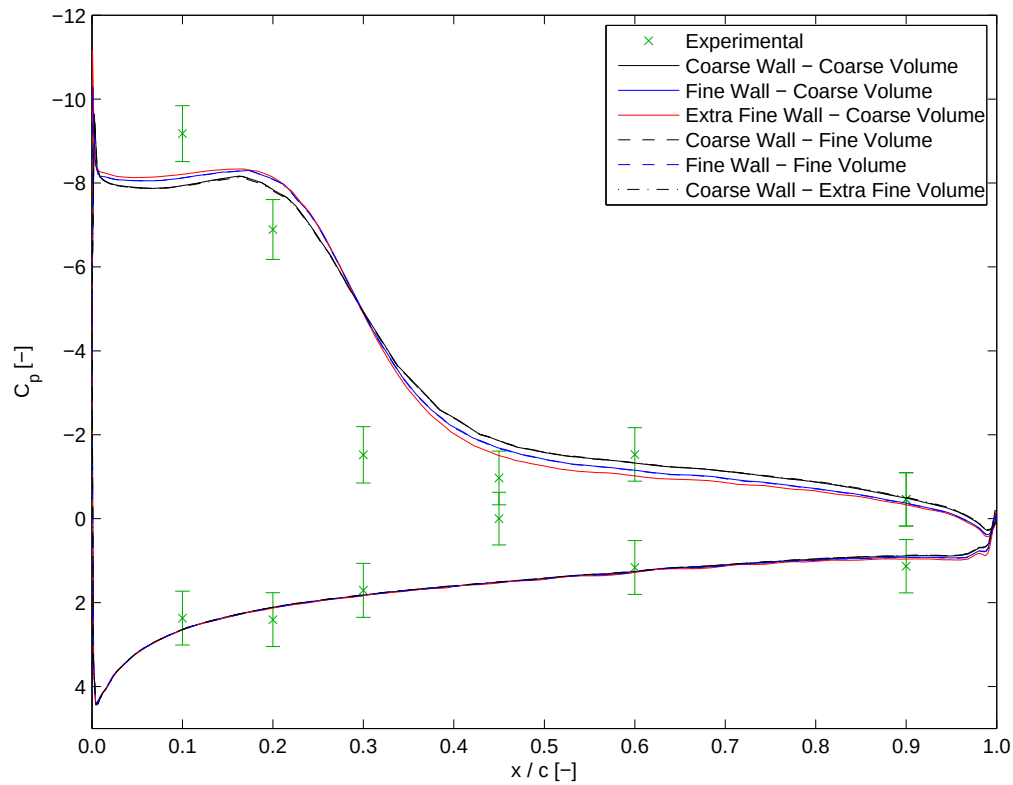


Figure 2.4: Pressure coefficient C_p distribution at $0.75R_p$ on the blade for the various wall and volume refinement of the mesh dependency study including experimental data with indication of the variation over time.

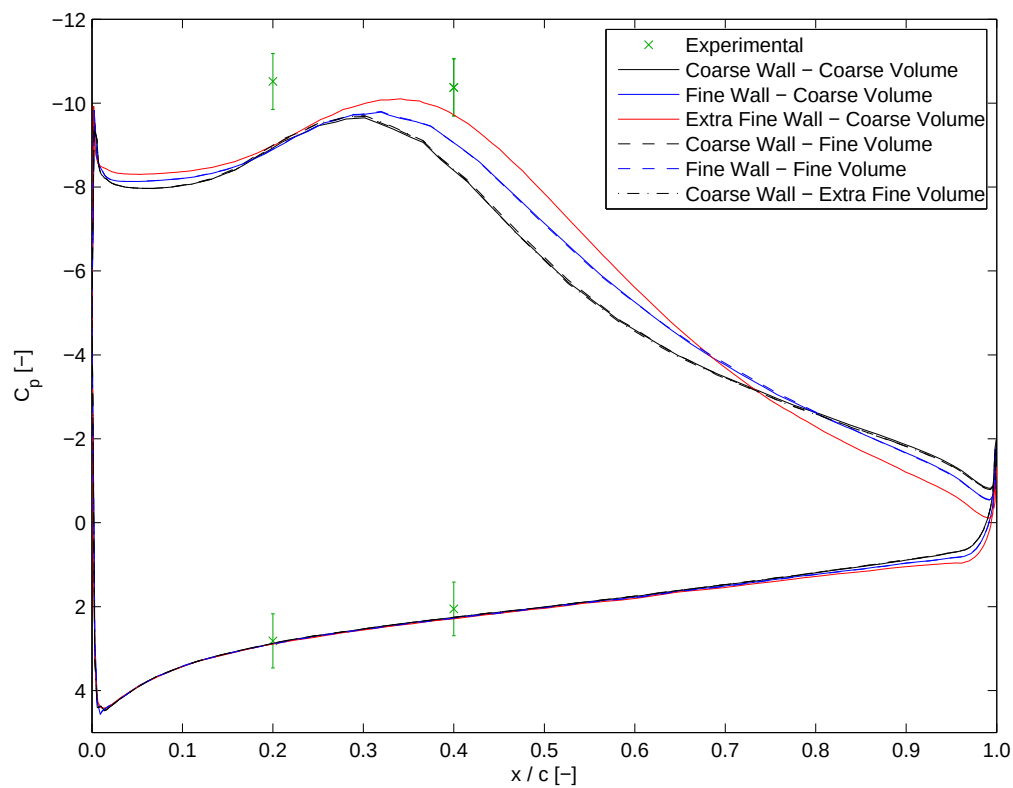


Figure 2.5: Pressure coefficient C_p distribution at $0.85R_p$ for the various wall and volume refinement of the mesh dependency study including experimental data with indication of the variation over time.

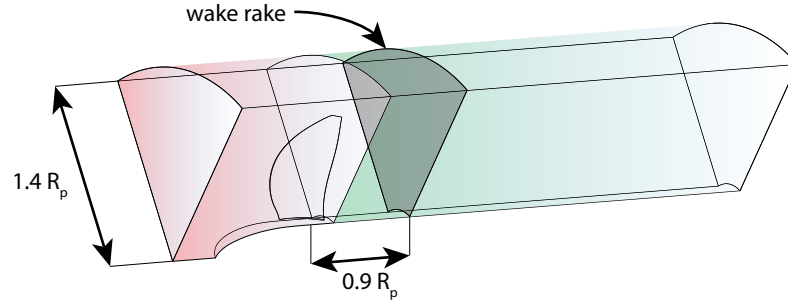


Figure 2.6: Detailed view of the rotating and wake region with the location of the wake rake.

extracted in the propeller CFD model and presented in Figures 2.8 through 2.13 alongside the experimental results. The variation of the circumferential-average axial velocity component with non-dimensionalised radial position r/R_p is shown in Figure 2.8. In general, the CFD results agree well with the experimental results. Contrary to the blade pressure and performance results, wall refinement has very little effect on the axial velocity while volume refinement effects the results more. In the region of the blade tip vortex $0.80 - 0.95R_p$ the largest deviation with respect to the experimental data is seen. Observe Figure 2.9 which shows the circumferential variation of the axial velocity component for an instance in time at two radial positions 0.8 and $0.5R_p$. Especially at $0.8R_p$ a large circumferential variation can be noticed which increases in amplitude with volume refinement: The large axial velocity gradients in the tip region of the wake are better captured and less dissipated due to the smaller mesh cell size. Similar conclusions can be drawn for the tangential velocity component in Figures 2.10 and 2.11. However, also a deviation with respect to the experimental data can be noticed in the region outside of the slipstream from $1.0R_p$ outward: While in the experiment the circumferential-average tangential velocity component decreases monotonically to zero, in the CFD simulations this component drops to zero faster and then oscillates around zero. The circumferential-average radial velocity component is plotted in Figure 2.12. Although the experimental data shows quite some scatter, all CFD results are within the bounds of this scattered field. The magnitude of this quantity is very small which increases the relative influence of measurement errors and thus results in this scattered field. At last, for the radial variation of the circumferential-average total pressure in Figure 2.13 similar conclusions can be drawn as for the axial velocity component.

The quality of resolving the vortex structure of the propeller is also very dependent on the volume refinement, as is shown in Figure 2.14. Iso-surfaces of the vorticity magnitude with the velocity magnitude indicated by colour contours are shown on the left. These 360° results are constructed by circumferential repetition of the results of the propeller CFD model. On the right, contours of the vorticity magnitude at the rake plane are shown. On the left, the root and tip vortex structure is better captured and less dissipated with increased volume refinement. Almost a full propeller rotation is captured by the finest volume mesh. Also the vorticity magnitude in the root and tip vortex structure core increases considerably with increasing volume refinement. In Westmoreland et al. [35] it is shown that the discretization scheme influences the dissipation of vorticity strongly as well: Changing from a 3rd to a 5th-order scheme leads to an increase in capturing the vortex structure from half to two and a half propeller rotations. For the geometry of the swirl recovery vanes and wing in this research, the extend of vortex structure capturing is considered sufficient, considering also the computational cost of increasing the order of the scheme, but it is recommended to look further into the influence of the discretization scheme when aerodynamic surfaces more downstream in the wake are present as well or the magnitude of the vorticity in the tip vortices needs to be enhanced.

In general it can be concluded that wall refinement enhances the propeller performance prediction while volume refinement enhances the capturing of flow quantities in the propeller wake, especially in regions with large gradients. Since the focus of this work is on the interaction of the propeller with swirl recovery vanes and a wing, capturing the wake is more important than estimating the correct propeller performance, as long as the propeller performance is consistent between all simulations. Taking the computational time into account, the coarse wall, fine volume mesh is chosen for the propeller region and the fine volume refinement is also chosen for the wake region in the remainder of the simulations.

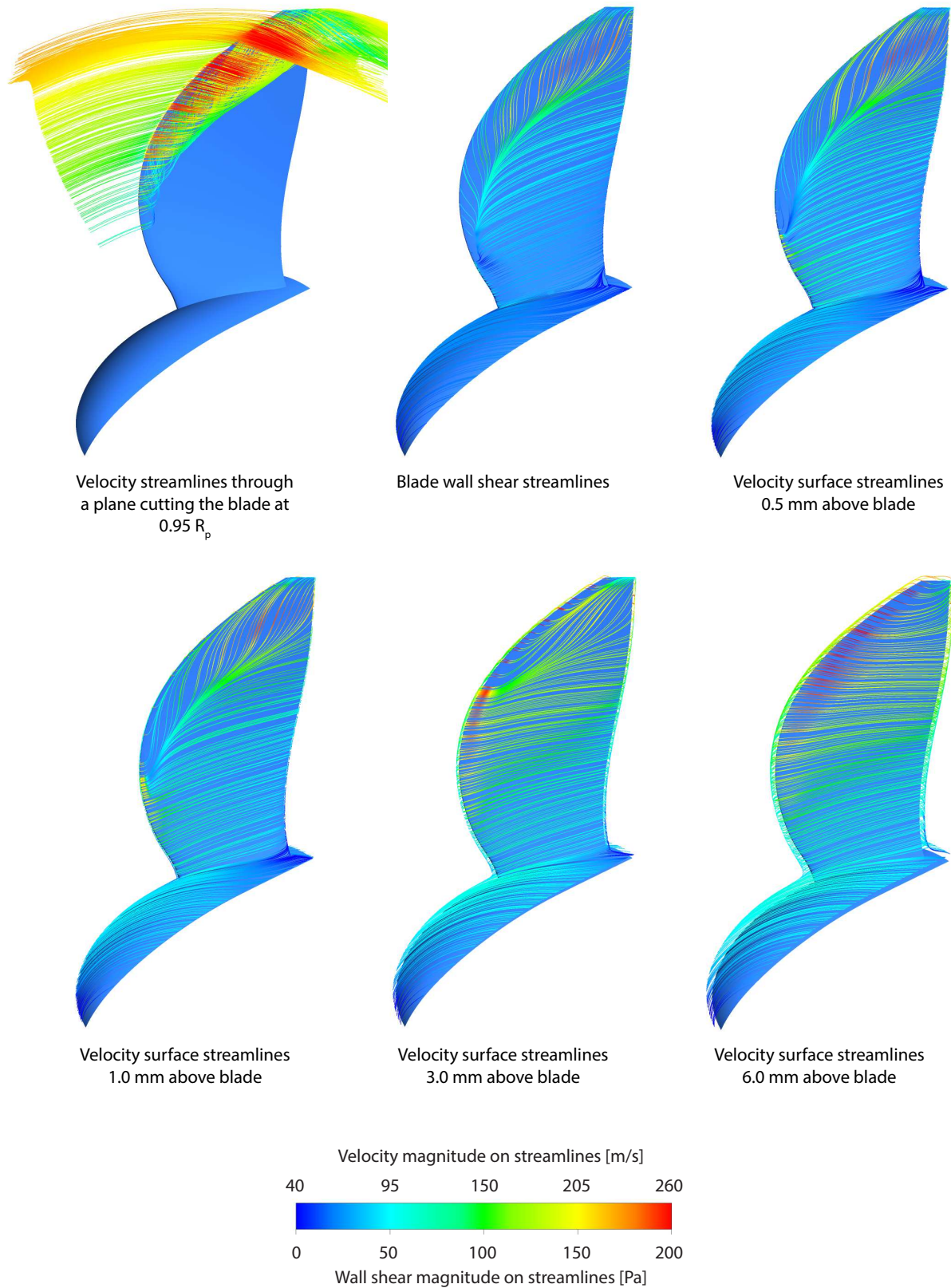


Figure 2.7: Velocity 3D and surface streamlines around the blade and wall shear streamlines on the blade showing the conical separation vortex at $J = 1.00$ simulated with the extra fine wall and coarse volume mesh.

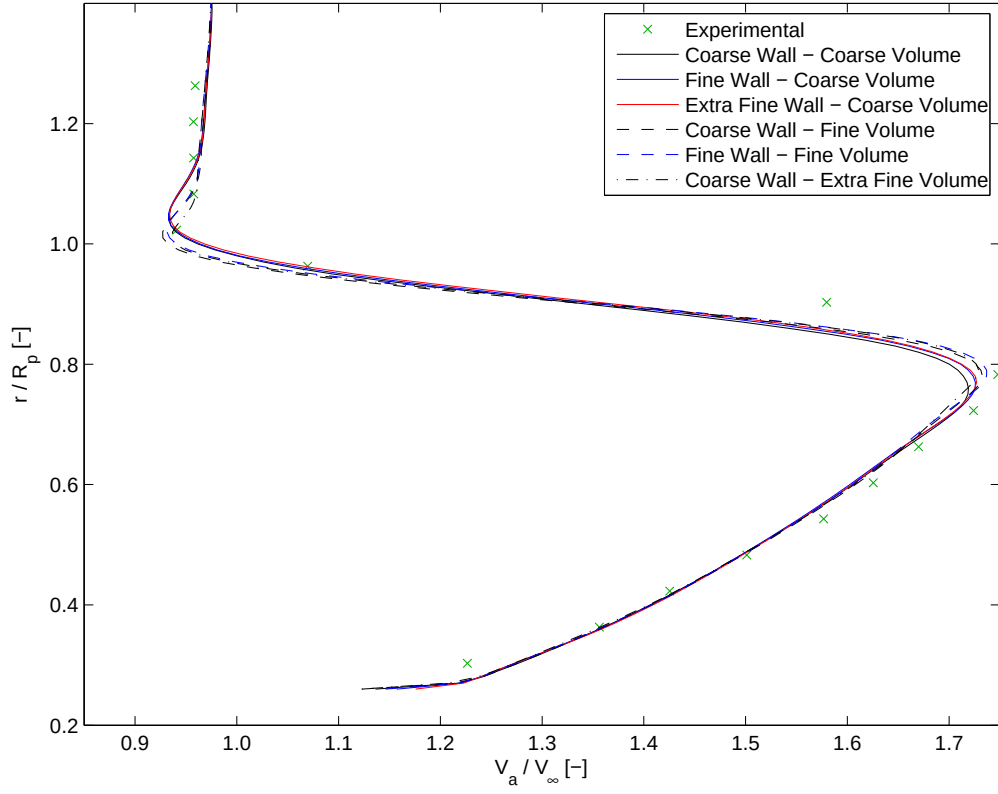


Figure 2.8: Circumferential-average axial velocity component V_a at the rake for the various wall and volume refinement of the mesh dependency study.

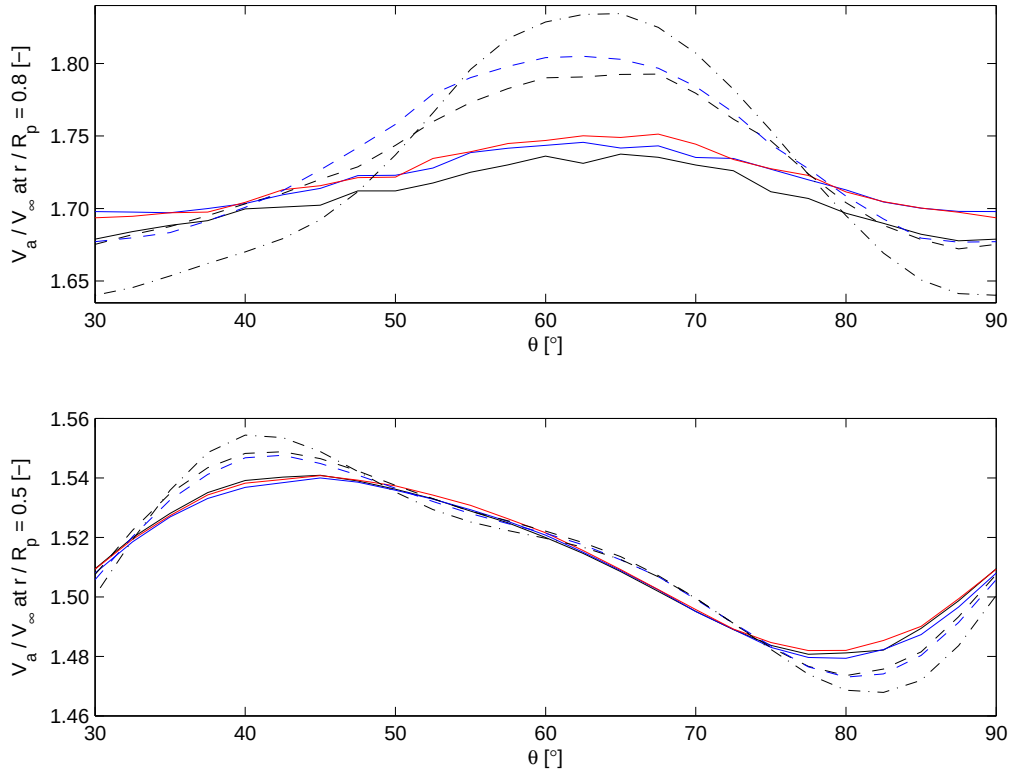


Figure 2.9: Circumferential variation of axial velocity component V_a at the rake at $0.8R_p$ and $0.5R_p$ for the various wall and volume refinement of the mesh dependency study.

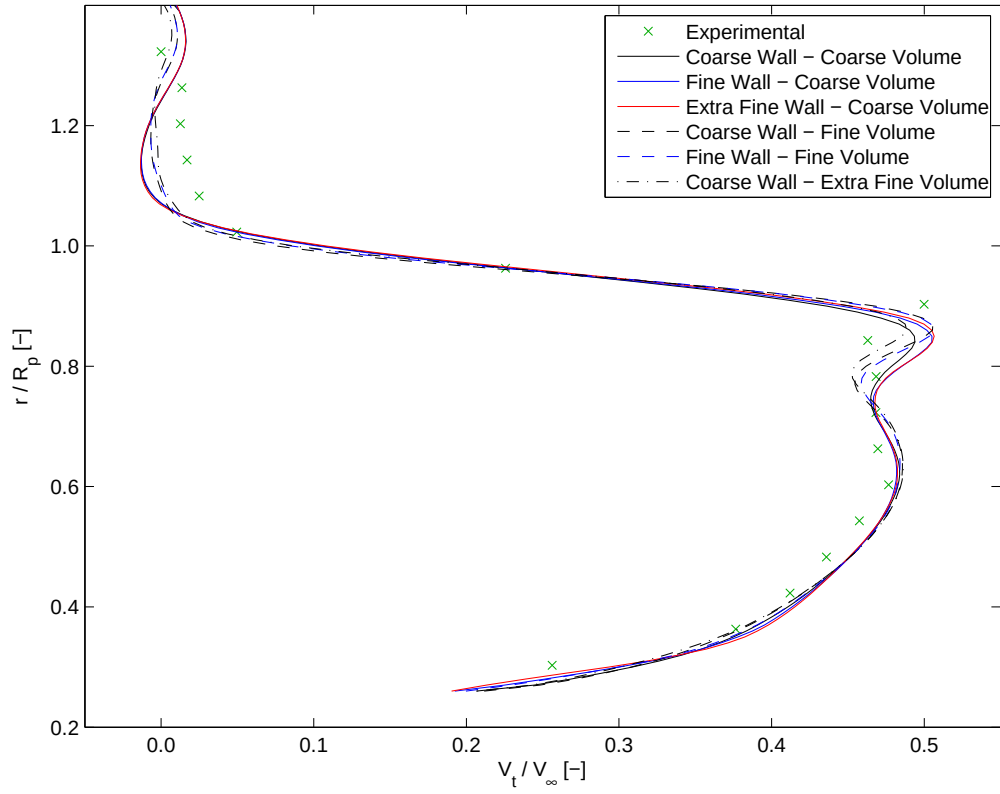


Figure 2.10: Circumferential-average tangential velocity component V_t at the rake for the various wall and volume refinement of the mesh dependency study.

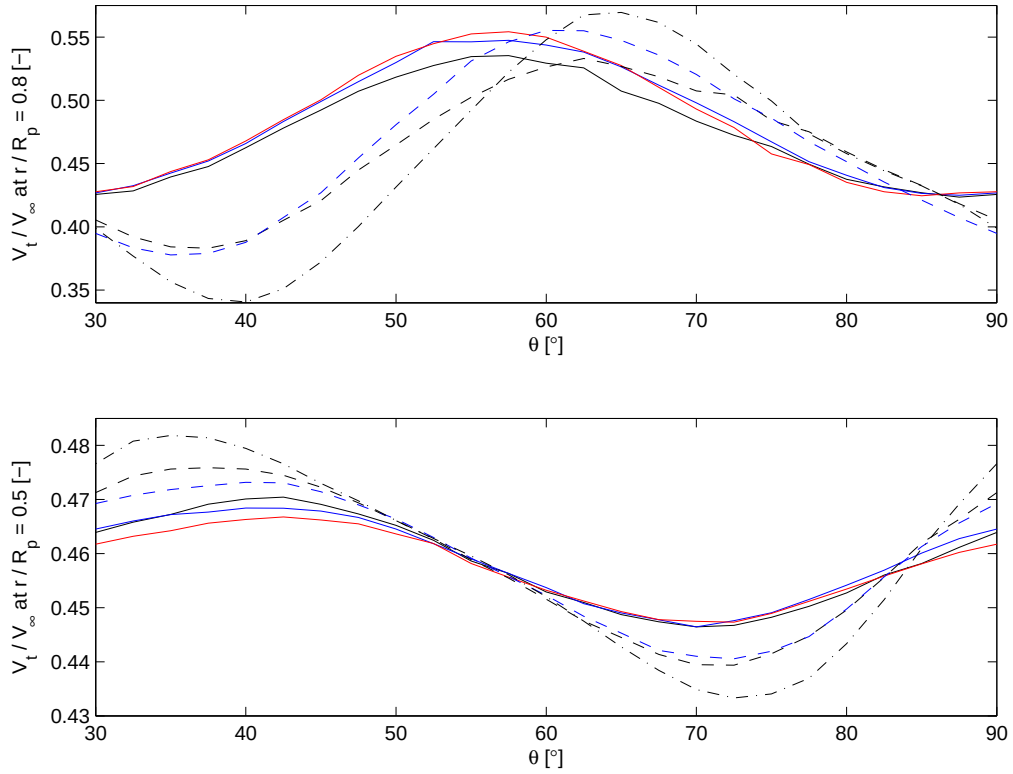


Figure 2.11: Circumferential variation of tangential velocity component V_t at the rake at 0.8 and $0.5R_p$ for the various wall and volume refinement of the mesh dependency study.

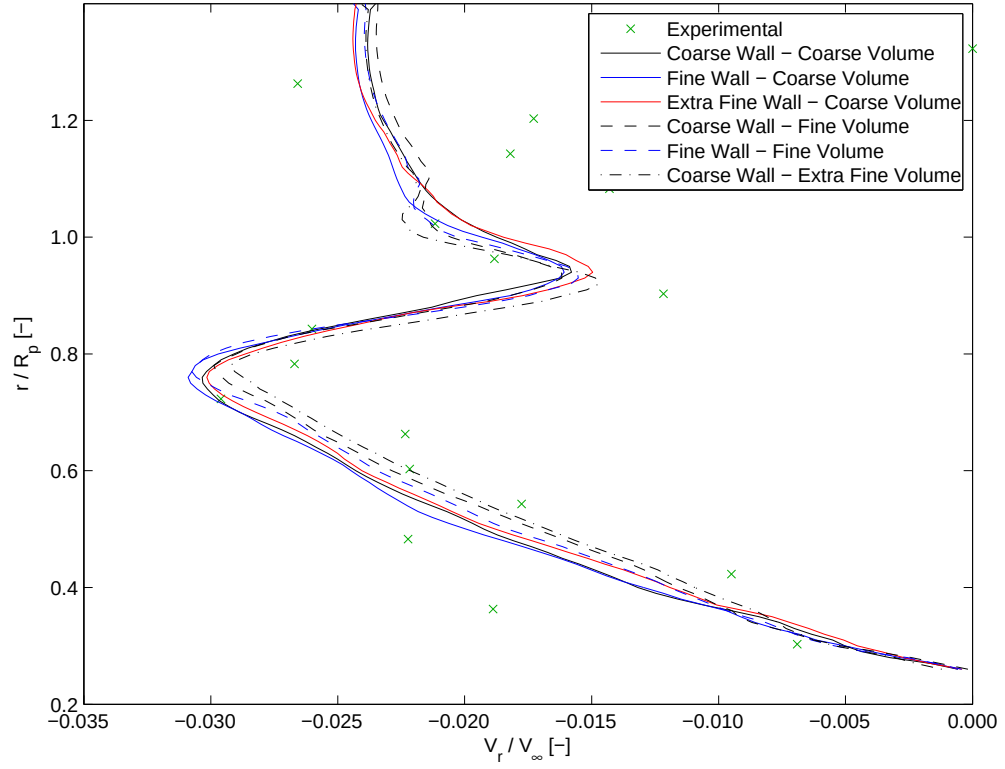


Figure 2.12: Circumferential-average radial velocity component V_r at the rake for the various wall and volume refinement of the mesh dependency study.

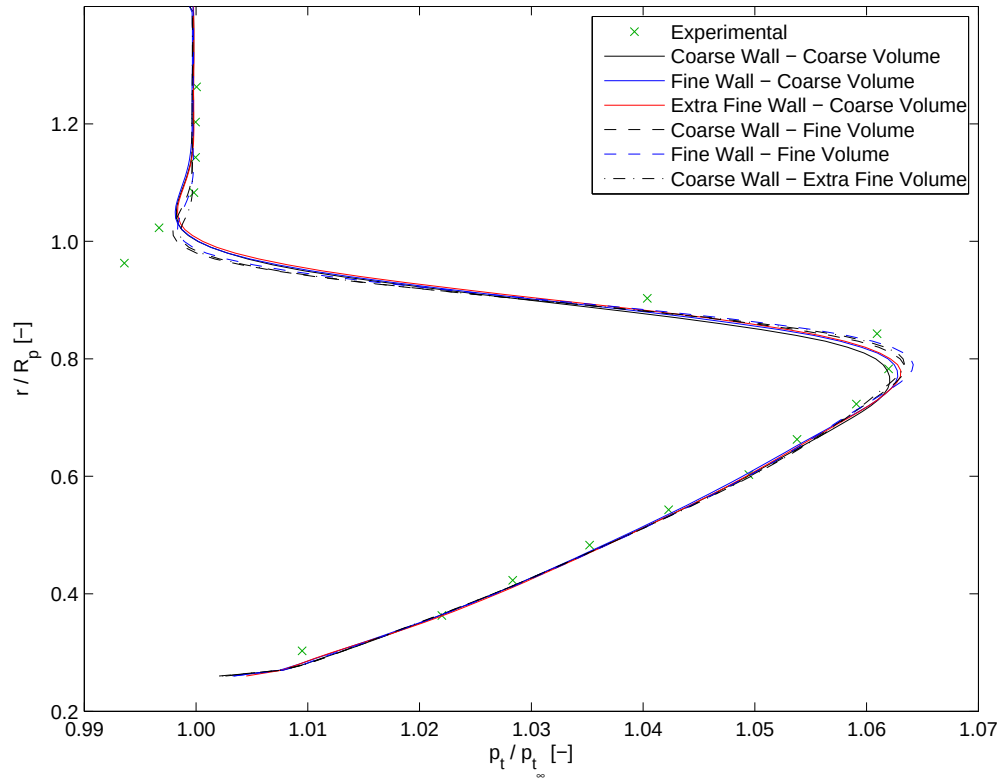


Figure 2.13: Circumferential-average total pressure p_t at the rake for the various wall and volume refinement of the mesh dependency study.

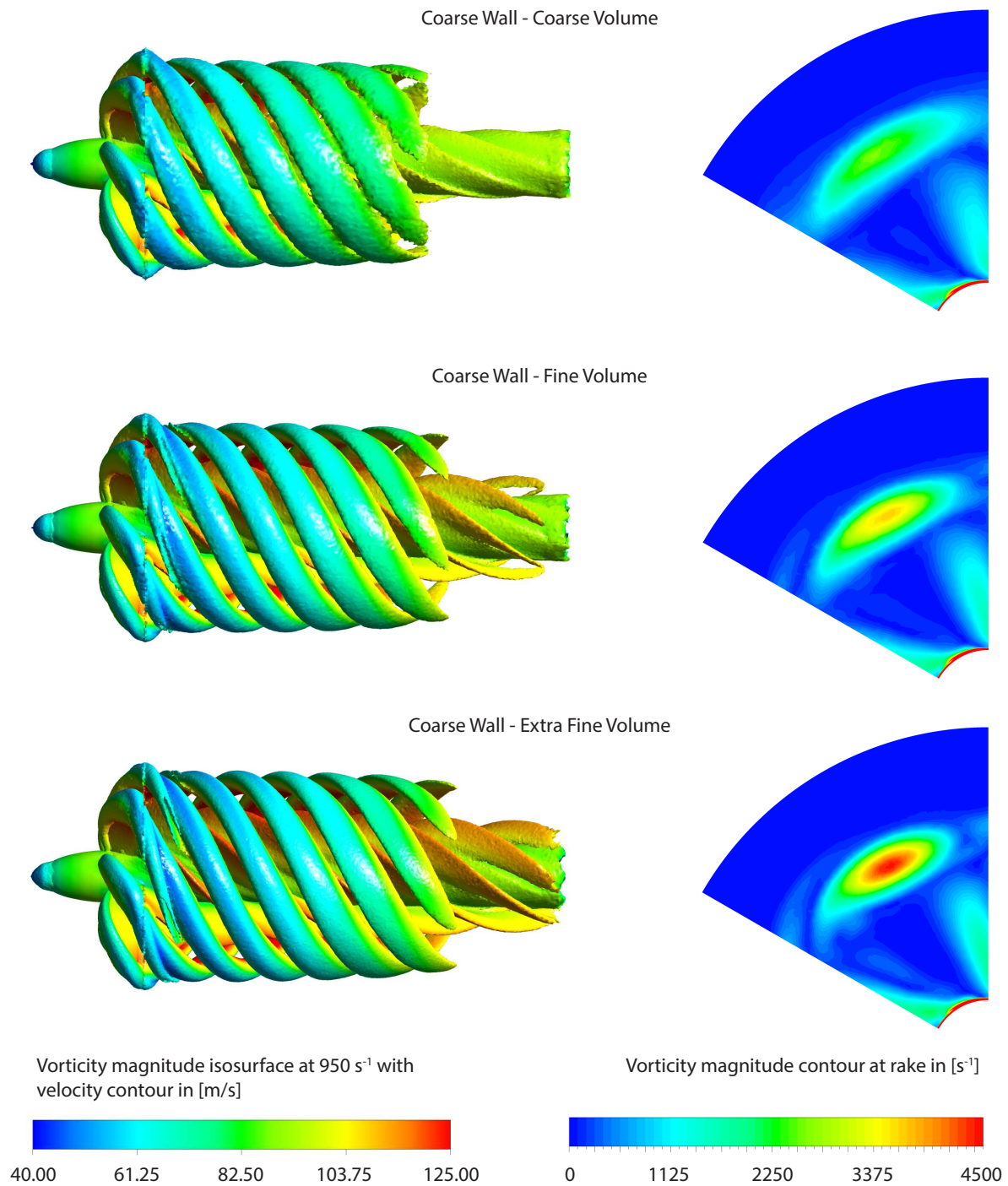


Figure 2.14: Comparison of the vorticity magnitude isosurface in the wake and vorticity magnitude contour at the rake for the three volume refinements of the mesh dependency study.

2.4.2. INFLUENCE OF INLET TURBULENCE AND BLADE TURBULENT MODELLING

The propeller performance may be dependent on the inlet turbulence specification. In order to investigate this dependency, for one simulation the inlet turbulence level and eddy viscosity ratio are reduced compared to the default low turbulence setting as specified in Table 2.6. This results in exactly the same thrust and power coefficient within the shown number of significant digits. Therefore, the default low turbulence intensity inlet condition is used in the remainder of this research. This table also shows the propeller performance when the blade, spinner and hub are modelled as slip walls. One would expect the actual propeller performance, when a laminar boundary layer is present over a part of the blade, to be in between these two extremes of inviscid and fully turbulent modelling of the blade. The slip walls result in a 3 % decrease in thrust coefficient and 5 % decrease in power coefficient. A reduction in power coefficient is expected since the viscous part of the blade drag is absent. One would expect an increase in thrust coefficient because of the lack of the de-cambering effect of a boundary layer like in Stuermer [17]. However, the thrust coefficient decreases in this specific case because of the lack of a 3-D conical separation vortex and its resulting enhanced suction region. It was shown earlier that this phenomenon is also present in the experimental results.

Table 2.6: Propeller performance quantities for two inlet turbulence conditions and inviscid wall blade modelling including deviations from the experimental data at $J = 1.00$ and $M_\infty = 0.200$.

Blade, spinner and hub wall	Turbulence level T_u [%]	Eddy viscosity ratio $\frac{\mu_t}{\mu}$ [-]	C_T [-] (deviation [%])	C_P [-] (deviation [%])	η_p [-] (deviation [%])
no slip	1.00	1.0	0.5317 (-3.59)	1.015 (+2.95)	0.5230 (-6.36)
no slip	0.05	0.1	0.5317 (-3.59)	1.015 (+2.85)	0.5230 (-6.36)
slip	1.00	1.0	0.5141 (-6.78)	0.966 (-1.99)	0.5321 (-4.73)

2.4.3. INFLUENCE OF ADVANCE RATIO

For a constant propeller blade angle, the advance ratio influences the propeller induced velocity field considerably. Observe Figures 2.15, 2.16 and 2.17 which show the circumferential-average axial and tangential velocity component and the circumferential-average swirl angle at the wake rake for various advance ratios for the coarse wall, fine volume mesh. These advance ratios are obtained by varying the propeller rotational speed. Experimental data is available for $J = 0.95, 1.00$ and 1.30 . The CFD results agree well with the experimental data, like was shown in the grid dependency study for $J = 1.00$. These figures make it clear that swirl recovery vanes behind a propeller need to be able to cope with a large range of flow fields for a single undisturbed velocity. The circumferential-average swirl angle increases over 14° when going from $J = 1.80$ to 0.95 . When considering the circumferential variation especially in the tip vortex region, this variation will be even higher. These velocity fields at the rake will be used in Chapter 4 to design swirl recovery vanes.

Table 2.7: Propeller performance quantities for the propeller CFD model.

J [-]	C_T [-]	C_P [-]	η_p [-]
0.95	0.5473	1.042	0.4991
1.00	0.5317	1.015	0.5230
1.30	0.3969	0.774	0.6666
1.60	0.2451	0.510	0.7690
1.80	0.1330	0.312	0.7684

Figure 2.18 shows the C_T , C_P and η_p variation with advance ratio including experimental data. Its numerical values are given in Table 2.7 for reference in the next chapters. As was discussed in the mesh dependency study, the thrust coefficient is slightly under-predicted while the power coefficient is slightly over-predicted over the range of advance ratios. The propulsive efficiency is under-predicted as a result of both deviations. A maximum can be found close to $J = 1.70$. As discussed in Section 1.2, from Gazzaniga and Rose [9] it is known that swirl recovery vanes tend to increase the propulsive efficiency for advance ratios lower than the maximum and decrease for advance ratios higher than the maximum. $J = 1.60$, which is slightly lower than the advance ratio corresponding to the maximum in propulsive efficiency, is therefore considered a representative cruise condition for the design of swirl recovery vanes in Chapter 4. $J = 0.95$ is considered a representative high-thrust condition encountered at lower speeds for instance in the climb phase.

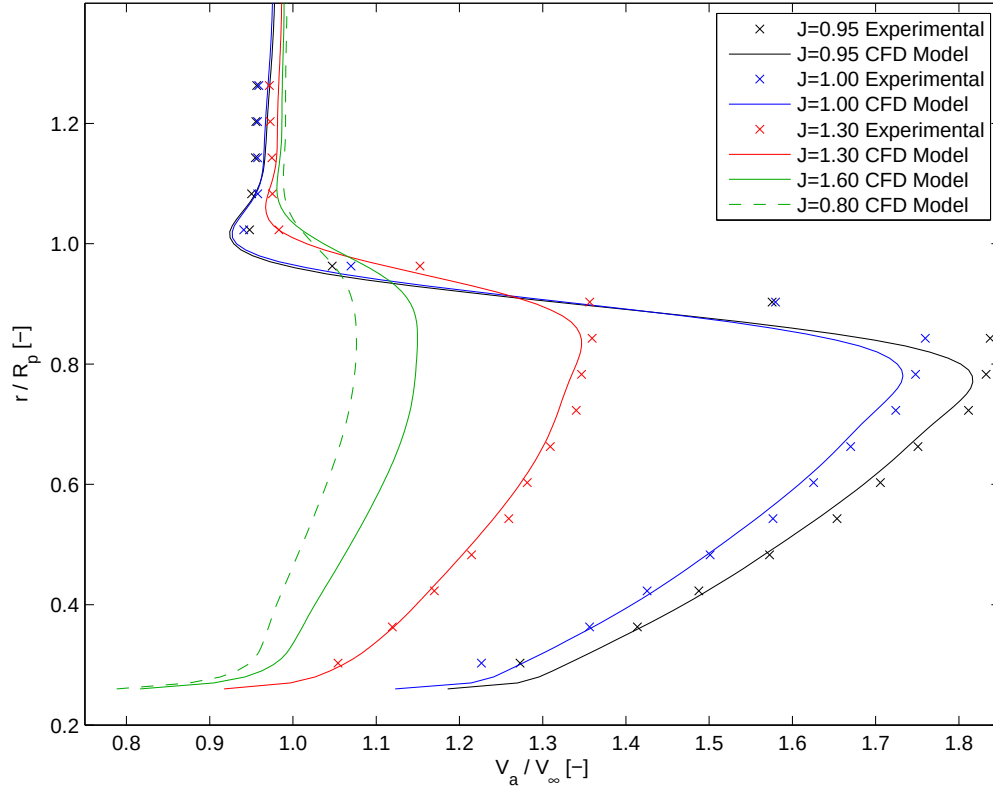


Figure 2.15: Circumferential-average axial velocity at the rake for a range of advance ratios.

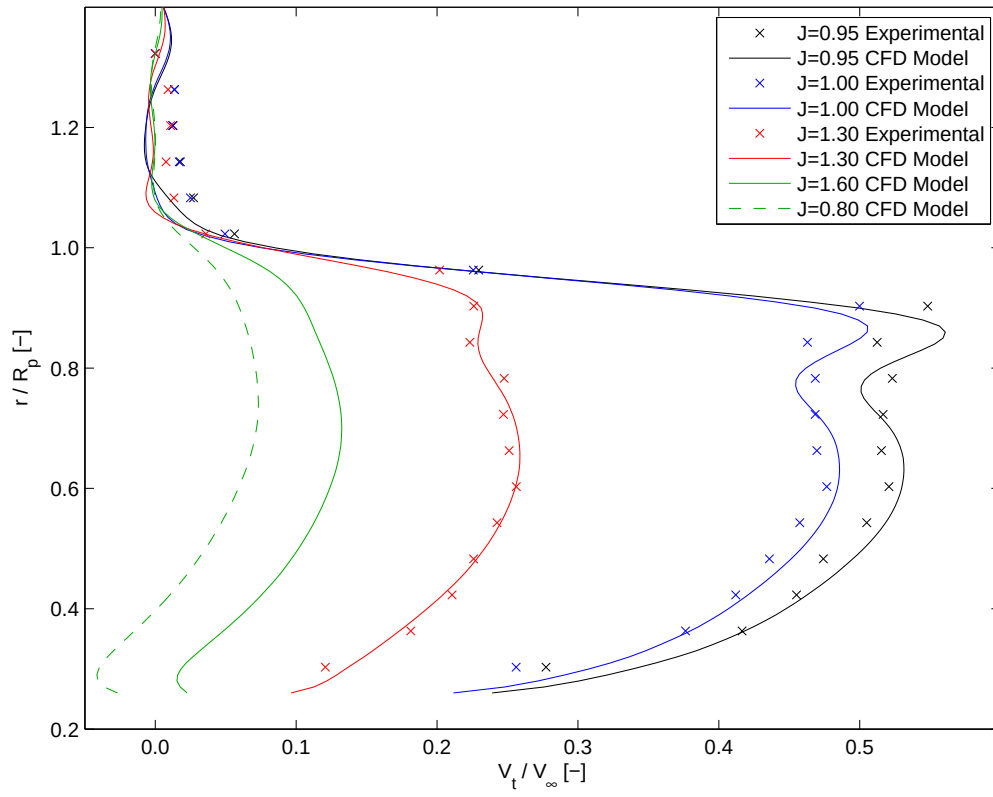


Figure 2.16: Circumferential-average tangential velocity at the rake for a range of advance ratios.

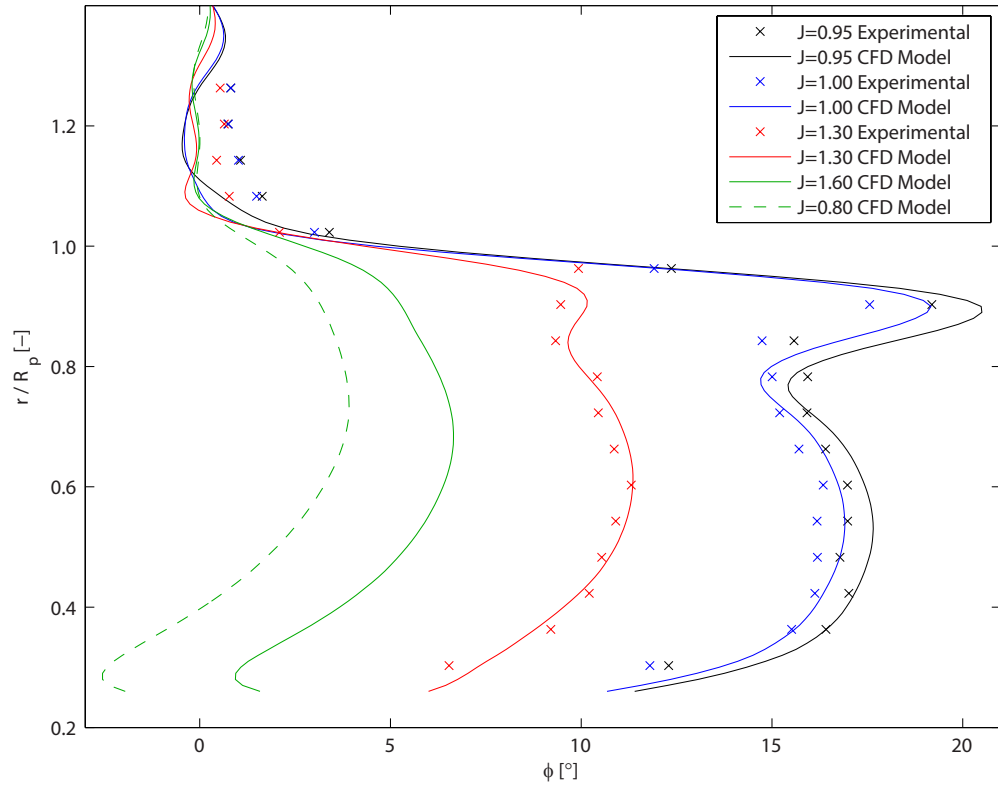


Figure 2.17: Circumferential-average swirl angle ϕ at the rake for a range of advance ratios.

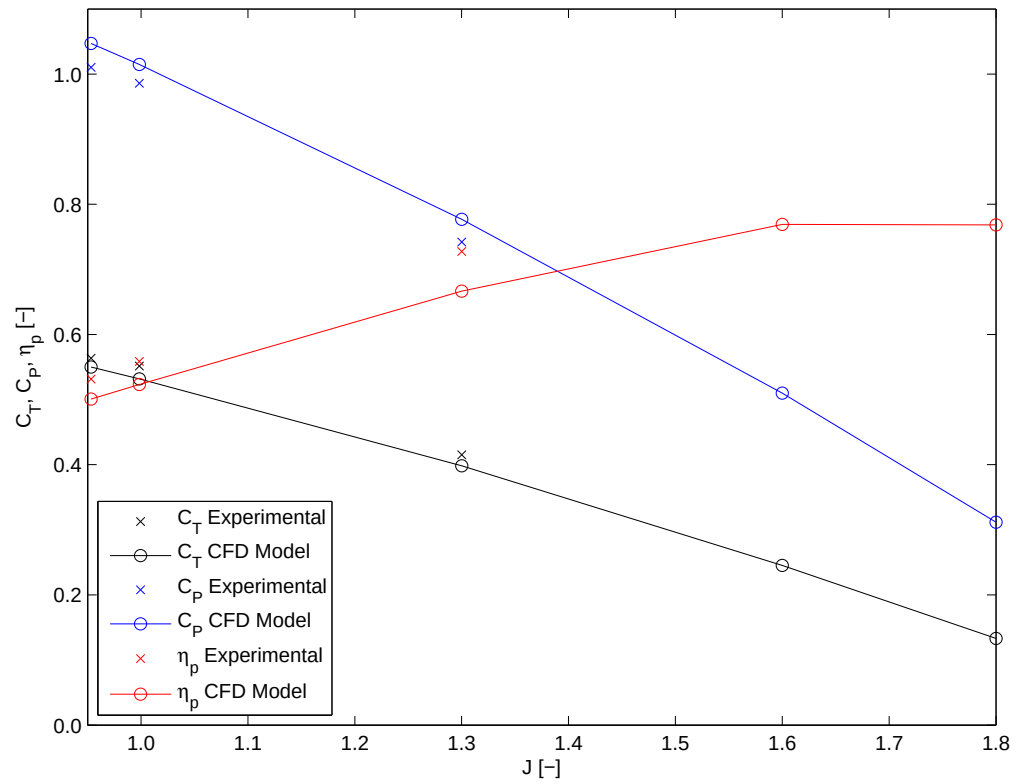


Figure 2.18: Power coefficient, thrust coefficient and propulsive efficiency versus advance ratio.

3

APIAN-INF CFD MODEL

For the APIAN propeller introduced in the previous chapter swirl recovery vanes have been designed by van Kuijk [21] and these have been tested in the DNW Large Low-speed Facility (LLF) as part of the APIAN-INF (APIAN In Non-uniform Flow) test program as described by Sinnige et al. [7]. In this chapter a CFD model of this configuration is discussed and compared to the design performance and wind tunnel PIV measurements. The initial plan was to use this SRV design for all subsequent simulations. In this chapter it will be shown that the propulsive efficiency benefit is much lower than predicted and therefore this design is not used further. However, this chapter does provide a valuable comparison of CFD and experiment with SRV. Also, all necessary solver and mesh settings that are needed to model a propeller with SRV are established for this APIAN-INF CFD model and used in subsequent simulations. This chapter is structured as follows: First, the geometry of the model is presented in Section 3.1. The characteristics of the corresponding mesh are given in Section 3.2. The solver settings are explained in Section 3.3. Wherever possible, the geometry, mesh and solver settings of the APIAN-INF CFD Model are the same as the propeller CFD model of the previous chapter. At last, the results are discussed in Section 3.4.

3.1. GEOMETRY

This section describes the APIAN-INF SRV geometry and the corresponding CFD model domain geometry.

3.1.1. SRV GEOMETRY

Figure 3.1 shows the SRV design including the mounting plates. The bolts and holes are removed compared to the wind tunnel test for simplification of the mesh. The SRV planform, pitch distribution and airfoil are shown in Figure 3.2. The details of the SRV design can be found in van Kuijk [21]. A short overview is provided here.

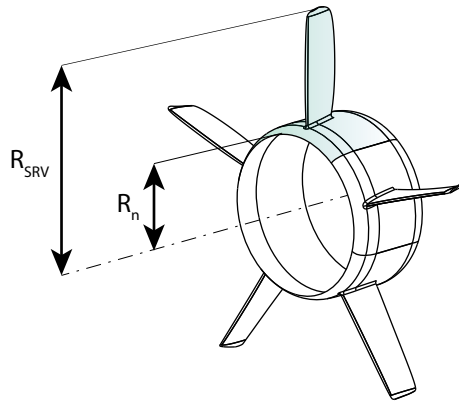


Figure 3.1: APIAN-INF swirl recovery vanes designed by van Kuijk [21].

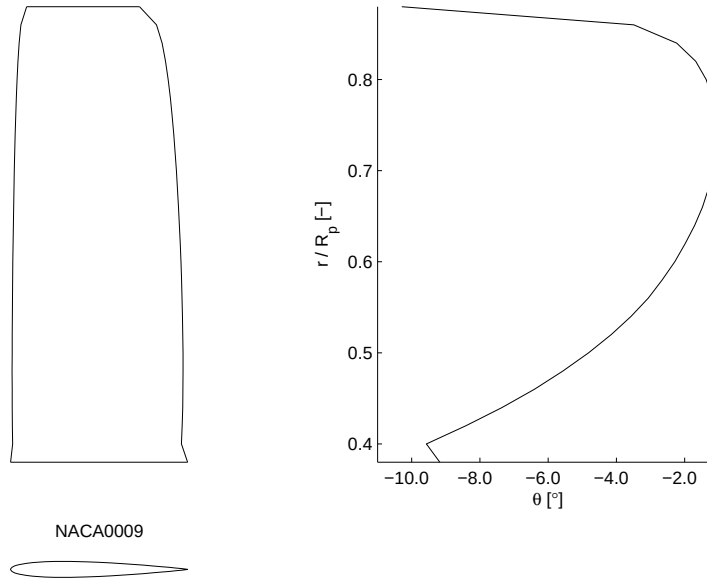


Figure 3.2: APIAN-INF SRV chord and pitch distribution and airfoil section. The pitch angle θ is defined positive as shown in Figure 4.2.

The minimum induced loss design routine in the propeller design and analysis tool XROTOR of Drela and Youngren [36] was used to design the SRV. This routine varies the spanwise chord and pitch angle to reach a constant overall lift coefficient over the blade as the optimal blade loading distribution. ‘While the code was still under development, various stator designs were already made for the APIAN rotor for the APIAN-INF test program [...]. The chosen design is by no means the very optimum design attainable, but [...] it will deliver an efficiency increase of about 2.5% in the lower end of the advance ratio range.’ [21] The design consists of five blades and not the same number of blades as the propeller to avoid noise by aerodynamic interference. Also, more blades than five would have increased the production time and cost. A symmetrical airfoil is chosen because of better performance over cambered airfoils. A NACA 0009 airfoil is used over the entire SRV span, the thickness being a compromise between performance and structural strength. The SRV tip radius is cropped with respect to the propeller to $R_{SRV} = 221.95 \text{ mm}$. The design advance ratio is $J = 1.75$ to have good performance in cruise condition. The wind tunnel test is performed for $J = 1.05$, 1.40 and 1.75 . According to the design tool, the propulsive efficiency increase $\Delta\eta_p$ by the addition of the SRV is 2.5 % for $J = 1.05$ and 1.8 % for $J = 1.40$. It is negative for the design advance ratio, but the exact value is unknown.

3.1.2. DOMAIN GEOMETRY

The structure of the domain is similar to that of the propeller CFD model displayed in Figure 2.2. It also consists of three regions: An outer region, rotating region and wake region. The latter two regions are shown in Figure 3.3. The rotating region is exactly the same as that of the propeller CFD model, which is a 60° wedge containing a single blade of the APIAN propeller including the spinner and hub. The outer and wake regions are slightly altered: Both regions are 72° wedges instead of 60° and contain a nacelle of larger radius R_n than that of the propeller CFD model in order to match the wind tunnel test conditions. The wake region also contains one of the vanes including mounting plate as shown in green in Figure 3.1.

3.2. MESH

The properties of the mesh are the same as for the propeller CFD model. Apart from the SRV wall refinement, wall and volume refinements are as chosen in the mesh dependency study of the previous chapter: the coarse wall, fine volume refinement. On the SRV wall two different wall refinements are specified in order to investigate mesh dependency of the results. The mesh size and refinement dimensions of this coarse and fine SRV wall refinement are given in Table 3.1. The results of these two refinements are discussed in Subsection 3.4.1.

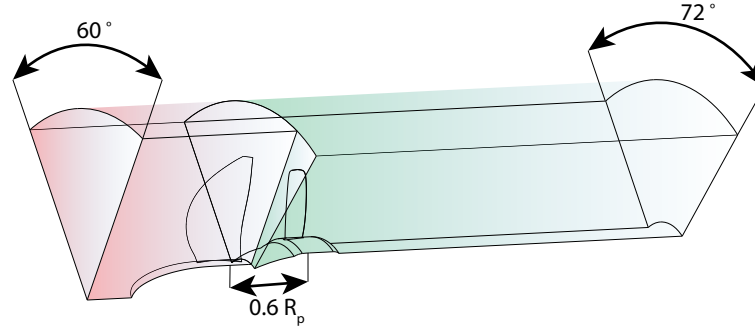


Figure 3.3: Detailed view of the rotating and wake region of the APIAN-INF CFD Model.

Table 3.1: Overview of mesh size and refinement dimensions for the mesh dependency study of the APIAN-INF CFD model.

SRV wall refinement	# nodes stationary region	# nodes rotating region	SRV wall size [mm]	l.e. & t.e. size [mm]
Coarse	965,663	762,551	1.50	0.40
Fine	1,297,959	762,551	1.00	0.25

3.3. SOLVER

Observe Table 3.2 for the flow conditions of the APIAN-INF CFD model. A single advance ratio is chosen for the CFD simulations of $J = 1.40$ in between the cruise and high-thrust condition to reduce the computational cost. The discretization schemes and turbulence model are equal to that of the propeller CFD model. The inlet, outlet, slip and no-slip boundary conditions are also the same. Because of the unequal pitch of the rotating region on the one hand and the outer and wake regions on the other hand, not the same interface conditions can be specified as in the propeller CFD model. Such a pitch change is very common between the rotor and stator stages in turbo-machinery CFD simulations as well. In this research field there are four solutions for this problem according to Blumenthal et al. [37]:

1. Increase the number of rotor blades and stator vanes such that the regions have equal pitch. The extreme case is a full 360° domain simulation. For the current number of propeller blades of 6 and swirl recovery vanes of 5, a full 360° domain is the only possibility.
2. Profile transformation: The flow is stretched or compressed in circumferential direction across the interface and the periodic boundary conditions remain unchanged. This leads to an error in the flow field that increases with increasing pitch difference.
3. Time transformation: The time coordinates of both regions are transformed in circumferential direction to make it fully periodic in transformed time. This means that the rotating region and outer and wake region march at different time steps, and are transformed back at the end of the run to the same physical time. This method becomes unstable when the pitch change is too large.
4. Fourier transformation: The flow history on the periodic boundary conditions is stored using Fourier series decomposition and then a phase shift is applied. In ANSYS CFX, this method uses a double passage, two rotor blades and two stator vanes per region, in order to accelerate convergence [27]. The Fourier coefficients of the flow variables are collected at the interface between the two passages and then applied to the periodic boundaries on both sides with the appropriate phase shift. This method can be applied to large pitch changes.

The time and Fourier transformation method are still beta features of ANSYS CFX. Both methods have been applied to the APIAN-INF CFD model but diverge after a while. The Fourier method, which is most promising because of the possibility of large pitch change, uses in the same order of memory as the 360° domain simulation, partly because of the double passage in the domain and because of the storage of the Fourier coefficients of the flow variables. High memory usage is undesirable because it requires a larger part of a node and thus more processing capacity when run on a cluster. The only two remaining methods are the

360° domain simulation and the profile transformation. These two methods will be considered in this chapter. The advantage of the simulation by means of profile transformation over the 360° domain simulation is the much smaller mesh size and thus much smaller computational cost. The disadvantage is the inherently wrong propeller wake in the outer and wake region.

Table 3.2: Overview of undisturbed flow quantities and inlet total pressure jump for the APIAN-INF CFD model and experiment without and with SRV.

Case	J [-]	V_∞ [m/s]	p_∞ [Pa]	ρ_∞ [kg/m ³]	T_{t_∞} [K]	$\Delta p_{t_{inlet}}$ [Pa]
CFD with and without SRV	1.40	60.00	101010.0	1.1760	300.00	2116.9
Experimental without SRV	1.42	60.29	101956.3	1.1877	299.36	2159.2
Experimental with SRV	1.42	60.20	100895.9	1.1670	301.33	2115.3

3.4. RESULTS

In this section the results of the APIAN-INF CFD model are discussed. Since the nacelle radius is larger than the one of the simulations in the previous chapter, those results cannot be used as a reference and a simulation without SRV and mounting plate is performed as well. The resulting performance quantities are given in Table 3.3 including deviations from the experimental values given in Table 3.4. Table 3.5 gives an overview

Table 3.3: Time-average performance of the APIAN-INF CFD model without SRV and mounting plate including deviation from experimental results of Table 3.4.

C_T [-] (deviation [%])	C_P [-] (deviation [%])	η_p [%] (deviation [%])
0.3679 (+5.23)	0.7228 (+1.73)	71.26 (+3.44)

Table 3.4: Time-average performance of the APIAN-INF experiment without SRV and relative to this the time-average performance with SRV as defined in Equations (3.1) to (3.4).

C_T [-]	ΔC_{T_p} [%]	C_P [-]	ΔC_P [%]	η_p [%]	$\Delta \eta_{p_p}$ [%]
0.3496	-2.77	0.7105	-5.00	68.89	-0.77

of the time-average performance results of the various simulated cases with SRV and mounting plate. Three cases are simulated, two simulations using a profile transformation with different SRV wall refinement, and one simulation using a 360° domain. These results and the experimental results with SRV in Table 3.4 are given relative to the results when no SRV are present by means of five relative performance quantities. These are defined as follows. ΔC_{T_p} is the increase in propeller thrust coefficient by the addition of SRV:

$$\Delta C_{T_p} = \frac{(C_{T_p})_{\text{with}} - (C_T)_{\text{without}}}{(C_T)_{\text{without}}} \quad (3.1)$$

ΔC_T is the increase in overall (propeller and SRV thrust combined) thrust coefficient by the addition of SRV:

$$\Delta C_T = \frac{(C_T)_{\text{with}} - (C_T)_{\text{without}}}{(C_T)_{\text{without}}} \quad (3.2)$$

ΔC_P is the increase in propeller power coefficient by the addition of SRV:

$$\Delta C_P = \frac{(C_P)_{\text{with}} - (C_P)_{\text{without}}}{(C_P)_{\text{without}}} \quad (3.3)$$

$\Delta \eta_{p_p}$ is the increase in propeller propulsive efficiency by the addition of SRV:

$$\Delta \eta_{p_p} = J \left(\left(\frac{C_{T_p}}{C_P} \right)_{\text{with}} - \left(\frac{C_T}{C_P} \right)_{\text{without}} \right) \quad (3.4)$$

Table 3.5: Overview of time-average performance results for the various cases of the APIAN-INF CFD model with SRV relative to the performance without SRV from Table 3.3 as defined in Equations (3.1) to (3.5).

SRV wall refinement	Interface type	ΔC_{T_p} [%]	ΔC_T [%]	ΔC_P [%]	$\Delta \eta_{p_p}$ [%]	$\Delta \eta_p$ [%]
Coarse	profile transformation	+0.68	+1.41	+0.50	+0.13	+0.65
Fine	profile transformation	+0.57	+1.49	+0.40	+0.12	+0.78
Coarse	360 ° domain	+0.35	+1.06	+0.26	+0.06	+0.57

The increase in overall propulsive efficiency by the addition of SRV $\Delta \eta_p$ as already defined in Equation (1.6) is:

$$\Delta \eta_p = J \left(\left(\frac{C_T}{C_P} \right)_{\text{with}} - \left(\frac{C_T}{C_P} \right)_{\text{without}} \right) \quad (3.5)$$

In these equations the subscripts *with* and *without* denote the results when SRV are present and when the SRV are not present respectively and when SRV are present C_T is defined as:

$$(C_T)_{\text{with}} = \frac{T_p + T_{SRV}}{\rho_\infty n^2 D_p^4} \quad (3.6)$$

where T_{SRV} is the SRV thrust force. When SRV are present, the propeller thrust coefficient C_{T_p} is explicitly defined as:

$$(C_{T_p})_{\text{with}} = \frac{T_p}{\rho_\infty n^2 D_p^4} \quad (3.7)$$

The performance results in these tables are discussed in Subsection 3.4.1 concerning the mesh dependency of the results, in Subsection 3.4.2 concerning the effect of the profile transformation and in Subsection 3.4.3 showing the effect of the SRV on the performance as predicted by the 360 ° domain simulation and providing a comparison with the experimental results. The chapter is concluded in Subsection 3.4.4 with a description of the slipstream and comparison with experimental results.

3.4.1. SRV WALL MESH DEPENDENCY STUDY

A coarse and fine SRV wall refinement have been simulated by means of a profile transformation in order to investigate the effect of refinement. The additional refinement of the SRV surface leads to a small increase in $\Delta \eta_p$. This increase can most likely be explained by an increase in the suction peak due to a finer mesh near the leading edge. A larger suction peak in this forward oriented region increases the SRV thrust. This lower pressure reduces the upstream propeller power and thrust slightly as well. It is thought that the difference is insignificant compared to the difference with respect to the design of $\Delta \eta_p = 1.8$ % and therefore the coarse SRV wall refinement is chosen for the 360 ° domain simulation. Also, the fine SRV wall refinement would increase the computational cost of the 360 ° domain simulation even more. A difference of this order should be taken into account as error margin for that simulation.

3.4.2. EFFECT OF PROFILE TRANSFORMATION

When a profile transformation is applied between the rotating region and outer and wake region, the flow is circumferentially compressed towards the rotating region and stretched from the rotating region outward. The largest influence of this method can be expected at the interface between the rotating and wake region, through which the wake of the propeller blade passes. Observe Figure 3.4 (a) and (b) which show contour plots of the axial and tangential velocity component respectively on both sides of this interface. For clarity, the full 360 ° equivalent is shown by repeating the wedges of 60 ° and 72 ° on the left and right respectively. From these figures it is clear that due to the profile transformation, the periodicity of the flow variables changes from six to five cyclic repetitions. The axial and tangential velocity magnitude range do not change but the flow features are circumferentially stretched and the mass flow increases from the rotating to the outer and wake region. The propeller wake structure is qualitatively wrong, it only consists of five propeller blade tip and root vortical structures instead of six, one for each blade. During one propeller rotation period the single vane that is present in the wake region and repeated by means of the periodic boundary conditions only sees these five periodic repetitions instead of six. This does not necessarily change the time-average vane loading but may change the instantaneous propulsive efficiency increase by the SRV due to the change in period of the periodic vane loading with respect to the possibly periodic blade loading.

From the results it appears that this error introduced by the profile transformation is small: Observe Table 3.5. When comparing the coarse SRV wall refinement simulation by means of profile transformation to the 360° domain simulation a small increase in $\Delta\eta_p$ can be noticed by the profile transformation. This difference is almost only the result of a difference in propeller propulsive efficiency $\Delta\eta_{pp}$ due to changes in propeller thrust coefficient C_{Tp} and power coefficient C_P : The profile transformation results in a 0.33 % increase in the propeller thrust coefficient and 0.24 % increase in the power coefficient. In other words only the propeller loading is affected and the SRV thrust is the same. These changes should be accounted for, or a 360° domain simulation should be used for estimation of this error.

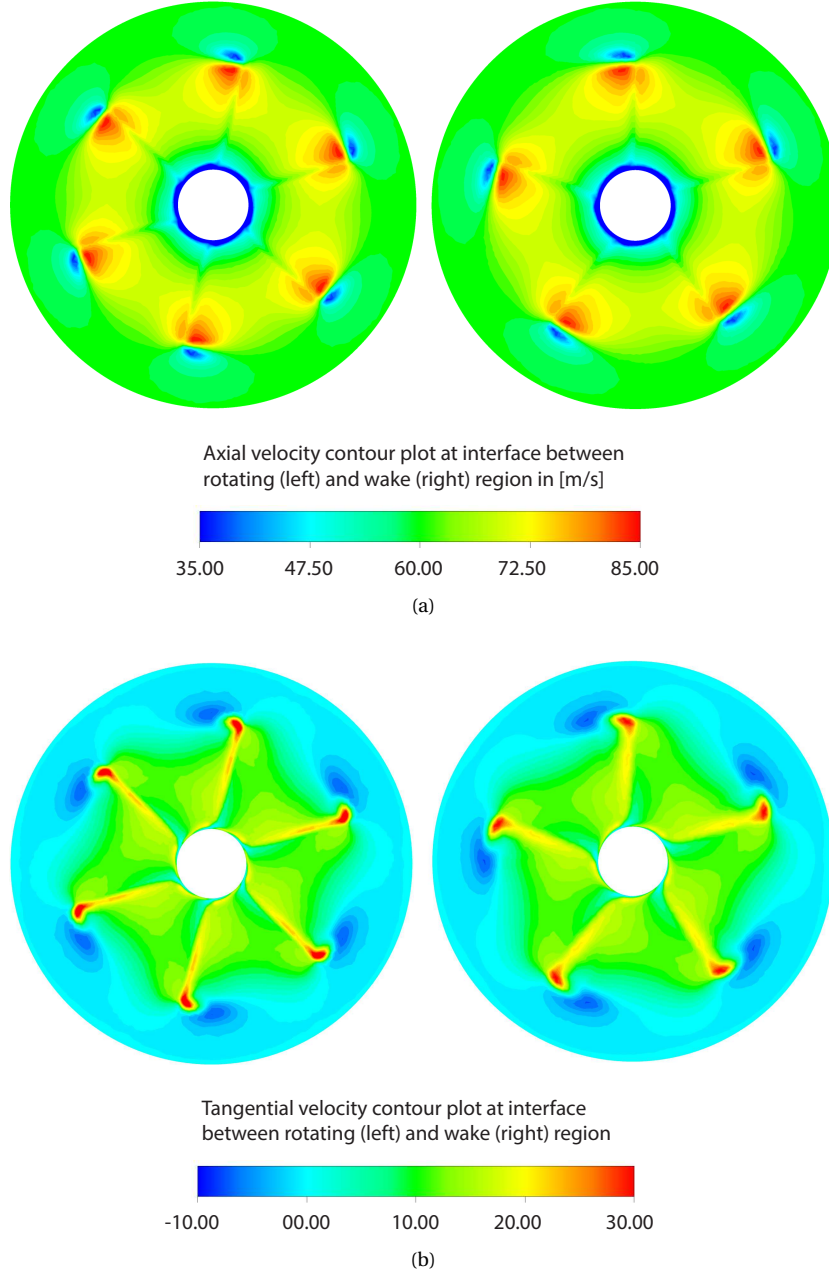


Figure 3.4: Profile transformation of the (a) axial and (b) tangential velocity. For clarity, the full 360° equivalent is shown by repeating the wedges of 60° and 72° on the left and right respectively.

One can argue that the same result could have been obtained by circumferentially averaging the flow quantities over the interface by a so called mixing plane as used in Montomoli et al. [38]. The difference is that by means of a profile transformation, the cyclic variation of the SRV loading is still captured and very similar to that of the full 360° domain simulation. This latter aspect is important for the design of the SRV,

which should account for this variation and not for instance stall for a part of the cyclic variation. Because of the relatively small computational cost the profile transformation is an excellent method for testing new SRV designs and will be used as such in Chapter 5 of this report.

3.4.3. 360° DOMAIN PERFORMANCE RESULTS AND COMPARISON WITH THE EXPERIMENT

In this subsection the performance results of the 360° domain simulation are discussed and compared to experimental measurements. First observe Table 3.2 for the flow conditions. The CFD flow conditions are very similar but do not match the experimental flow conditions exactly. The undisturbed velocity and advance ratio are slightly higher in the experiment. In Chapter 2 it was shown that the propeller CFD model underpredicts the thrust coefficient by 3.6 % and overpredicts the power coefficient by 2.9 %. Table 3.3 indicates that APIAN-INF CFD model without SRV overpredicts the thrust coefficient by 5.23 % and overpredicts the power coefficient by 1.73 %. The overprediction of the thrust coefficient can partly be explained by the slightly different flow conditions, in particular the lower advance ratio. However, one would expect a larger overprediction of the power coefficient.

In Table 3.4 it is shown that the difference in propeller thrust and power coefficient in the experiment by the addition of SRV is quite large, $\Delta C_{t_p} = -2.77\%$ and $\Delta C_p = -5.00\%$, while the prediction of these differences in the CFD models in Table 3.5 are only $\Delta C_{t_p} = +0.35\%$ and $\Delta C_p = +0.26\%$. The difference in density and undisturbed dynamic pressure (total pressure jump $\Delta p_{t_{inlet}}$) between the two experimental cases may explain part of the difference. Sinnige et al. [7] concludes that the effect of the SRV on the propeller loading in the experiment is within the statistical uncertainty when a larger data set is observed and thus no such effect is observed. In general it can be concluded that the experimental results can be used for qualitative comparison with the CFD model results, but for quantitative comparison the differences are too large, in the order of the benefit of SRV. Since the flow conditions of the two CFD models without and with SRV match, quantitative comparison between these two cases is possible, but not validated.

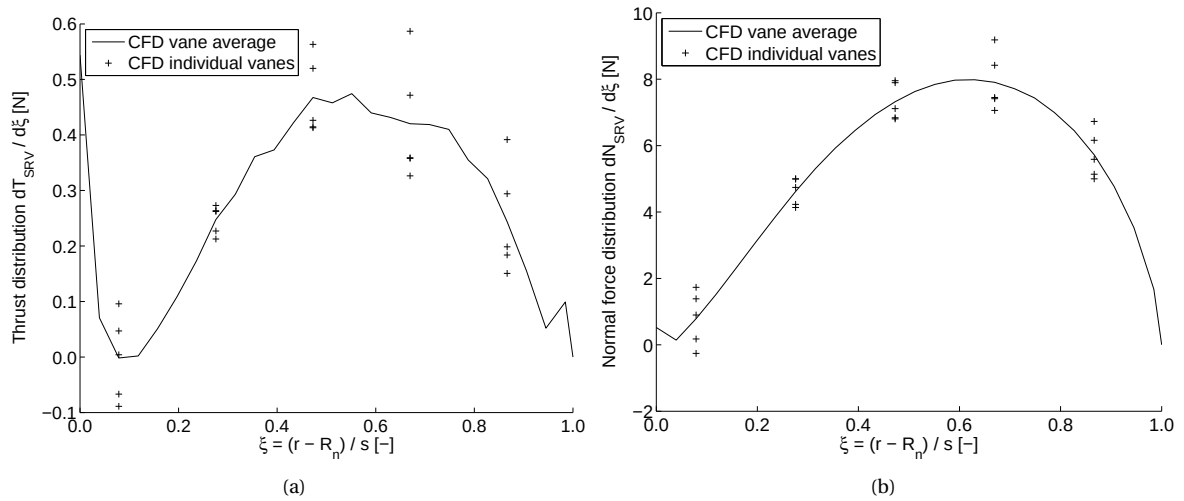


Figure 3.5: Average and individual vane (a) thrust and (b) normal force distribution at an instance of the propeller blade passage defined in Figure 3.6.

When comparing the performance of the CFD model without and with SRV, it can be concluded that the design SRV benefit of $\Delta \eta_p = 1.8\%$ as given by van Kuijk [21] is not reached and only a benefit of $\Delta \eta_p = 0.57\%$ is predicted. The main design flaw is the large negative pitch angle near the root as shown in Figure 3.2. In Figure 3.5 the average and individual vane thrust and normal force distribution is plotted: While the vanes are loaded outboard with a net thrust force, in the region close to the root the loading is low and the thrust distribution is on average zero. At the root the thrust distribution shows a large positive peak due to a thick trailing edge by the applied fillet at the vane-nacelle junction. Because of the limited propulsion efficiency benefit of this SRV, new SRV designs will be proposed in the next chapter and used in subsequent simulations. The remainder of this chapter does provide a valuable description of the slipstream predicted by the CFD simulation including a comparison with experimental PIV results.

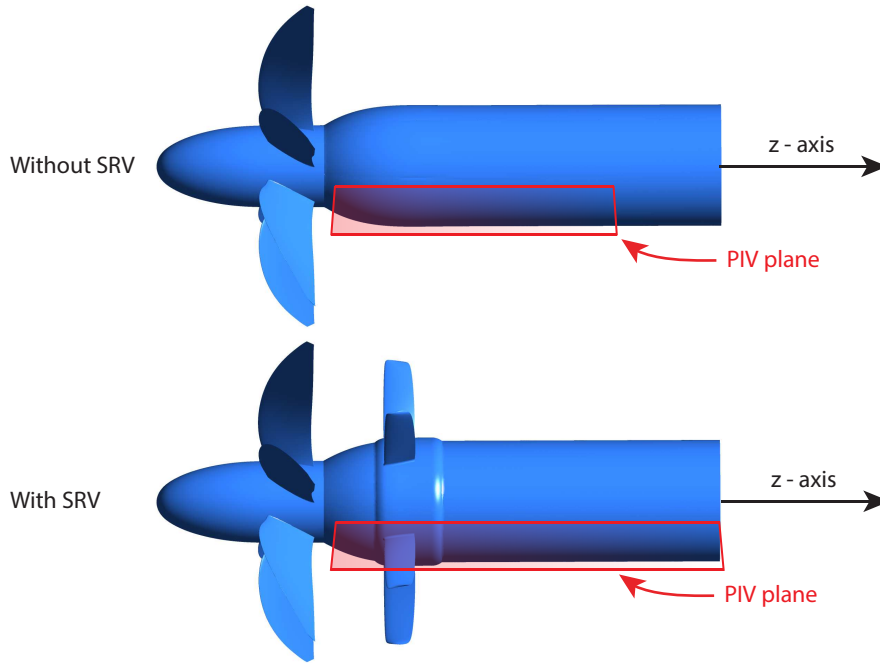


Figure 3.6: Schematic of the PIV plane without and with SRV and the definition of an instance of the propeller blade passage.

3.4.4. DESCRIPTION OF THE SLIPSTREAM AND COMPARISON WITH THE EXPERIMENT

Observe Figure 3.6 which shows the PIV plane for the case without and with SRV. At this plane, the velocity components and the vorticity perpendicular to the plane ω_t are known for one propeller blade passage. Figure 3.7 shows the axial velocity component V_a non-dimensionalized by the undisturbed speed V_∞ in these PIV planes without and with SRV from the experimental PIV measurements and from the CFD simulations at the same instance in the propeller blade passage. The propeller orientation for this instance is as shown in Figure 3.6. First consider the results without SRV. Qualitatively the wake structure is very similar and the same wake features are present. The features in the PIV measurement are more distinct with sharper gradients. The magnitude of the axial velocity extrema around the tip vortex deviate much more from the surrounding flow for the PIV measurements than for the CFD simulation and in the CFD simulation these extrema dissipate much more in downstream direction. Also, observe Figure 3.8 which shows the average over the propeller blade passage. Clearly, the axial velocity component in the wake is on average somewhat higher in the CFD simulation, while outside of the wake, the free stream velocity is equal. This is in line with the higher thrust coefficient in the CFD simulation due to the difference in advance ratio. Also there is a small difference in the contraction of the slipstream, which is only present in the CFD result.

When the SRV and mounting plate are present, the axial velocity component increases slightly in the wake of the SRV due to the reduction of the swirl angle. This phenomenon is present in the PIV measurement and CFD simulation. Also, for both the PIV measurements and the CFD simulations, the contraction of the wake is larger when the SRV are present, which can especially be seen well in the averaged results: This additional contraction can be explained by the conservation of mass and the increased axial velocity component. The PIV measurement lacks some data due to partial blocking of the PIV field of view by the SRV. Again features in the PIV measurement are more distinct and extrema more extreme. Downstream from $z/R_p = 2.0$ the flow features also deviate qualitatively: Two additional pairs of maximum and minimum different from those of the propeller tip vortex can be noticed in the PIV measurements. It will be shown later that these correspond to the tip vortex of one of the swirl recovery vanes. On average, this decreases the average axial velocity considerably in the region $0.8 < r/R_p < 1.0$ downstream of $z/R_p = 2.0$. Near the nacelle, a region of very low axial velocity can be noticed. It is unclear whether this is a measurement artefact or that it may be the wake of the mounting plate.

Figures 3.9 and 3.10 show the non-dimensionalized tangential velocity component V_t/V_∞ at the same instance in the propeller blade passage and averaged over the propeller blade passage respectively. When no SRV is present, similar conclusions can be drawn when comparing the PIV measurements and CFD simula-

tion: The wake features are more distinct in the PIV measurements and the extrema are more extreme. Also two patches of near zero tangential velocity close to the nacelle can be seen in the PIV measurement which are not as low in the CFD simulation. Together with the increased tangential velocity at slightly higher radii they are the result of the propeller blade root vortex. This root vortex is apparently more distinct in the PIV measurement. The average tangential velocity in the wake is slightly higher in the CFD simulation. This may be explained by the higher propeller power due to the fully turbulent modelling of the blade boundary layer and the difference in advance ratio.

When the SRV are present, the tangential velocity decreases considerably downstream of the SRV. This recovery of the swirl is expected: The loading on the SRV results in a downwash, opposite to the swirl of the propeller. Close to the nacelle, the PIV measurement shows regions of zero and negative tangential velocity. It is unclear whether these are the result of flow separation from the mounting plate or the loading on the SRV. These regions are not present in the CFD simulation. Again, downstream of $z/R_p = 2.0$ two additional extrema pairs can be found in the PIV measurement corresponding the tip vortex of one of the swirl recovery vanes. From the averaged tangential velocity contours, it can be concluded that the recovery of swirl is on the same order away from the nacelle, although the remaining tangential velocity in the CFD simulation is somewhat higher. That latter can be explained by the already higher tangential velocity when no SRV are present.

The vortex structure in the wake of the propeller is shown in Figure 3.11 without and with SRV at the same instance of the propeller blade passage as the previous figures. On this isosurface of the vorticity magnitude, a contour plot of the tangential vorticity component ω_t is plotted. For this latter parameter Figure 3.12 shows the corresponding contour plot in the plane as previously defined. Note the different colour bars. The vortex structure of the APIAN propeller without SRV is very regular: A root and tip vortex form a helical structure for each propeller blade. The pitch of these helices are a function of propeller rotational speed n and the wake axial and tangential velocity components, or defined together as the swirl angle. The root vortex is of a larger pitch than the tip vortex due to the larger swirl angle in that region of the wake. The tangential vorticity component is of opposite sign at the root vortex than at the tip vortex due to the opposite sign of the slope of the blade loading. The tangential vorticity component measured in the PIV plane is qualitatively similar to the CFD simulation. The tip vortex is of much higher magnitude and smaller in diameter. The root vortex is thinner and spans over a larger range of radii but is in the same order of magnitude.

With the SRV present, the wake vortical structure is disturbed: The propeller blade tip vortices are only slightly deformed by the tip vortices trailing from the SRV, since these SRV are cropped to avoid interaction with the propeller tip vortices. The propeller blade root vortices are deformed more: The SRV cut the blade root vortices and in the CFD simulation no reattachment downstream of the SRV can be noticed. The SRV root and tip vortices move downstream with the local swirl angle. The magnitude of the tangential vorticity component in both the root and tip vortices is very small, and thus they consist mainly of axial vorticity. The same differences between the PIV measurement and the CFD simulation as without SRV can be noticed. Furthermore, two additional vortices can be seen aft of $z/R_p = 2.0$ as mentioned earlier as well. These are likely the result of the tip vortex of the vane below the PIV plane: The tip vortex does not stream downstream in a straight line but 'dances' around the propeller tip vortices. Most likely, the vane tip vortex moves into or touches the PIV plane at $z/R_p = 2.1$ and $r/R_p = 0.85$ and moves into the plane again or touches it again at $z/R_p = 2.5$ and $r/R_p = 0.85$. Because of the very low tangential velocity component downstream of the SRV it is unlikely that these vortices are of two different vanes.

Figure 3.13 summarises the vortex structure without and with SRV in a schematic way including the PIV measurements. Note that the direction of vorticity is indicated by the right-hand-rule. The red propeller root vortex sheets are cut by each of the vanes. The path of the green SRV tip vortex is highly influenced by the propeller tip vortices, which is confirmed by the two additional tangential vorticity extrema in the PIV plane. When the SRV tip vortex is upstream of a propeller tip vortex, it is pushed towards the root and when it is downstream it is pulled towards the tip again. Because the tip vortices are not perpendicular to each other, the SRV tip vortex is moved in swirl direction upstream and in opposite direction downstream. Therefore it can move in and out of the PIV plane. The propeller tip vortex is also influenced by the SRV tip vortices, but because of the difference in strength, it is less disturbed.

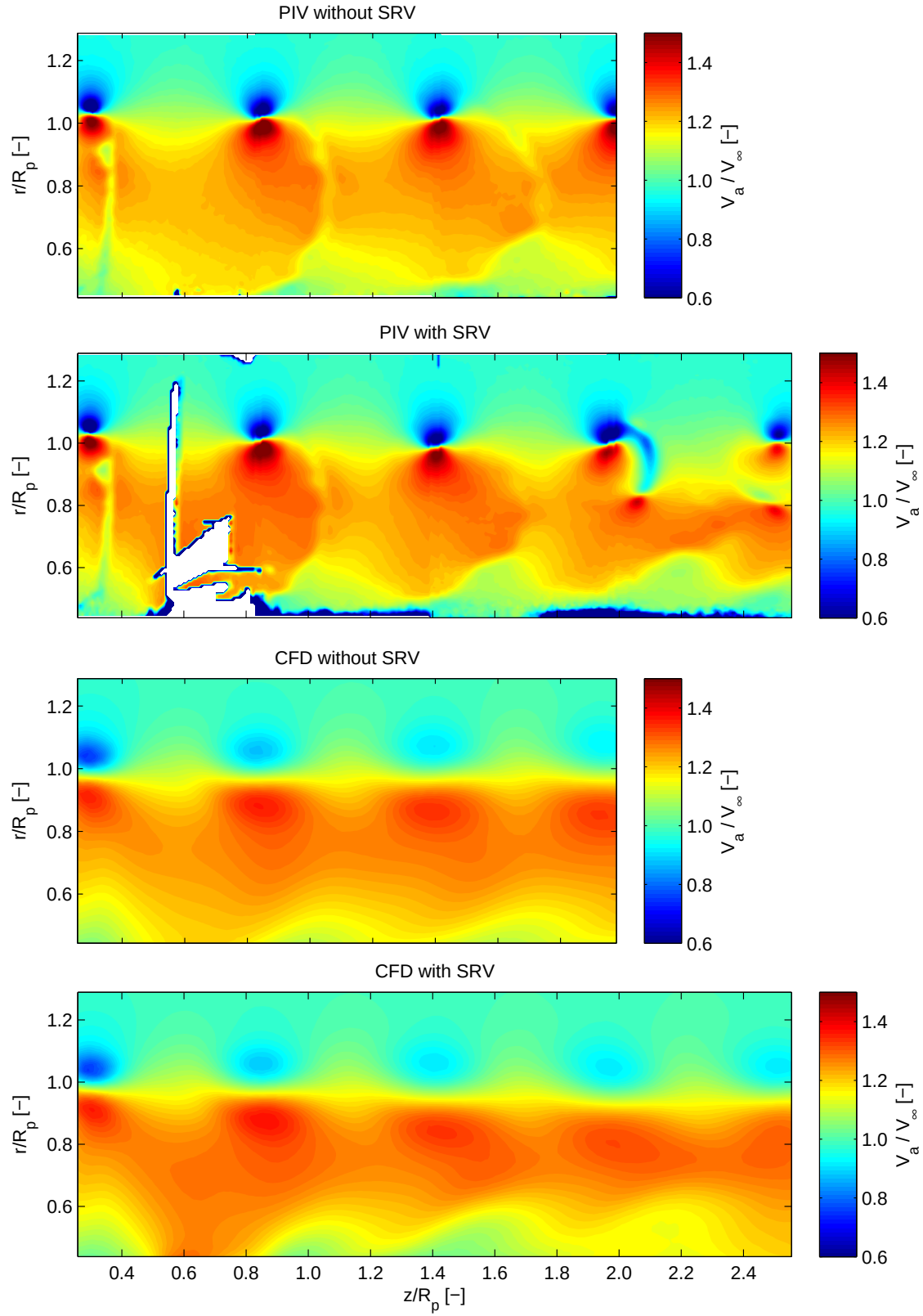


Figure 3.7: Contour plots of the non-dimensional axial velocity measured by means of PIV in the APIAN-INF wind tunnel test and simulated by means of the APIAN-INF CFD model without and with SRV in a plane and instance of the propeller blade passage defined in Figure 3.6.

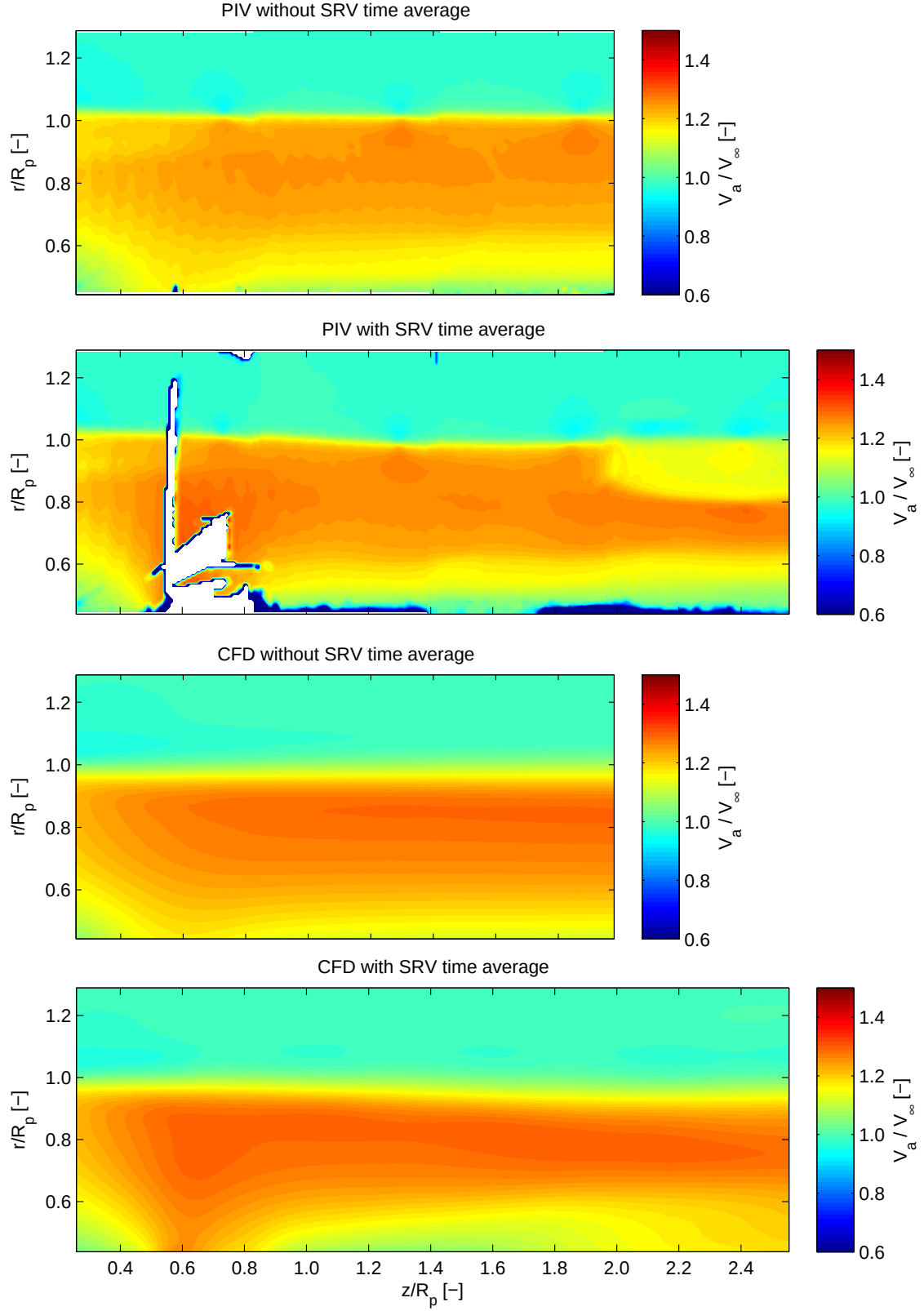


Figure 3.8: Contour plots of the non-dimensional axial velocity measured by means of PIV in the APIAN-INF wind tunnel test and simulated by means of the APIAN-INF CFD model without and with SRV in a plane defined in Figure 3.6 averaged over the propeller blade passage.

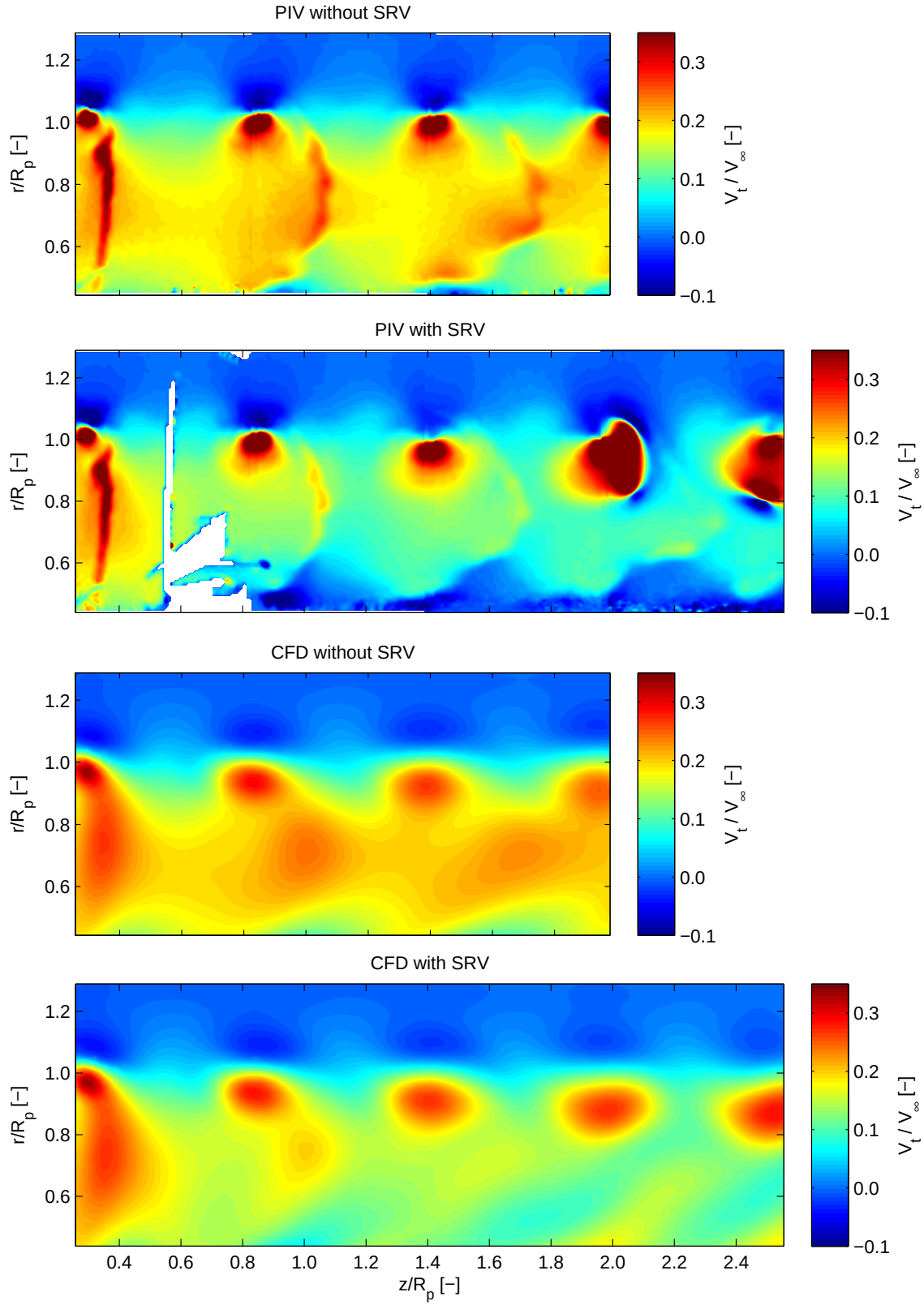


Figure 3.9: Contour plots of the non-dimensional tangential velocity measured by means of PIV in the APIAN-INF wind tunnel test and simulated by means of the APIAN-INF CFD model without and with SRV in a plane and instance of the propeller blade passage defined in Figure 3.6.

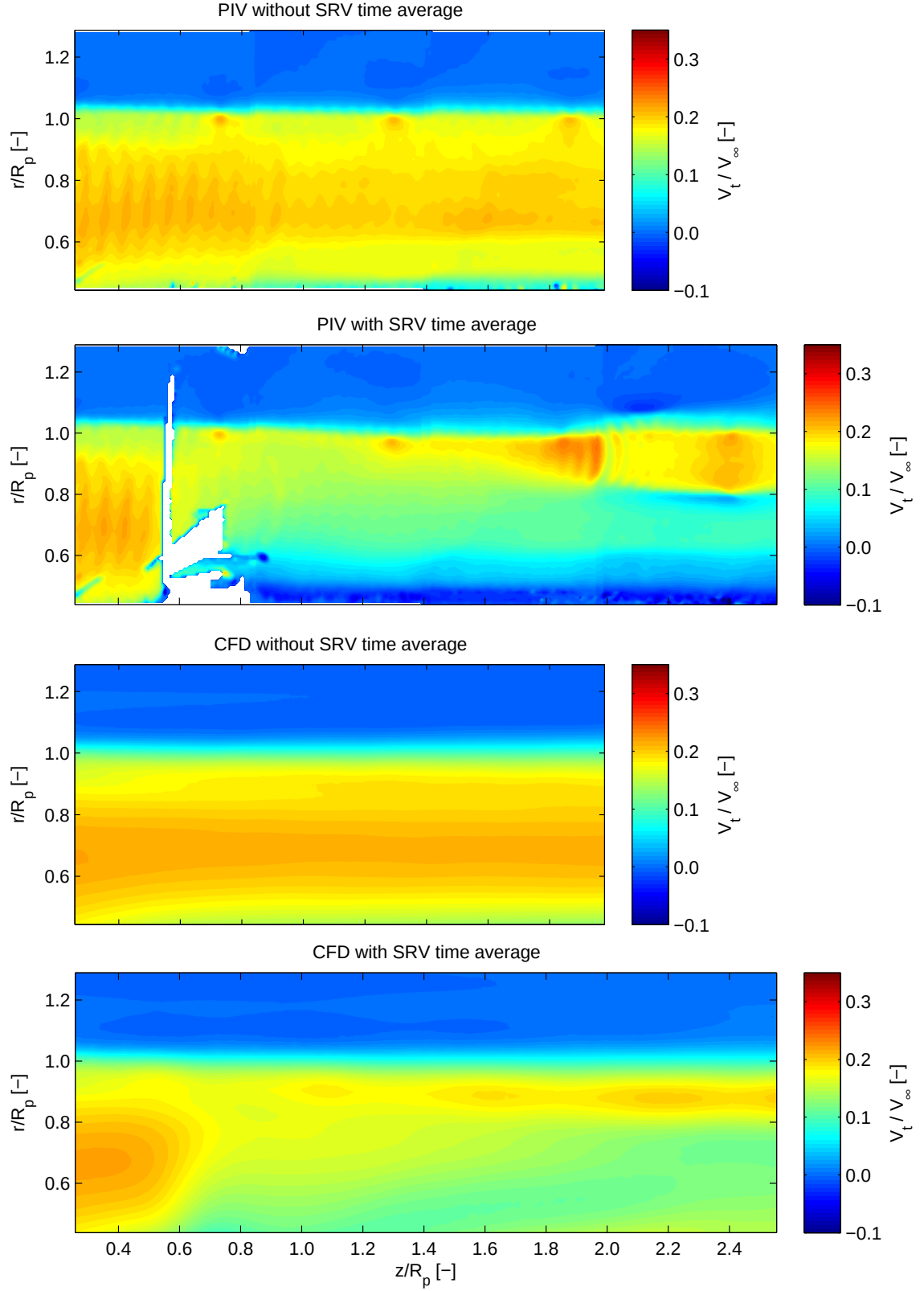


Figure 3.10: Contour plots of the non-dimensional tangential velocity measured by means of PIV in the APIAN-INF wind tunnel test and simulated by means of the APIAN-INF CFD model without and with SRV in a plane defined in Figure 3.6 averaged over the propeller blade passage.

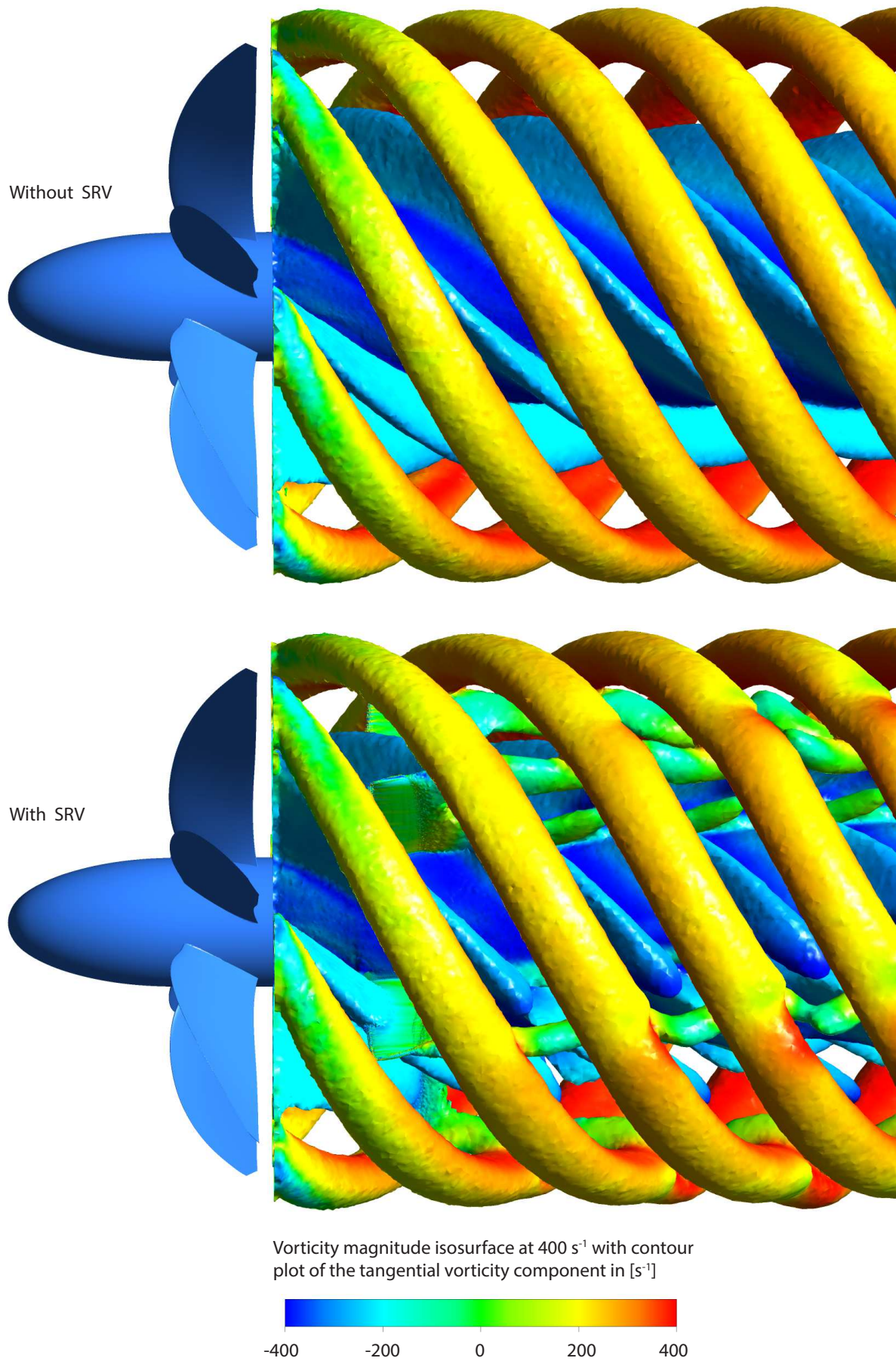


Figure 3.11: Isosurface of the vorticity magnitude behind the APIAN propeller without and with SRV including a contour plot of the tangential vorticity component at an instance of the propeller blade passage defined in Figure 3.6

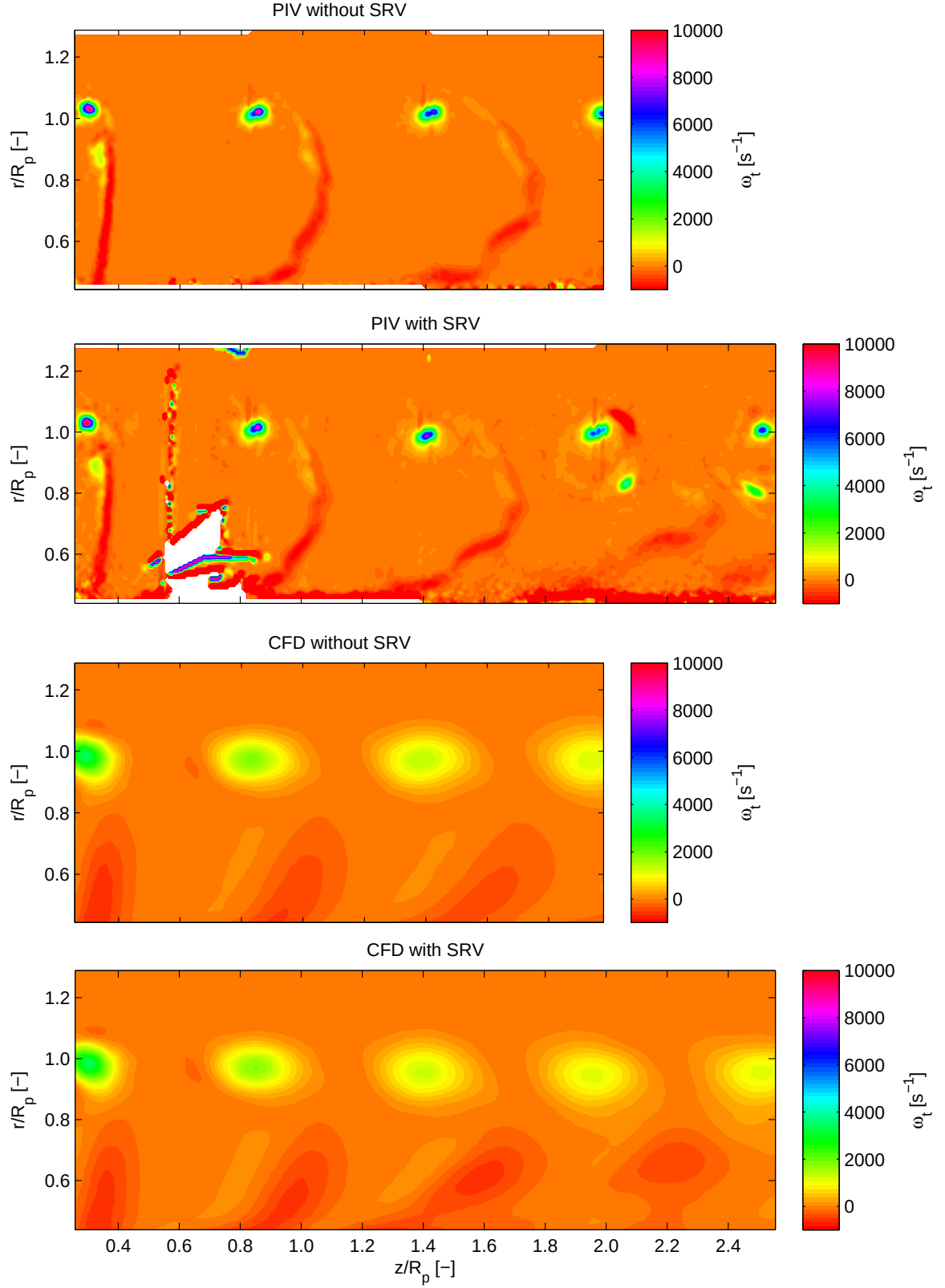


Figure 3.12: Contour plots of the tangential vorticity component measured by means of PIV in the APIAN-INF wind tunnel test and simulated by means of the APIAN-INF CFD model without and with SRV in a plane and instance of the propeller blade passage defined in Figure 3.6.

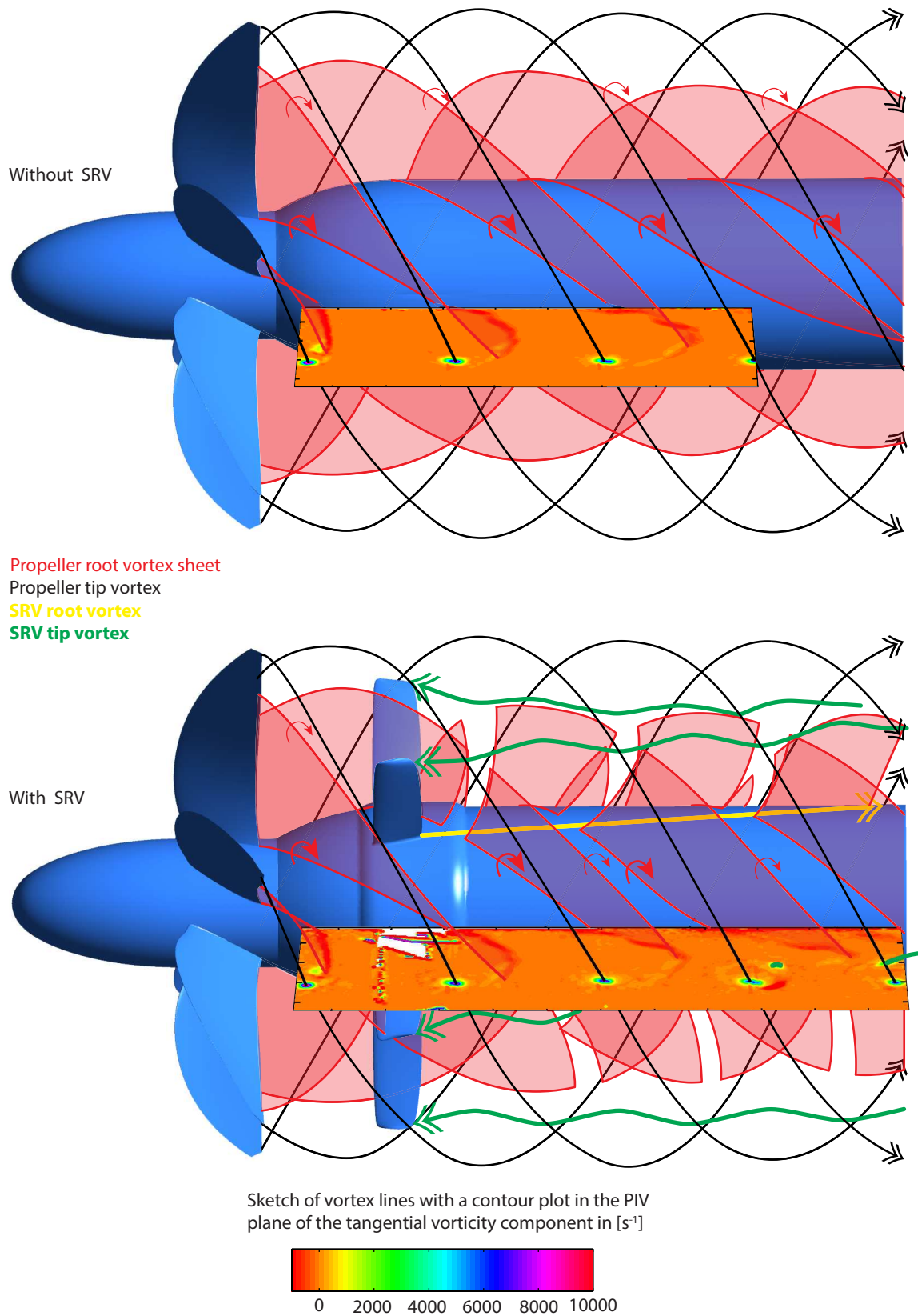


Figure 3.13: Sketch of the vortex lines without and with SRV including a contour plot of the tangential vorticity component on the PIV plane at an instance of the propeller blade passage defined in Figure 3.6. The direction of vorticity is given by the right-hand-rule.

4

SRV DESIGN PROCEDURE

The propulsive efficiency gain of the SRV from the APIAN-INF CFD model of the previous chapter was much lower than predicted by its design. Therefore, this chapter describes a procedure to design swirl recovery vanes for an isolated propeller with nacelle. The first Section 4.1 discusses whether to design for maximum SRV thrust or minimum downstream swirl. Then an SRV analysis tool is presented in Section 4.2. The routine to optimise the SRV design for the chosen objective by means of the introduced analysis tool is discussed in Section 4.3. At last, a number of designs for the APIAN propeller are presented in Section 4.4, of which two designs will be used for the propeller-SRV CFD model in the next chapter.

4.1. OPTIMISE FOR MAXIMUM THRUST OR MINIMUM DOWNSTREAM SWIRL?

As the name suggests, swirl recovery vanes are intended to recover the swirl or tangential velocity component in the wake of the propeller. When designing SRV using an optimisation routine, one can optimise the design to maximise their thrust, or to remove the swirl velocity downstream of the SRV. The question can be raised whether these two objectives lead to the same design. One can derive a set of closed equations for a single elliptical vane as pictured in Figure 4.1 in a constant swirl wake to investigate this question. Observe Figure 4.2 for the definition of the variables.

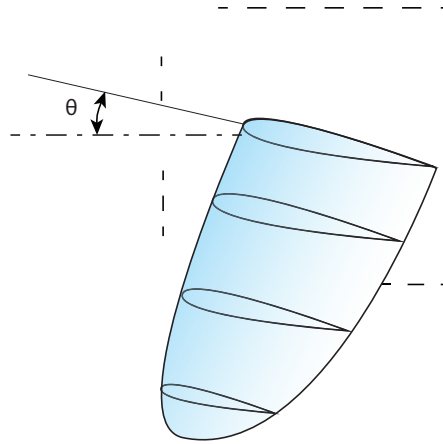


Figure 4.1: Elliptical zero-twist swirl recovery vane mounted on an infinite wall under a pitch θ .

Define an SRV thrust coefficient C_T as:

$$C_T = \frac{2T}{\rho_\infty V_\infty^2 S_{SRV}} \quad (4.1)$$

where T is the SRV net thrust, ρ_∞ is the undisturbed density, V_∞ the undisturbed air speed and S_{SRV} the SRV planform area. An equation for C_T can be derived as function of the SRV lift coefficient C_L , drag coefficient

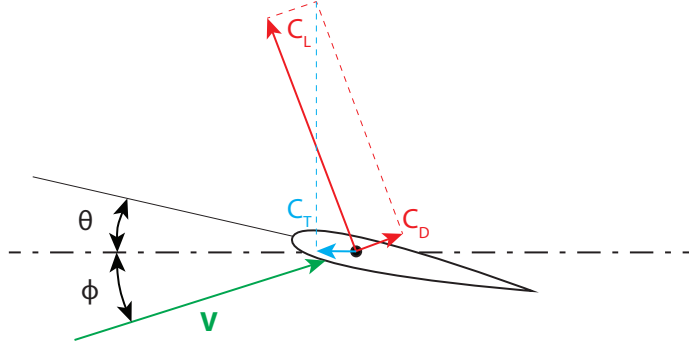


Figure 4.2: Elliptical zero-twist swirl recovery vane mounted on an infinite wall under a pitch θ with a velocity $\mathbf{V}$ with swirl angle ϕ .

C_D and swirl angle ϕ :

$$C_T = C_L \sin(\phi) - C_D \cos(\phi) \quad (4.2)$$

Lets maximise C_T for a given ϕ , V_∞ and SRV geometry. The only variable is the SRV pitch θ , which controls the SRV loading indirectly. A maximum is found when:

$$\frac{dC_T}{d\theta}(\theta_{max}) = 0 \quad (4.3)$$

with the condition that at that point

$$\frac{d^2C_T}{d\theta^2}(\theta_{max}) < 0 \quad (4.4)$$

For an elliptical wing without twist and constant airfoil section, the following analytical approximations for C_L and C_D can be derived from for instance Abbott and Von Doenhoff [39]:

$$C_L = C_{l_\alpha} \left(\frac{\mathcal{R}}{\mathcal{R} + 2} \right) (\theta + \phi - \alpha_0) \quad (4.5)$$

$$C_D = C_{d_0} + b(C_{l_0} - C_L)^2 + \frac{C_L^2}{\pi \mathcal{R}} \quad (4.6)$$

where C_{l_α} is the section lift curve slope, $\mathcal{R}$ the SRV aspect ratio, α_0 the section zero-lift angle of attack, C_{d_0} the section zero-lift drag coefficient b a constant determining the assumed parabolic section drag polar and C_{l_0} the section lift coefficient at zero angle of attack. Taking the first and second derivative of C_T to θ result in:

$$\frac{dC_T}{d\theta} = \frac{dC_L}{d\theta} \sin(\phi) - \frac{dC_D}{d\theta} \cos(\phi) \quad (4.7)$$

$$\frac{d^2C_T}{d\theta^2} = \frac{d^2C_L}{d\theta^2} \sin(\phi) - \frac{d^2C_D}{d\theta^2} \cos(\phi) \quad (4.8)$$

Taking the first and second derivative of C_L and C_D to θ give

$$\frac{dC_L}{d\theta} = C_{l_\alpha} \left(\frac{\mathcal{R}}{\mathcal{R} + 2} \right) \quad (4.9)$$

$$\frac{d^2C_L}{d\theta^2} = 0 \quad (4.10)$$

and

$$\frac{dC_D}{d\theta} = \left(-2b(C_{l_0} - C_L) + \frac{2C_L}{\pi \mathcal{R}} \right) \frac{dC_L}{d\theta} = \left(-2b(C_{l_0} - C_L) + \frac{2C_L}{\pi \mathcal{R}} \right) C_{l_\alpha} \left(\frac{\mathcal{R}}{\mathcal{R} + 2} \right) \quad (4.11)$$

$$\frac{d^2C_D}{d\theta^2} = \left(2b \frac{dC_L}{d\theta} + \frac{2}{\pi \mathcal{R}} \frac{dC_L}{d\theta} \right) C_{l_\alpha} \left(\frac{\mathcal{R}}{\mathcal{R} + 2} \right) = 2 \left(b + \frac{1}{\pi \mathcal{R}} \right) \left(C_{l_\alpha} \left(\frac{\mathcal{R}}{\mathcal{R} + 2} \right) \right)^2 \quad (4.12)$$

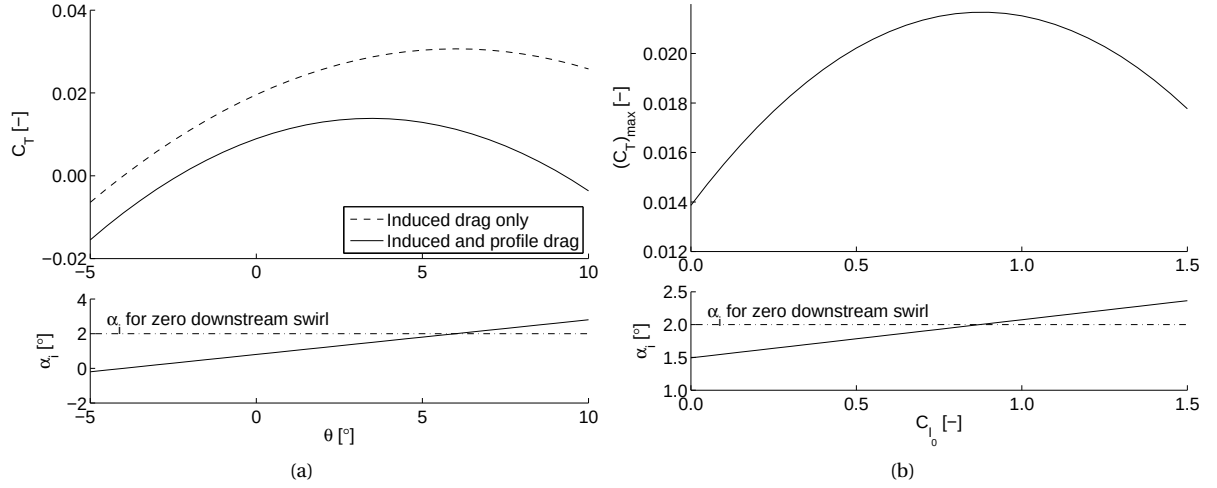


Figure 4.3: (a) SRV thrust C_T and induced angle of attack α_i versus pitch angle θ and (b) maximum SRV thrust $(C_T)_{max}$ and α_i versus section lift coefficient at zero angle of attack C_{l0} for an elliptical SRV with zero twist ($C_{d0} = 0.009$, $b = 0.0136$, $C_{la} = 2\pi \text{ rad}^{-1}$) mounted on an infinite wall under a constant swirl inflow of $\phi = 4^\circ$.

The resulting equation to solve is

$$\begin{aligned} \frac{dC_T}{d\theta} &= C_{l\alpha} \left(\frac{R}{R+2} \right) \sin(\phi) - \left(-2b(C_{l0} - C_L) + \frac{2C_L}{\pi R} \right) C_{l\alpha} \left(\frac{R}{R+2} \right) \cos(\phi) = 0 \\ \sin(\phi) - \left(-2b(C_{l0} - C_L) + \frac{2C_L}{\pi R} \right) \cos(\phi) &= 0 \\ \tan(\phi) &= \left(-2b(C_{l0} - C_L) + \frac{2C_L}{\pi R} \right) \end{aligned} \quad (4.13)$$

with the condition that

$$\frac{d^2 C_T}{d\theta^2} = -2 \left(b + \frac{1}{\pi R} \right) \left(C_{l\alpha} \left(\frac{R}{R+2} \right) \right)^2 \cos(\phi) < 0 \quad (4.14)$$

So a maximum is found for Equation (4.13) when:

$$\left(b + \frac{1}{\pi R} \right) \cos(\phi) > 0 \quad (4.15)$$

which is always true for cases of interest. When only the induced drag is considered and for small angles ϕ , the following condition results in a maximum for C_T :

$$\phi = \frac{2C_L}{\pi R} = 2\alpha_i \quad (4.16)$$

where α_i is the induced angle of attack. When the assumption is made that the downwash far downstream of the SRV is twice the induced angle of attack as proven by Prandtl [40], then the maximum in C_T is found when the entire swirl is removed behind the SRV. However, when profile drag is considered as well, the maximum in C_T shifts. Observe Figure 4.3 (a) which shows C_T and α_i versus θ for a symmetric airfoil with and without profile drag. When only the induced drag is considered, the maximum in C_T corresponds to $\alpha_i = \frac{\phi}{2} = 2^\circ$, which is the induced angle of attack that results in zero downstream swirl. When profile drag is considered too, the maximum in C_T shifts to a lower pitch angle θ and the corresponding α_i is also lower due to the lower SRV loading: Maximum SRV thrust is found when some swirl is not recovered. If θ would have been optimised for minimum downstream swirl, the maximum SRV thrust and hence maximum efficiency benefit would not have been found. In order to further investigate the influence of the airfoil properties, Equation (4.13) can be solved for C_L :

$$C_L = \frac{\pi R (\tan(\phi) + 2bC_{l0})}{2b\pi R + 2} \quad (4.17)$$

When inserting this equation and Equation (4.6) into Equation (4.2), the maximum in C_T is found:

$$(C_T)_{max} = \frac{-4 \left(C_{d0} + bC_{l0}^2 + b\pi R C_{d0} \right) \cos(\phi) + \pi R \sin(\phi) (4bC_{l0} + \tan(\phi))}{4b\pi R + 4} \quad (4.18)$$

This equation is plotted in Figure 4.3 (b) against C_{l0} to investigate the influence of camber. The corresponding α_i is plotted as well. At $C_{l0} = 0$ the maximum of Figure 4.3 (a) is located. When C_{l0} is increased and thus the effect of camber is increased, a maximum in $(C_T)_{max}$ is found corresponding to $\alpha_i = \frac{\phi}{2} = 2^\circ$, again complete removal of the downstream swirl. This maximum corresponds to quite a large value of C_{l0} which needs to be even larger for larger swirl angles. Since it was shown in Figure 2.17 that the swirl angle behind a propeller can be considerably higher than $\phi = 4^\circ$, it is not necessarily possible to find an airfoil section with these properties. Based on this simplified analysis of swirl recovery vanes, it can be concluded that in an optimisation routine the SRV thrust should be the objective to maximise, since the most optimum SRV which also removes the downstream swirl may not be found because of unrealistic airfoil section property requirement.

4.2. SRV ANALYSIS TOOL

This section presents a tool written by the author in MATLAB R2014a to analyse the performance of swirl recovery vanes. The concept is to analyse swirl recovery vanes with a low order method using a velocity inflow field extracted from the propeller CFD model. This velocity field is used as input for a lifting line code adapted for non-uniform inflow to find the induced velocity field of the swirl recovery vanes mounted on a nacelle. XFOIL by Drela and Youngren [41] is used to find the SRV airfoil properties. A flow chart of this program is given in Figure 4.4. The main assumption for this design procedure is that the swirl recovery vanes have a negligible effect on the propeller performance. Gazzaniga and Rose [9] has shown that the upstream effect of the vanes is present but small, especially for larger propeller-SRV spacing. Another important assumption is that the static pressure gradient in the propeller slipstream does not influence the SRV loading. The airfoil properties are discussed in Subsection 4.2.1, the definition of the SRV planform geometry in Subsection 4.2.2, the adapted lifting line theory in Subsection 4.2.3 and the assumptions are summarized in Subsection 4.2.4.

4.2.1. SRV AIRFOIL PROPERTIES

In the current version of the tool a single airfoil type is used along the whole span of all swirl recovery vanes. This is currently limited to airfoils of the NACA-4 series as defined for instance in Abbott and Von Doenhoff [39]. Three variables are required to define a NACA 4 series airfoil: The airfoil thickness $\frac{t}{c}$, the maximum camber $\frac{cm_{max}}{c}$ and the position of the maximum camber $\frac{x_{cm_{max}}}{c}$. These are given as a ratio of the chord. XFOIL by Drela and Youngren [41] is used to find the lift and profile drag coefficient curve, including differentiation of friction and pressure drag coefficient. This program for the design and analysis of subsonic isolated airfoils makes use of a linear-vorticity panel method with Karman-Tsien compressibility correction for the inviscid flow calculation according to Drela [42]. A two-equation lagged dissipation integral method is used for the viscous layers. Transition from the laminar to turbulent boundary layer is determined by $e^{n_{cr}}$ type amplification factor with a suitable value for n_{cr} as user input. For the purpose of this SRV analysis tool, the required input for XFOIL is limited to the airfoil coordinates, the number of panels, the number of iterations, a suitable Reynolds number based on chord Re_c , the undisturbed Mach number M_∞ and critical amplification factor n_{cr} . For the purpose of speed, the same output data is used for all airfoils along the span, irrespective of different local Re_c , M and n_{cr} . The Reynolds number in this tool is based on the root chord c_r , the Mach number on free-stream conditions M_∞ and n_{cr} is chosen very low (a value of 0.01 works well) to trigger turbulent flow over most of the airfoil. A low value of n_{cr} is chosen for convergence of XFOIL, because the chosen CFD turbulence model considers only turbulent boundary layers and because the turbulence level in the wake of the propeller is quite high. XFOIL is required to converge for a large range of airfoils easily, especially when used in an optimisation routine as discussed in Section 4.3. The turbulence level can be defined as [43]:

$$T_u \equiv \frac{u'}{U} = \frac{\sqrt{\frac{2}{3}k}}{U} \quad (4.19)$$

Where u' is the root mean square of the turbulent velocity fluctuations, U the mean velocity and k the turbulence kinetic energy. Figure 4.5 shows a contour plot of the turbulence level in the flow behind the APIAN propeller at the location of the wake rake. It also shows experimental results of the influence of the turbulence level on the critical amplification factor for a flat plate. Clearly, low values of n_{cr} can be expected when swirl recovery vanes are placed in this flow.

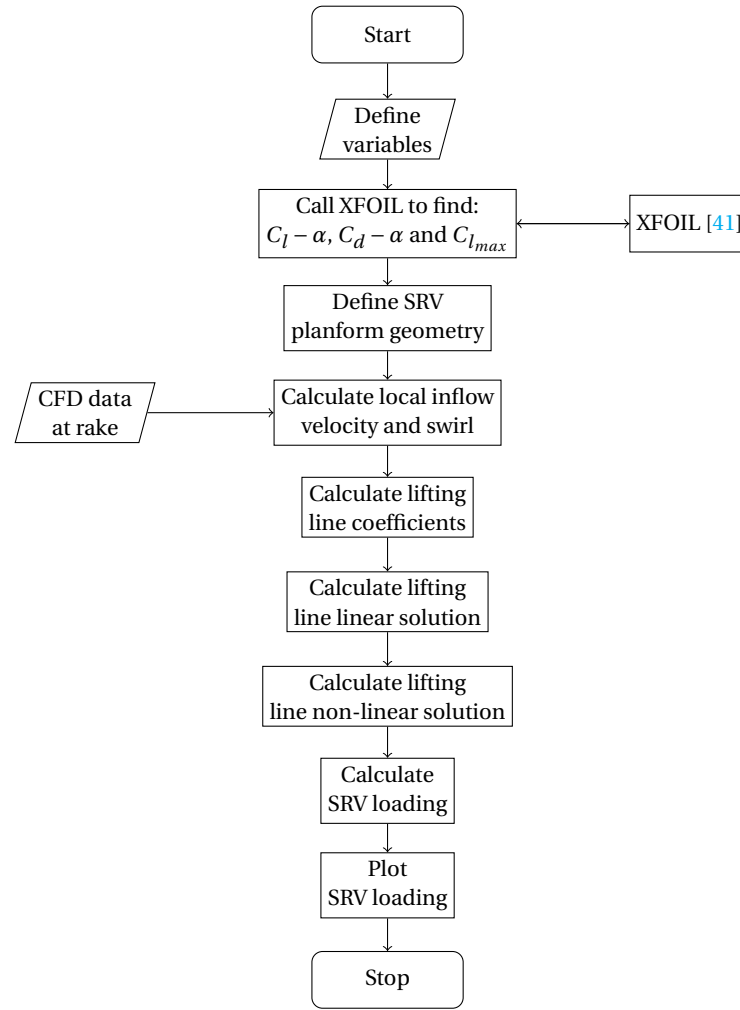


Figure 4.4: Flow chart of the SRV analysis tool.

4.2.2. SRV PLANFORM GEOMETRY

For the purpose of aerodynamic shape optimisation, as few variables as possible are desired to define the SRV planform geometry. Figure 4.6 gives an overview of all the variables needed to define the SRV planform. The root section located downstream of the propeller at a distance z_{SRV} is defined by the nacelle radius R_n , root chord c_r and root pitch θ_r . The pitch is defined positive as shown in Figure 4.2. The tip section is then defined with respect to the root section by a quarter chord sweep angle $\Lambda_{c/4}$, the SRV span s , a tip-to-root chord taper ratio λ and a twist angle Θ (wash-in positive for positive swirl). The chord and pitch distribution in between the root and tip section are defined by a superposition of the linear distribution defined by the root and tip section and a non-linear distribution based on Class/Shape Transformation (CST). This transformation method explained in Kulfan [44] is desirable for aerodynamic design optimizations to limit the number of variables. The parameter ζ defining the distribution of a geometric quantity along a coordinate ψ from 0 to 1, is defined by the multiplication of a class and shape function:

$$\zeta = C_{N_2}^{N_1}(\psi)S(\psi) \quad (4.20)$$

where $C_{N_2}^{N_1}(\psi)$ is the class function and $S(\psi)$ the shape function. The class function defines the class of shapes. For the purpose of the chord and pitch distribution, the following class function is well suited:

$$C_{N_2}^{N_1}(\psi) = (\psi)^{N_1} (1 - \psi)^{N_2} \quad (4.21)$$

The variables N_1 and N_2 determine the root and tip slope. $N_1 = 0.5$ and $N_2 = 1.0$ describes for instance the shape of NACA-type round nose and point aft end airfoil, while $N_1 = 0.5$ and $N_2 = 0.5$ describes an ellipse.

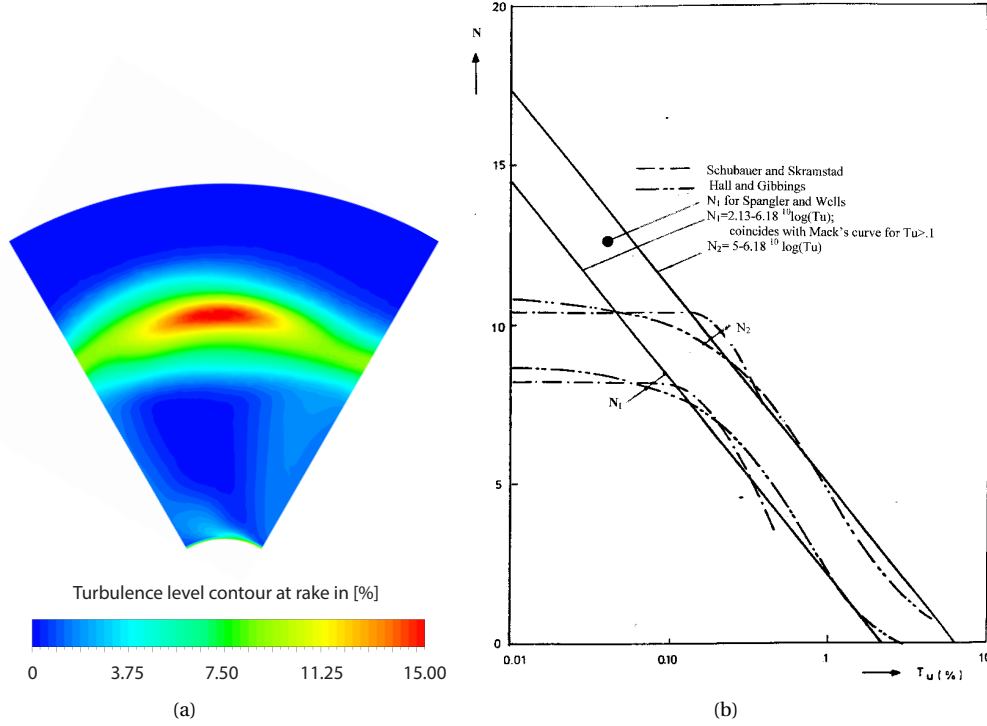


Figure 4.5: (a) Turbulence level T_u at the wake rake of the propeller CFD model for $J = 1.0$ and (b) corresponding values of n_{cr} for a flat plate from Van Ingen [43] (Note that N in figure (b) is the critical amplification factor n_{cr}).

The shape function is given as:

$$S(\psi) = \sum_{i=1}^m A_i S_i(\psi) \quad (4.22)$$

where A_i are the shape function coefficients and the component shape function $S_i(\psi)$ is a Bernstein polynomial of order n :

$$S_i(\psi) = K_i \psi^i (1 - \psi)^{n-i} \quad (4.23)$$

with the binomial coefficient K_i as:

$$K_i = \binom{m}{i} = \frac{m!}{i!(n-i)!} \quad (4.24)$$

The number of shape function coefficients m is free to choose and determines the freedom of possible shapes. It is found that $m = 3$ provides sufficient freedom for the chord and pitch distribution. While for the pitch distribution the class function with $N_{1,\theta}$ and $N_{2,\theta}$ fixed to 1.0 is appropriate, the chord distribution class function is chosen as $N_{1,c} = 1.0$ and $N_{2,c}$ free between 0.5 and 1.0. This allows for a leading and trailing edge at the tip tangent to the tip section. As an example, in Figure 4.6 is $N_{2,c}$ equal to 0.5. to conclude, in order to define the non-linear part of the pitch and chord distribution, two shape function coefficient vectors $\mathbf{A}_\theta$ and $\mathbf{A}_c$ and one class function parameter $N_{2,c}$ are introduced.

4.2.3. ADAPTED LIFTING LINE THEORY

The lifting-line method used to calculate the lift and induced drag of the swirl recovery vanes is based on Phillips and Snyder [45]. The classical lifting line theory is based on the hypothesis that each spanwise section of a finite wing has a section lift equal to that on a similar section of an infinite wing with the same section circulation. A two-dimensional version of the Kutta-Joukowski theorem is then applied at each section. Instead of applying the two-dimensional Kutta-Joukowski theorem, this numerical lifting-line method is based on a three-dimensional vortex lifting law. As well, the bound vorticity of each section is taken into account in calculating the induced velocity vector at a section. In this way sweep is better accounted for. Furthermore, this lifting-line method can take the non-linear aerodynamic behaviour into account at large angles of attack,

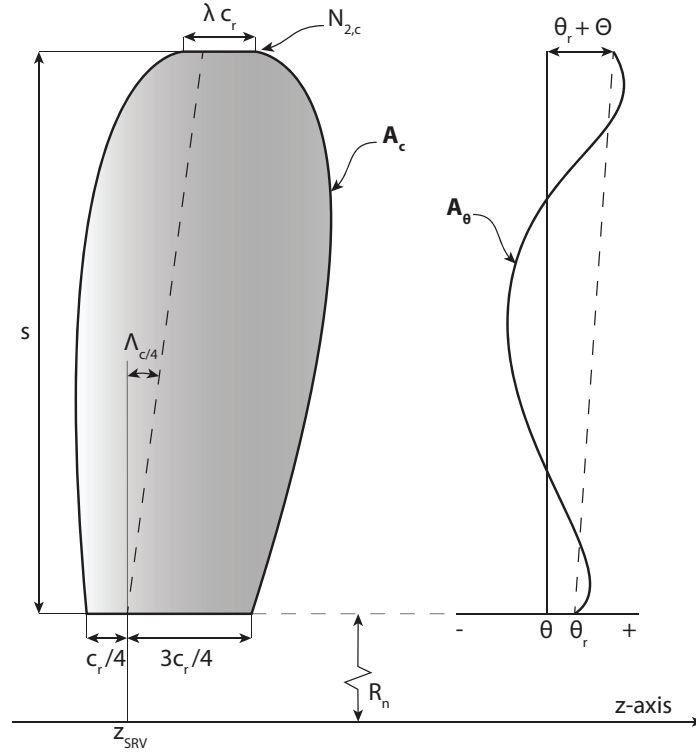


Figure 4.6: Definition of SRV planform.

up to $C_{l_{max}}$. This is especially important for evaluating off-design conditions. This method is further modified by the author to allow for a non-uniform inflow and to account for the effect of the nacelle. This method including these modifications are discussed here.

In this lifting line method an aerodynamic surface is synthesized using a distribution of horseshoe vortices along the quarter chord line. The bound portion of each horseshoe vortex is coincident with the quarter chord line, the trailing vortices run downstream in an assumed direction $\mathbf{u}_t$. $\mathbf{u}_t$ is assumed to be in the rotation axis direction, since most swirl will be recovered. Observe Figure 4.7 for a sketch of the definition of the horseshoe vortices in case of swirl recovery vanes. The velocity vector induced at a point k in space by a horseshoe vortex at panel j is:

$$\mathbf{V}_{i_k} = \frac{\Gamma_j}{4\pi} \left[\frac{\mathbf{u}_t \times \mathbf{r}_{jB}}{r_{jB}(r_{jB} - \mathbf{u}_t \cdot \mathbf{r}_{jB})} + \frac{(r_{jA} + r_{jB})(\mathbf{r}_{jA} \times \mathbf{r}_{jB})}{r_{jA}r_{jB}(r_{jA}r_{jB} + \mathbf{r}_{jA} \cdot \mathbf{r}_{jB})} - \frac{\mathbf{u}_t \times \mathbf{r}_{jA}}{r_{jA}(r_{jA} - \mathbf{u}_t \cdot \mathbf{r}_{jA})} \right] \quad (4.25)$$

In this Equation Γ_j is the strength of the horseshoe vortex at panel j and $\mathbf{r}_{jA}$ and $\mathbf{r}_{jB}$ the vectors from corner points j_A and j_B of panel j to the control point of panel k . The general three dimensional vortex lifting law is described by:

$$\mathbf{F} = \iiint_V \rho (\mathbf{V} \times \boldsymbol{\omega}) dV \quad (4.26)$$

where $\mathbf{F}$ is the force vector, ρ the air density and $\boldsymbol{\omega}$ the vorticity. Applied to a differential segment of the lifting line with spanwise length vector $d\mathbf{l}_k$ an equation for the aerodynamic force vector on panel k is obtained:

$$d\mathbf{F}_k = \rho \Gamma_k \mathbf{V}_k \times d\mathbf{l}_k \quad (4.27)$$

The velocity vector at panel k $\mathbf{V}_k$ is the sum of the non-uniform inflow velocity $\mathbf{V}_{in_k}$ and the induced velocity:

$$\mathbf{V}_k = \mathbf{V}_{in_k} + \mathbf{V}_{i_k} = \mathbf{V}_{in_k} + \sum_{j=1}^N \frac{\Gamma_j (v_{jk} + (v_{jk})_{im})}{c_j} \quad (4.28)$$

where c_j is the chord of panel j and v_{jk} and $(v_{jk})_{im}$ the dimensionless velocity induced by the real part and the imaginary part of the lifting line respectively. The imaginary part of the lifting line, located inside the

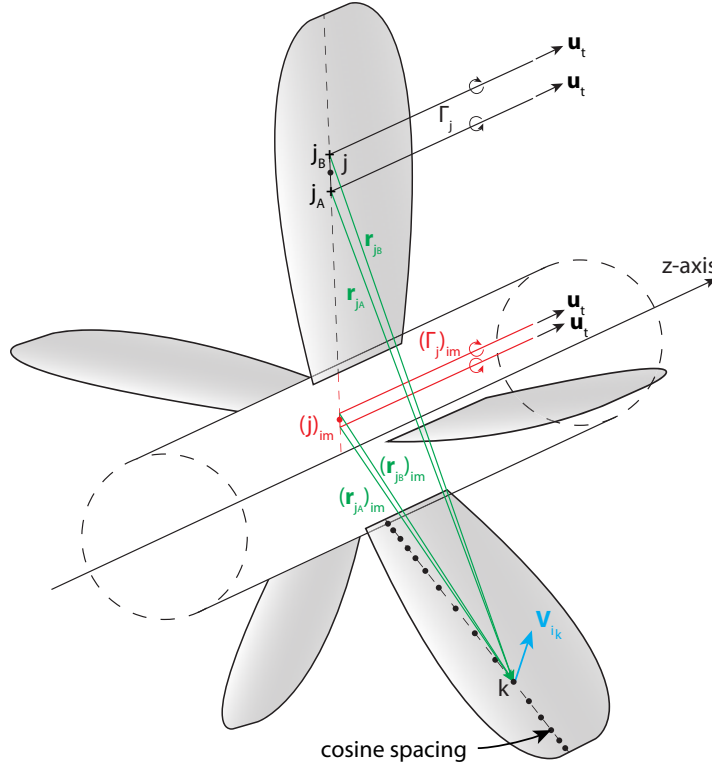


Figure 4.7: Definition of the lifting line parameters.

nacelle, is the image of the real part with equal vortex strengths to account for the effect of the nacelle. The image is constructed with the following equation for calculating the radial position of the horseshoe vortex image inside a cylinder from Durand [46]:

$$(r)_{\text{im}} = \frac{R_n^2}{r} \quad (4.29)$$

The dimensionless velocity by the real part v_{jk} can be derived from Equation (4.25) as:

$$v_{jk} \equiv \begin{cases} \frac{c_j}{4\pi} \left[\frac{\mathbf{u}_t \times \mathbf{r}_{jB}}{r_{jB}(r_{jB} - \mathbf{u}_t \cdot \mathbf{r}_{jB})} + \frac{(r_{jA} + r_{jB})(\mathbf{r}_{jA} \times \mathbf{r}_{jB})}{r_{jA} r_{jB} (r_{jA} r_{jB} + \mathbf{r}_{jA} \cdot \mathbf{r}_{jB})} - \frac{\mathbf{u}_t \times \mathbf{r}_{jA}}{r_{jA}(r_{jA} - \mathbf{u}_t \cdot \mathbf{r}_{jA})} \right], & j \neq k \\ \frac{c_j}{4\pi} \left[\frac{\mathbf{u}_t \times \mathbf{r}_{jB}}{r_{jB}(r_{jB} - \mathbf{u}_t \cdot \mathbf{r}_{jB})} - \frac{\mathbf{u}_t \times \mathbf{r}_{jA}}{r_{jA}(r_{jA} - \mathbf{u}_t \cdot \mathbf{r}_{jA})} \right], & j = k \end{cases} \quad (4.30)$$

When $j = k$ the bound vortex filament is excluded since it cannot induce a velocity when the control point is located on the vortex filament. The dimensionless induced velocity by the imaginary part $(v_{jk})_{\text{im}}$ is calculated as:

$$(v_{jk})_{\text{im}} \equiv \frac{c_j}{4\pi} \left[\frac{\mathbf{u}_t \times (\mathbf{r}_{jB})_{\text{im}}}{(r_{jB})_{\text{im}}((r_{jB})_{\text{im}} - \mathbf{u}_t \cdot (\mathbf{r}_{jB})_{\text{im}})} + \frac{((r_{jA})_{\text{im}} + (r_{jB})_{\text{im}})((\mathbf{r}_{jA})_{\text{im}} \times (\mathbf{r}_{jB})_{\text{im}})}{(r_{jA})_{\text{im}}(r_{jB})_{\text{im}}((r_{jA})_{\text{im}}(r_{jB})_{\text{im}} + (\mathbf{r}_{jA})_{\text{im}} \cdot (\mathbf{r}_{jB})_{\text{im}})} - \frac{\mathbf{u}_t \times (\mathbf{r}_{jA})_{\text{im}}}{(r_{jA})_{\text{im}}((r_{jA})_{\text{im}} - \mathbf{u}_t \cdot (\mathbf{r}_{jA})_{\text{im}})} \right] \quad (4.31)$$

This is similar to Equation (4.30) for $j \neq k$. No special condition for $j = k$ is needed since the induced velocity is not evaluated on the imaginary vortex filaments.

For the magnitude of the aerodynamic force on panel k the following equation can be defined:

$$|d\mathbf{F}_k| = \frac{1}{2} \rho V_{\text{in}_k}^2 C_{l_k}(\alpha_k) dS_k \quad (4.32)$$

In this equation $C_{l_k}(\alpha_k)$ is the section lift coefficient as a function of the local angle of attack α_k and dS_k the area of panel k . This equation can now be set equal to the magnitude of Equation (4.27) making use of

Equation (4.28) as well:

$$\left| \rho \Gamma_k \left(\mathbf{v}_{\text{in}_k} + \sum_{j=1}^N \frac{\Gamma_j (v_{jk} + (v_{jk})_{\text{im}})}{c_j} \right) \times d\mathbf{l}_k \right| = \frac{1}{2} \rho V_{\text{in}_k}^2 C_{l_k}(\alpha_k) dS_k \quad (4.33)$$

The following dimensionless quantities are introduced, the inflow velocity unit vector $\mathbf{v}_{\text{in}_k}$, dimensionless spanwise panel length vector ζ_k and dimensionless vortex strength G_k of panel k :

$$\mathbf{v}_{\text{in}_k} = \frac{\mathbf{V}_{\text{in}_k}}{V_{\text{in}_k}} \quad \zeta_k = c_k \frac{d\mathbf{l}_k}{dS_k} \quad G_k = \frac{\Gamma_k}{c_k V_{\text{in}_k}} \quad (4.34)$$

Equation (4.35) can now be written as:

$$2 \left| \left(\mathbf{v}_{\text{in}_k} + \sum_{j=1}^N G_j (v_{jk} + (v_{jk})_{\text{im}}) \right) \times \zeta_k \right| G_k - C_{l_k}(\alpha_k) = 0 \quad (4.35)$$

This equation can be written for N different control points, one for each of the N real horseshoe vortices. This results in a system of N non-linear equations that can be solved for the N unknown dimensionless vortex strengths G_k . In order to take the mutual influence of the swirl recovery vanes on each other into account, a lifting line per vane is necessary, resulting in a total number of unknowns of:

$$N = N_{hv} N_{SRV} \quad (4.36)$$

where N_{hv} are the number of real horseshoe vortices on a single vane. A cosine distribution with decreasing spacing towards the root and tip as sketched in Figure 4.7 with $N_{hv} = 40$ horseshoe vortices per vane gives the best compromise between speed and accuracy according to Phillips and Snyder [45].

The system of non-linear equations from Equation (4.35) can be solved by means of Newton's method with an initial linearized estimate. Appendix A describes this method in more detail. Necessary for an accurate solution are accurate estimates of the variation of section aerodynamic properties with angle of attack. This is where XFOIL comes in as discussed in Subsection 4.2.1.

In order to verify the SRV analysis tool, it can be compared to the analytical solution of the lift and induced drag for an elliptical zero-twist wing. For such a wing both the distribution of C_l and C_{d_i} are constant along the span and can be derived from Equations (4.5) and (4.6). Figure 4.8 shows a comparison of the lift and induced drag distribution from the analytic solution and from the SRV analysis tool. The nacelle radius R_n is set to a very large value and the number of vanes is set to one to mimic reflection of an infinite wall. Both curves coincide which gives some indication that the lifting line tool functions correctly. Also a curve for a smaller nacelle radius $R_n = 60.75 \text{ mm}$ and a curve for the same smaller nacelle radius with five vanes are plotted in this figure to show the effect of the nacelle and the effect of the other vanes on the lift and induced drag distribution. Both effects reduce the lift and increase the induced drag towards the root.

4.2.4. ASSUMPTIONS

In the SRV analysis tool a number of assumptions are made in order to simplify the tool. Most of these assumptions are already mentioned in the previous subsections. In this subsection all assumptions are summarized and their validity is tested in the next chapter.

A1: The propeller performance is not affected by the addition of SRV.

A2: The static pressure gradient in the propeller slipstream does not affect SRV thrust.

A3: The SRV planform aspect ratio is large enough ($\mathcal{R} \geq 4$ according to Phillips and Snyder [45] and $\mathcal{R} \geq 3$ according to Prandtl [40]) such that the loading can be represented by a discrete lifting line and no chordwise distribution of loading needs to be considered.

A4: The trailing vortex filaments are assumed to flow downstream in the direction of the propeller rotation axis $\mathbf{u}_t = (0, 0, 1)$ instead of in the flow direction $\mathbf{u}_t = \mathbf{v}$.

A5: Airfoil section properties are calculated in XFOIL for a Reynolds number and Mach number based on the undisturbed air speed, density and mean chord, instead of the local air speed and density in the propeller slipstream and the local chord.

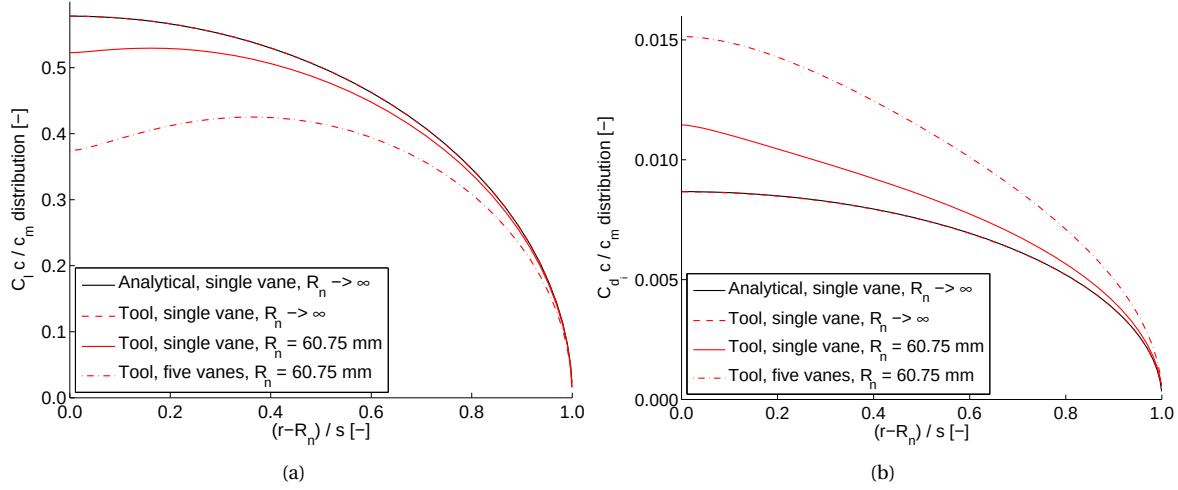


Figure 4.8: Comparison of analytical solution and result of SRV analysis tool of dimensionless (a) lift and (b) induced drag distribution for an elliptical vane mounted on an infinite wall. Also the effect of the nacelle with $R_n = 60.75 \text{ mm}$ and the effect of multiple vanes $N_{SRV} = 5$ is shown. c_m is the mean chord and the aspect ratio is 9.6.

A6: The flow is assumed two-dimensional for each SRV section and no spanwise flow over the SRV is considered.

A7: The loading is assumed quasi-steady, and thus no unsteady loading effects are considered, taking into account the periodicity and phase shift of the loading only by variation of the inflow field for the different vanes.

4.3. DESIGN OPTIMISATION ROUTINE

In order to design swirl recovery vanes for particular propeller inflow fields, an optimisation routine is constructed around the SRV analysis tool. In Section 4.1 it is concluded that in general one can best optimise for maximum swirl recovery vane thrust instead of minimum downstream swirl as the name suggests. This routine uses the gradient-based `fmincon` function from Matlab, which attempts to find a constrained minimum of a non-linear multivariable function starting at an initial estimate [47]. Specifically the SQP algorithm is used. This function is often used for the optimisation of aerodynamic surfaces, for instance for the optimisation of minimum drag wings with constraints on the lift, weight, pitching moment, and stall speed in Ning and Kroo [48]. In general the optimisation can be described as:

$$\min f(\mathbf{x}) \text{ such that } \begin{cases} \mathbf{C}_{in}(\mathbf{x}) \leq 0 \\ \mathbf{C}_{eq}(\mathbf{x}) = 0 \\ \mathbf{lb} \leq \mathbf{x} \leq \mathbf{ub} \end{cases} \quad (4.37)$$

where $f(\mathbf{x})$ is the objective, $\mathbf{C}_{in}(\mathbf{x})$ and $\mathbf{C}_{eq}(\mathbf{x})$ the non-linear inequality and equality constraints and $\mathbf{x}$ the design vector, bounded by a lower and upper bound $\mathbf{lb}$ and $\mathbf{ub}$. In this case, the objective that needs to be minimized is the SRV drag. The design vector consists of 14 variables, of which 3 variables for both $\mathbf{A}_c$ and $\mathbf{A}_\theta$:

$$\mathbf{x} = \left[c_r \quad \lambda \quad \mathbf{A}_c \quad N_{2,c} \quad \theta_r \quad \Theta \quad \mathbf{A}_\theta \quad \frac{t}{c} \quad \frac{cm_{\max}}{c} \quad \frac{x_{cm_{\max}}}{c} \right] \quad (4.38)$$

Inequality constraints are used to avoid the region of SRV stall as predicted by XFOIL. Both a design condition and a constraint condition at a different advance ratio are checked for stall. Furthermore, a minimum propulsive efficiency benefit $\Delta\eta_p$ can be prescribed for the constraint condition. Equality constraints are used whenever no feasible solution is found for a particular design vector, for instance when the airfoil is such that XFOIL does not converge. This routine searches for a local minimum. By varying the initial condition the probability that a global minimum is found increases. Since the goal of this work is to analyse the performance and slipstream characteristics of the combination of a propeller, swirl recovery vanes and a wing in general, and not necessarily to find the best SRV design for a particular case, it is thought that with a few variations of the initial condition a satisfactory design is obtained.

4.4. DESIGN RESULTS

A number of SRV designs are discussed in this section. In order to investigate the effect of different design objectives on the design variables, four different designs are proposed:

Design 1: Design for the cruise condition ($J = 1.60$) with a constraint on stall for the high-thrust condition ($J = 0.95$).

Design 2: Design for the high-thrust condition with a constraint on the cruise condition for zero or more thrust.

Design 3: Design for the cruise condition without constraint condition. For this design find the highest thrust for the high-thrust condition by changing the SRV pitch angle.

Design 4: Design for the high-thrust condition without constraint condition. For this design find the highest thrust for the cruise condition by changing the SRV pitch angle.

Design 1 and 2 represent designs where the swirl recovery vanes have a fixed pitch in flight, while design 3 and 4 represent designs with variable pitch in flight. The choice for cruise and high-thrust condition is based on the performance of the APIAN propeller as discussed in Subsection 2.4.3. For these designs the flow fields at the rake plane from the propeller CFD model of Chapter 2 are used. The flow field at an instance in time varies not only radially but also circumferentially as indicated by Figures 2.9 and 2.11 which show this variation for the axial and tangential velocity components respectively. Thus each vane encounters a different flow field at an instance in time and this variation is taken into account in the optimisation. In other words, the swirl recovery vanes are designed for the flow field at an instance in time, and not for the time-averaged flow field. This approach is thought to be advantageous since in this way the stall constraint is better taken into account: The varying swirl angle in time may result in vane stall which would not have been noticed when using the time-averaged field. This approach assumes a quasi-steady vane loading while in reality the loading on the vanes is periodic with a high-frequency and transient effects on the loading occur.

When interpreting these designs, one should first take a close look at the axial velocity and swirl angle radial distribution in Figures 2.15 and 2.17. In the cruise condition ($J = 1.60$) the axial velocity at the root is zero (no-slip wall) and increases first steeply in the boundary layer of the nacelle and then gradually to a maximum of $1.15V_\infty$ at $r/R_p = 0.85$. Then it steeply decreases to the undisturbed axial velocity. The swirl angle at the root first shows a small decrease in the boundary layer and then increases to a maximum of 7° around $r/R_p = 0.65$. The swirl angle gradually decreases to 0° just outside the blade tip. In the high-thrust condition ($J = 0.95$) the shape of the axial velocity profile is very similar, except with a much higher maximum of $1.80V_\infty$ at a slightly lower radial position of $r/R_p = 0.80$ and a small minimum of $0.92V_\infty$ at the blade tip. The swirl angle increases steeply to a local maximum of 17.5° at $r/R_p = 0.50$, then decreases to a local minimum of 15° at $r/R_p = 0.80$ and then steeply increases to the global maximum slightly higher than 20° at $r/R_p = 0.90$. A very sharp drop in swirl angle occurs at larger radii to 0° just outside the blade tip.

The number of vanes is kept constant at 5 for all designs. The effect of the number of vanes on the propulsive efficiency will be shown for design 1 in the next subsection. An SRV to propeller cropping ratio of 1 is used for all designs, since cropping showed to lower the design propulsive efficiency benefit: When cropping is applied, a part of the tangential kinetic energy of the slipstream is left unused. This will also be shown in the next subsection. Also no quarter chord sweep is applied for simplification and reduction of the design vector. The resulting four designs are discussed in the next subsections.

4.4.1. DESIGN FOR CRUISE WITH CONSTRAINT CONDITION

This design maximises the propulsive efficiency benefit by the addition of SRV for the cruise condition, while preventing SRV stall in the high-thrust condition. In the high thrust condition the average swirl angle is considerably larger, up to 15° near the tip as is shown in Figure 2.17. The instantaneous range of swirl angle even exceeds this. Such a design is therefore not straight forward and this constraint profoundly affects the propulsive efficiency benefit in the cruise condition. Observe Table 4.1 which shows the variation of the propulsive efficiency benefit $\Delta\eta_p$ in the cruise condition with changing stall constraint: The design is optimised with a stall constraint that allows a design up to the angle of attack at maximum lift coefficient α_{max} and a reduction of this maximum angle of 1° through 4° . In the next chapter it will be shown that stall occurs at a smaller angle in the CFD simulation than predicted by XFOIL and no converged solution is found for the design for α_{max} . Eventually the design for $\alpha_{max} - 3^\circ$ is chosen for the propeller-SRV CFD model and this design is shown here.

Table 4.1: Effect of stall constraint on propulsive efficiency increase by the addition of SRV $\Delta\eta_p$.

Stall constraint [Design °]	$\Delta\eta_p$ [%]
α_{max}	0.652
$\alpha_{max} - 1$	0.649
$\alpha_{max} - 2$	0.600
$\alpha_{max} - 3$	0.499
$\alpha_{max} - 4$	0.336

The design properties are given in Table 4.2. While a small propulsive efficiency benefit is found in the cruise condition, the gain in the high-thrust condition is considerably larger, $\Delta\eta_p = 3.49\%$, due to the larger swirl angle of the propeller slipstream. Figure 4.9 gives the chord and pitch distribution and airfoil section and Figure 4.10 shows the airfoil (a) $C_l - \alpha$ and (b) $C_d - \alpha$ curve. The corresponding geometric and effective angle of attack are shown in Figure 4.11 for the (a) design and (b) constraint condition. Both the average over the vanes (solid lines) and the instantaneous angle of attack (symbols) for each vane is plotted.

There are basically three parameters to influence the local thrust force: Chord, pitch angle and airfoil section. Regarding the latter, as was shown in Section 4.1 for a simplified vane, ideally the airfoil section for swirl recovery vanes is cambered such that the lift coefficient at zero angle of attack C_{l_0} is the design point. The airfoil section of this design is highly cambered as expected and also reasonably thin. The small thickness results in a low pressure part of the profile drag, especially important to find a feasible design in cruise. Due to the assumption of a fully turbulent boundary layer, the resulting viscous part of the profile drag is quite high and thus the influence of the profile drag on the optimisation is considerable. $\alpha_{max} = 12.5^\circ$ for this airfoil and since the stall constraint for this design is set at $\alpha_{max} - 3^\circ$ the instantaneous effective angle of attack in the constraint condition should not exceed $\alpha_{max} = 9.5^\circ$.

The chord gradually increases from the root to a maximum around $r/R_p = 0.85$. This maximum is in line with that of the axial velocity distribution of the design cruise condition. The pitch increases, i.e. increasing the angle of attack, up to $r/R_p = 0.6$ and then decreases again, with a small increase again at the tip. This maximum is more in line with that of the swirl angle distribution. The resulting geometric and effective angle of attack in the cruise condition have a maximum corresponding to the maximum in the pitch distribution. At this maximum, the average induced angle of attack is close to 3.5° , which is half of the swirl angle at this point. Since the downwash far downstream is about twice that of the local upwash in front of the vane according to vortex theory, this can be related to complete recovery of the swirl downstream of this point. Also, the effective angle of attack is zero, and thus the theoretical ideal condition is reached at this point that the swirl is completely recovered with a cambered airfoil operating at C_{l_0} .

The remainder of the design is however very much driven by the constraint condition. One can see a large variation in geometric angle of attack, especially close to the tip for the constraint condition due to the strong propeller tip vortices. The effective angle of attack varies less due to the induced field of the all vanes, but varies still considerably. As the design prescribes, the instantaneous effective angle of attack in the constraint condition does not exceed $\alpha_{max} = 9.5^\circ$. In order to reach this constraint the pitch angle near the tip is highly negative to avoid stall where this maximum swirl angle occurs. Also near the root the variation in instantaneous effective angle of attack is quite large and to comply with the stall constraint in this region, a large negative pitch angle is needed. This results in a very small loading for the design cruise condition in this region. Since near the root the effective angle of attack does not reach $\alpha_{max} = 9.5^\circ$ but is close to it, a stricter tolerance on the objective function or smaller minimum steps in the design variables in the optimisation could have resulted in a slightly better SRV root performance.

Table 4.3 gives some insight into the effect of cropping and number of vanes on the SRV performance. The span of swirl recovery vanes can be made smaller to avoid the region of the propeller tip vortices. In this case, an SRV to propeller radius cropping ratio of 0.8 is needed to avoid the steepest velocity gradients. For this cropping ratio, a design has been optimised with the same constraints as the just presented design. The effect of cropping on the propulsive efficiency benefit in the design and constraint condition is very large: A decrease to almost half of the benefit without cropping. An increase in the number of vanes for the given range leads to an increase in propulsive efficiency for the design cruise condition and constraint high-thrust condition. This is in line with Çelik and Güner [13] where it is shown that for a marine application of SRV an increasing gain in propulsive efficiency is found with increasing number of vanes up to 9 vanes. Although for at least up to 9 vanes, more vanes is better, the choice for 5 vanes has been made for a number of reasons: First, preliminary investigations suggested a lesser increase in propulsive efficiency. Second, the

SRV designed for the APIAN-INF wind tunnel test consist of 5 vanes and is cropped, and thus keeping the number of vanes equal allows for a better qualitative investigation into the effect of cropping. This design consists of five vanes and not the same as the number of propeller blades to avoid noise by aerodynamic interference. Third, mesh size and thus computational cost increases with the number of vanes for 360° domain simulations. In retrospect a design of more than 5 vanes might have been a better choice.

Table 4.2: Overview of design for cruise with constraint condition SRV properties.

Variable	Value
Number of vanes N_{SRV}	5
Span s	189.25 mm
SRV to propeller cropping ratio	1.0
Quarter chord sweep $\Lambda_{c/4}$	0°
Root chord c_r	30.54 mm
Taper ratio λ	1.30
Chord shape function coefficients $\mathbf{A}_c$	[0.02, 1.30, 1.29]
Chord tip class function parameter $N_{2,c}$	0.50
Root pitch θ_r	-5.34°
Twist Θ	-0.18°
Pitch shape function coefficients $\mathbf{A}_\theta$	[-0.002, -0.35, 0.08]
Airfoil thickness $\frac{t}{c}$	8.52 %c
Airfoil maximum camber $\frac{cm_{\max}}{c}$	4.20 %c
Airfoil location of maximum camber $\frac{x_{cm_{\max}}}{c}$	40.68 %c
$\Delta\eta_p$ for design condition $J = 1.60$	0.50%
$\Delta\eta_p$ for constraint condition $J = 0.95$	3.49%

Table 4.3: Design for cruise with constraint condition propulsive efficiency benefit and effect of cropping and number of swirl recovery vanes.

N_{SRV} [-]	Cropping ratio [-]	Design $\Delta\eta_p$ [%]	Constraint $\Delta\eta_p$ [%]
5	1	0.50	3.49
5	0.8	0.28	1.83
7	1	0.52	4.18
9	1	0.60	4.53

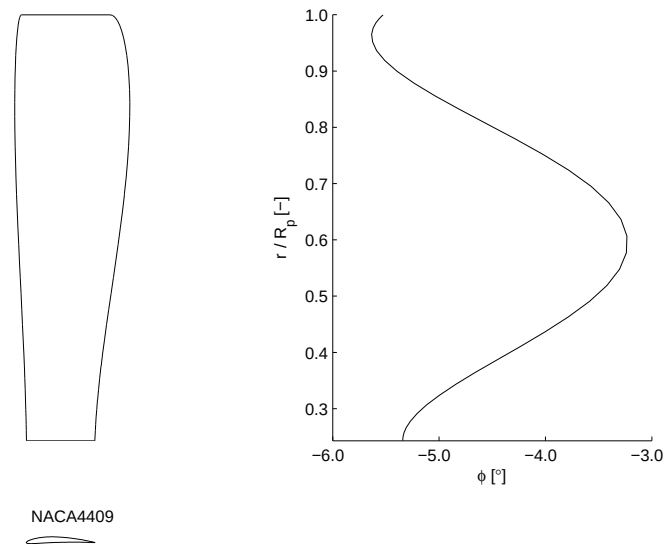


Figure 4.9: Design for cruise SRV with constraint condition chord and pitch distribution and airfoil section.

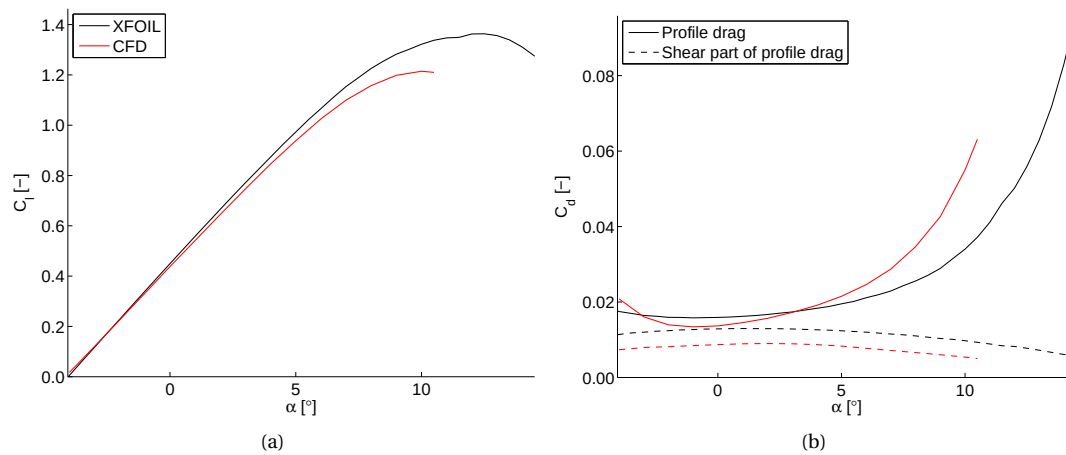


Figure 4.10: Design for cruise with constraint condition SRV airfoil (a) C_l - α curve and (b) C_d - α curve as predicted by XFOIL and resulting from CFD simulation of the same section at undisturbed conditions.

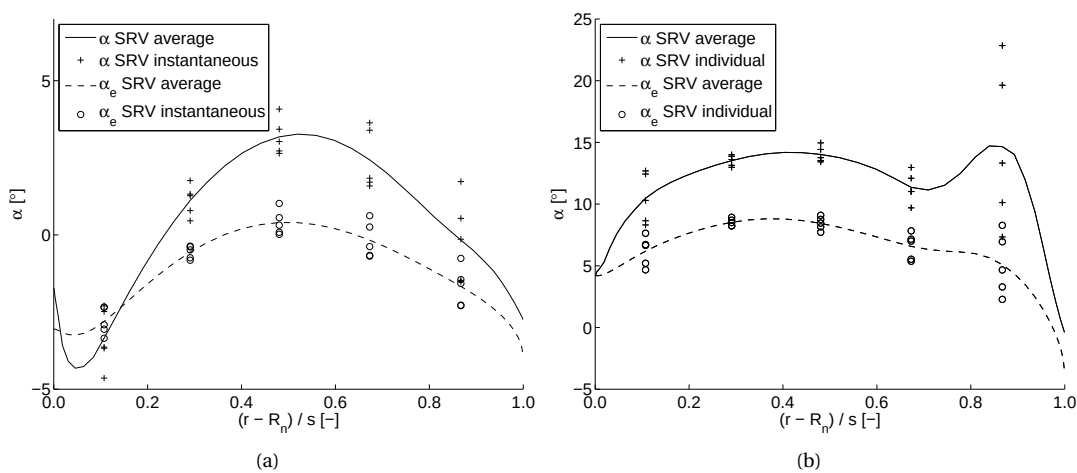


Figure 4.11: Average and individual geometric and effective angle of attack distribution along the span of the SRV for (a) cruise design condition and (b) high-thrust constraint condition.

4.4.2. DESIGN FOR HIGH-THRUST WITH CONSTRAINT CONDITION

This design maximises the increase in thrust by the swirl recovery vanes for a low advance ratio ($J = 0.95$) when the propeller thrust is high. The constraint condition is such that in the cruise no loss of propulsive efficiency results from the swirl recovery vanes. The same results are given as for the previous design in Figures 4.12 through 4.14 and Table 4.4. Indeed, the propulsive efficiency benefit for this design is slightly higher ($\Delta\eta_p = 3.84\%$) than for the previous design, at the cost of zero gain in the constraint cruise condition. The chord gradually increases from the root to a maximum around $r/R_p = 0.7$. For a large part of the span the chord is twice that of the just presented design for cruise. The pitch curve is of opposite shape compared to the design for cruise and is turned into the swirl angle, decreasing the angle of attack, with a minimum pitch slightly outward of $r/R_p = 0.7$. The airfoil section is substantially thinner and has more camber than the previous design, reducing the angle of attack at $C_{l_{max}}$ to $\alpha_{max} = 8^\circ$ and increasing C_{l_0} . The corresponding profile drag has a minimum at $\alpha = 2^\circ$ and remains low for a region of a few degrees around this minimum. The constraint on stall is at α_{max} but no reduction of this angle is needed like in the design for cruise due to the low design effective angle of attack: The average effective angle of attack distribution for both the design high-thrust and constraint cruise condition is quite constant around $\alpha_e = 4^\circ$ and $\alpha_e = -2^\circ$ respectively. Due to the different pitch distribution, this effective angle of attack in both conditions is lower than for the previous design, resulting in a zero SRV performance gain in the constraint cruise case and better SRV performance in the design high-thrust condition due to an improved $\frac{C_l}{C_d}$ ratio.

For the design condition, the induced angle of attack is about 8° , recovering most but not all of the swirl downstream: This is as expected from the analysis of the simplified vane in Section 4.1, since the camber is not such that the design operates at C_{l_0} .

Table 4.4: Overview of design for high-thrust with constraint condition SRV properties.

Variable	Value
Number of vanes N_{SRV}	5
Span s	189.25 mm
SRV to propeller cropping ratio	1.0
Quarter chord sweep $\Lambda_{c/4}$	0°
Root chord c_r	60.37 mm
Taper ratio λ	0.80
Chord shape function coefficients $\mathbf{A}_c$	[0.26, 1.37, 0.84]
Chord tip class function parameter $N_{2,c}$	0.55
Root pitch θ_r	-1.92°
Twist Θ	-0.18°
Pitch shape function coefficients $\mathbf{A}_\theta$	[0.19, 0.20, 0.48]
Airfoil thickness $\frac{t}{c}$	6.53 %c
Airfoil maximum camber $\frac{cm_{max}}{c}$	4.20 %c
Airfoil location of maximum camber $\frac{x_{cm_{max}}}{c}$	69.58 %c
$\Delta\eta_p$ for design condition $J = 0.95$	3.84%
$\Delta\eta_p$ for constraint condition $J = 1.60$	0.00%

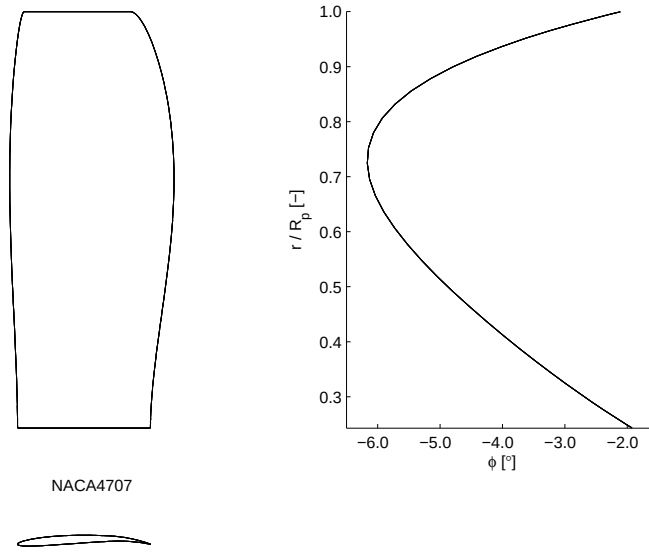


Figure 4.12: Design for high-thrust with constraint condition SRV chord and pitch distribution and airfoil section.

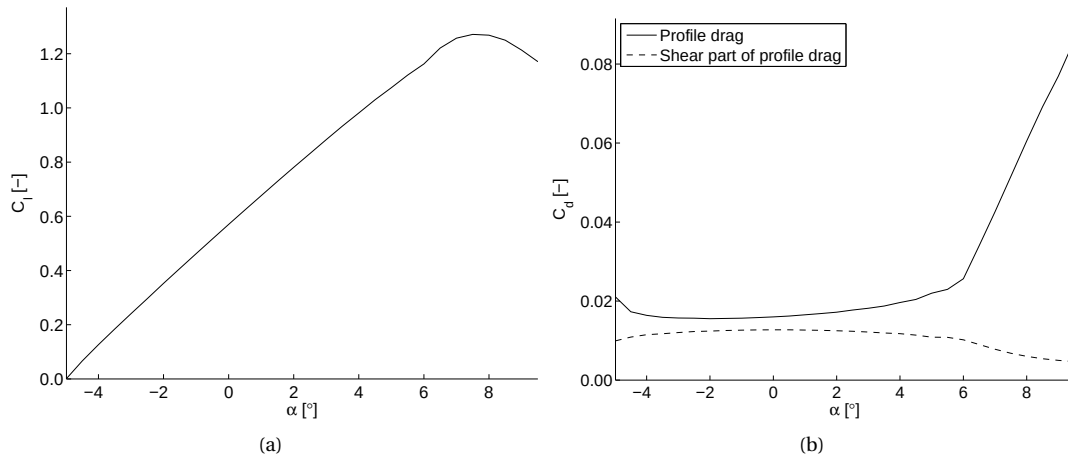


Figure 4.13: Design for high-thrust with constraint condition SRV airfoil (a) $C_l - \alpha$ curve and (b) $C_d - \alpha$ curve.

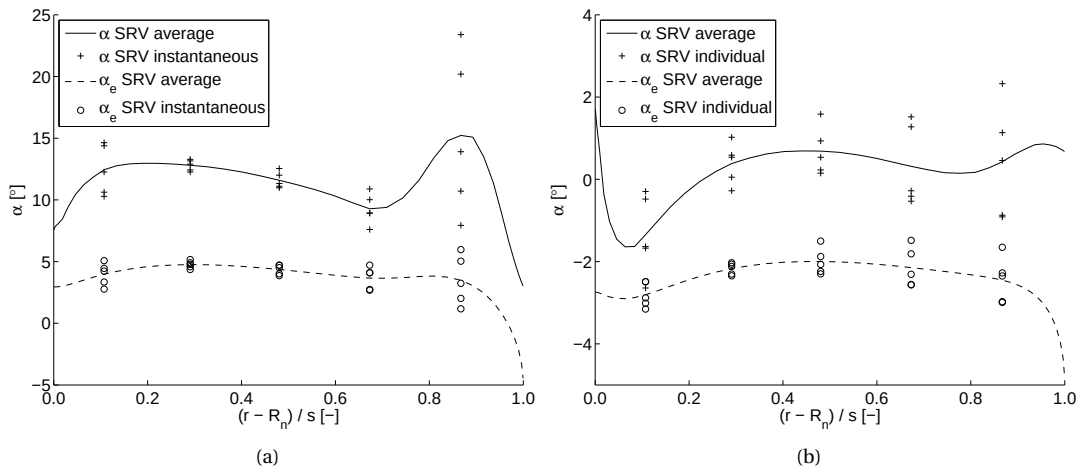


Figure 4.14: Average and individual geometric and effective angle of attack distribution along the span of the SRV for (a) high-thrust design condition and (b) cruise constraint condition.

4.4.3. DESIGN FOR CRUISE WITHOUT CONSTRAINT CONDITION

These swirl recovery vanes are optimised for the cruise condition and by means of variable pitch are turned to maximise the performance and avoid stall in the high-thrust condition. The same results are given as for the previous designs in Figures 4.15 through 4.17 and Table 4.5. Table 4.6 gives the propulsive efficiency increase and corresponding root pitch angles θ_r for the design and constrain condition. The propulsive efficiency in cruise more than doubles using this strategy. However, since the constraint condition is not taken into account anymore during the optimisation, the propulsive efficiency in the high-thrust condition reduces by a factor two compared to constrained cruise design. The shape of the chord distribution is very similar to the constrained cruise design but the actual chord is only half. The shape of the pitch distribution is also similar but the maximum is more than 5° higher. So in general, chord has been traded for pitch which was not possible for the constrained cruise design due to the stall constraint for the high-thrust condition. The airfoil section is also thinner, while the camber is similar. The low thickness leads to a low angle of attack at $C_{l_{max}}$ of $\alpha_{max} = 9^\circ$ with a region of low profile drag similar to the design for high-thrust. This is also permitted due to the omission of the constraint condition. The thin airfoil section and small chord may be difficult for the structural design, especially considering the loading in the high-thrust condition. Therefore this design is not considered a feasible design. For a variable pitch design it is recommended to use a certain weighting factor to take both the loading in cruise and in the high-thrust condition into account, to arrive at a design with better high-thrust performance and of larger chord, with a slightly lower cruise performance.

Table 4.5: Overview of design for cruise without constraint condition SRV properties.

Variable	Value
Number of vanes N_{SRV}	5
Span s	189.25 mm
SRV to propeller cropping ratio	1.0
Quarter chord sweep $\Lambda_{c/4}$	0°
Root chord c_r	15.84 mm
Taper ratio λ	1.07
Chord shape function coefficients $\mathbf{A}_c$	[0.04, 1.25, 1.12]
Chord tip class function parameter $N_{2,c}$	0.63
Root pitch θ_r	-4.78°
Twist Θ	-0.11°
Pitch shape function coefficients $\mathbf{A}_\theta$	[-0.35, -0.75, 0.17]
Airfoil thickness $\frac{t}{c}$	6.72 %c
Airfoil maximum camber $\frac{cm_{max}}{c}$	4.35 %c
Airfoil location of maximum camber $\frac{x_{cm_{max}}}{c}$	50.71 %c

Table 4.6: Overview of design for cruise without constraint condition SRV propulsive efficiency benefit.

J [-]	θ_r [$^\circ$]	$\Delta\eta_p$ [%]
1.6	-4.78	1.14
0.95	-15.55	1.77

4.4.4. DESIGN FOR HIGH-THRUST WITHOUT CONSTRAINT CONDITION

The results for this design are omitted. It happens to be that the constraint on zero or better performance in the cruise is not limiting the design for the high-thrust condition. The design without constraint is almost exactly the same. When the root pitch angle is changed to maximise the efficiency increase in cruise, it is found that the current angle is almost the optimum for the cruise as well and no significant gain in propulsive efficiency can be reached with this design in the cruise.

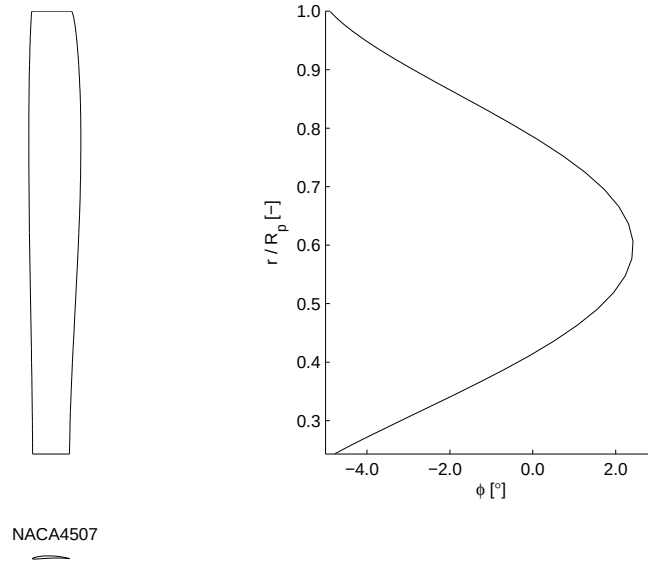


Figure 4.15: Design for cruise without constraint SRV chord and pitch distribution and airfoil section.

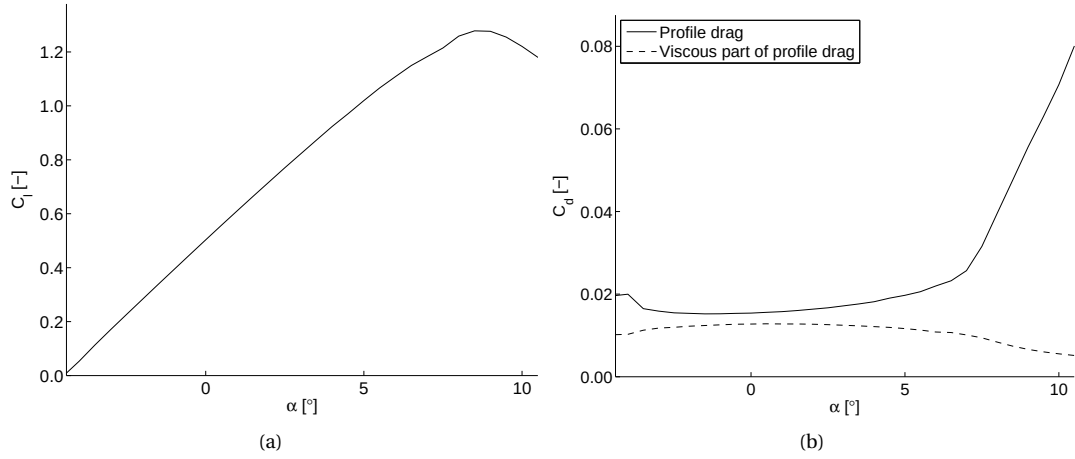


Figure 4.16: Design for cruise without constraint SRV airfoil (a) C_l - α curve and (b) C_d - α curve.

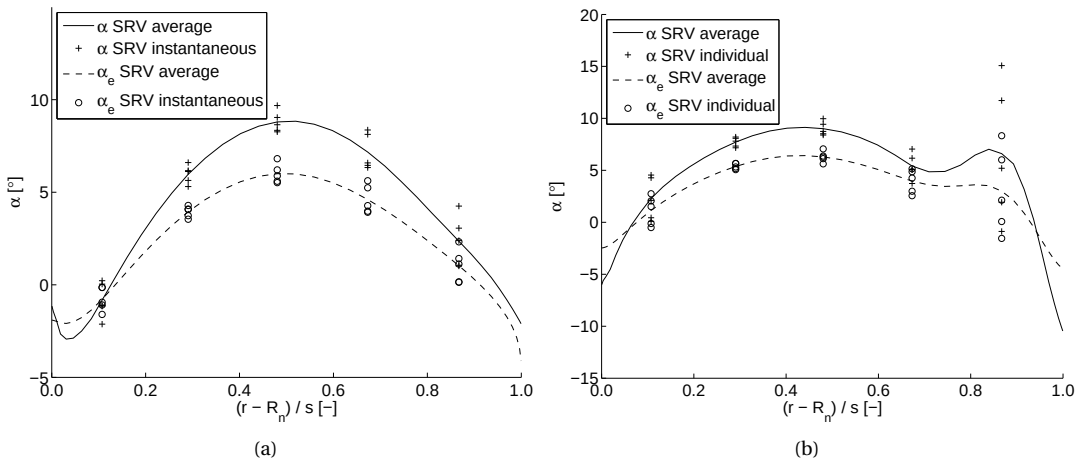


Figure 4.17: Average and individual geometric and effective angle of attack distribution along the span of the SRV for (a) cruise design condition and (b) high-thrust condition.

5

PROPELLER-SRV CFD MODEL

In the previous chapter a swirl recovery vane analysis tool and optimisation routine have been introduced and designs with various design objectives have been presented. In this chapter the swirl recovery vanes of design 1 and 2 are used for the propeller-SRV CFD model. The propeller-SRV CFD model consist of the APIAN propeller and swirl recovery vanes mounted on an isolated nacelle. While design 1 is optimised for the cruise condition and stall is prevented in the high-thrust condition, design 2 is optimised for the high-thrust condition and has a constraint on zero or positive SRV thrust in the cruise condition. The main goals of this chapter are to validate the SRV analysis tool, to compare the performance and slipstream of a propeller with SRV to the same configuration without SRV from Chapter 2 and to provide data for comparison when a wing is introduced in the next chapter. First the geometry of the domain and the mesh size is shortly discussed in Section 5.1 and 5.2 and then the results are presented in Section 5.3. The solver settings are the same as presented in Chapter 3.

5.1. GEOMETRY

The geometry consists of the APIAN propeller introduced in Chapter 2 with corresponding nacelle and the swirl recovery vanes of design 1 and 2 as presented in Section 4.4. The simulations in this chapter consist of simulations by means of profile transformation between the rotating region and stationary region as well as 360° domain simulations. The structure of the domain is the same as in Chapter 3 except for the nacelle radius, SRV design and downstream position of the SRV, as shown in Figure 5.1. The downstream position corresponds to the rake plane as shown in Figure 2.6, used to extract the flow quantities for the design of the SRV.

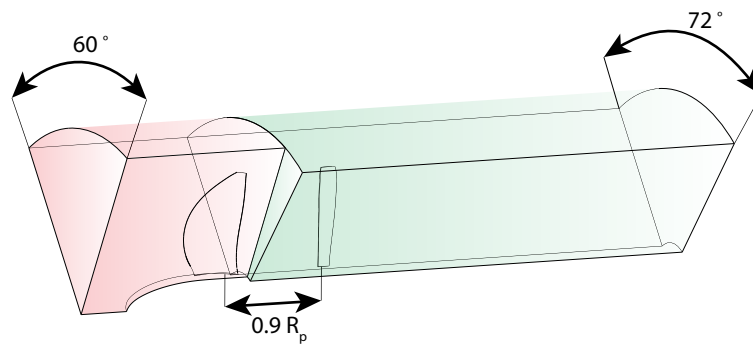


Figure 5.1: Detailed view of the rotating and wake region of the Propeller-SRV CFD Model for SRV design 1.

5.2. MESH

The mesh for these simulations is very similar to the one presented in Chapter 3 for the APIAN-INF CFD model and therefore no mesh dependency study is performed. The SRV wall refinement is very similar but

adjusted slightly to better match the design and is presented in Table 5.1. The resulting number of nodes for the domain of a single blade and vane are given as well and these should be multiplied by the blade and vane count for the 360° domain simulations.

Table 5.1: Overview of mesh size and refinement dimensions for the propeller-SRV CFD model.

SRV design	# nodes stationary region	# nodes rotating region	SRV wall size [mm]	l.e. size [mm]	t.e. and tip size [mm]
Design 1	1,136,025	762,551	1.50	0.25	0.40
Design 2	1,288,423	762,551	1.50	0.25	0.40

5.3. RESULTS

Simulations by means of profile transformation of SRV designs 1 and 2 are discussed in this section. Design 1 is optimised for maximum increase in propulsive efficiency in the cruise condition ($J = 1.60$) with a constraint on stall for the high-thrust condition ($J = 0.95$). Design 2 is optimised for the high-thrust condition with a constraint on the cruise condition for zero or positive SRV thrust. Since the design for cruise is chosen for further analysis in installed condition, 360° domain simulations are performed for this design in order to describe its slipstream in more detail. These simulations also give a good estimation of the error in performance quantities from the profile transformation. First the effect of the SRV on the propeller performance will be investigated: The SRV analysis tool neglects this effect when estimating the propulsive efficiency increase by the addition of SRV and therefore it is important to see the validity of assumption. This is followed by a discussion on the SRV performance and loading for both designs, which provides a means to validate the SRV analysis tool. At last, a description of the slipstream for the design for cruise is given alongside results without SRV for comparison.

Table 5.2: Time-average performance quantities of propeller-SRV CFD model designed for cruise, relative to the performance without SRV from Table 2.7 as defined in Equations (3.1) to (3.5).

J [-]	Interface type	ΔC_{T_p} [%]	ΔC_T [%]	ΔC_P [%]	$\Delta \eta_{p_p}$ [%]	$\Delta \eta_p$ [%]
0.95	profile transformation	+0.65	+6.03	+0.75	-0.05	+2.62
1.00	profile transformation	+0.57	+5.68	+0.57	-0.04	+2.64
1.30	profile transformation	+0.76	+3.96	+0.83	-0.04	+2.07
1.60	profile transformation	+0.35	+1.01	+0.29	+0.06	+0.55
1.80	profile transformation	+0.68	-2.40	+0.57	+0.11	-2.27
0.95	360° domain	+0.04	+5.32	+0.19	-0.08	+2.56
1.30	360° domain	+0.32	+3.52	+0.45	-0.09	+2.04
1.60	360° domain	-0.06	+0.55	+0.04	-0.10	+0.39

Table 5.3: Time-average propeller performance quantities of propeller-SRV CFD model designed for high-thrust, relative to the performance without SRV from Table 2.7 as defined in Equations (3.1) to (3.5).

J [-]	Interface type	ΔC_{T_p} [%]	ΔC_T [%]	ΔC_P [%]	$\Delta \eta_{p_p}$ [%]	$\Delta \eta_p$ [%]
0.95	profile transformation	+0.53	+6.47	+0.69	+0.11	+3.07
1.00	profile transformation	+0.74	+6.52	+0.72	-0.02	+3.00
1.30	profile transformation	+0.98	+4.39	+1.04	-0.04	+2.21
1.60	profile transformation	-0.49	-0.32	-0.58	+0.09	+0.20
1.80	profile transformation	+1.36	-2.51	+0.95	+0.40	-2.63

5.3.1. EFFECT OF SRV ON PROPELLER PERFORMANCE

For the designs for cruise and high-thrust, the propeller and SRV performance quantities are given in Tables 5.2 and 5.3 respectively relative to the propeller CFD model without SRV from Chapter 2. The performance quantities are given as defined in Equations (3.1) through (3.5). Considering the effect of the SRV on the propeller performance, the difference in the propeller thrust coefficient ΔC_{T_p} and power coefficient ΔC_P are important, as well as the resulting difference in propeller propulsive efficiency $\Delta \eta_{p_p}$. One should distinguish

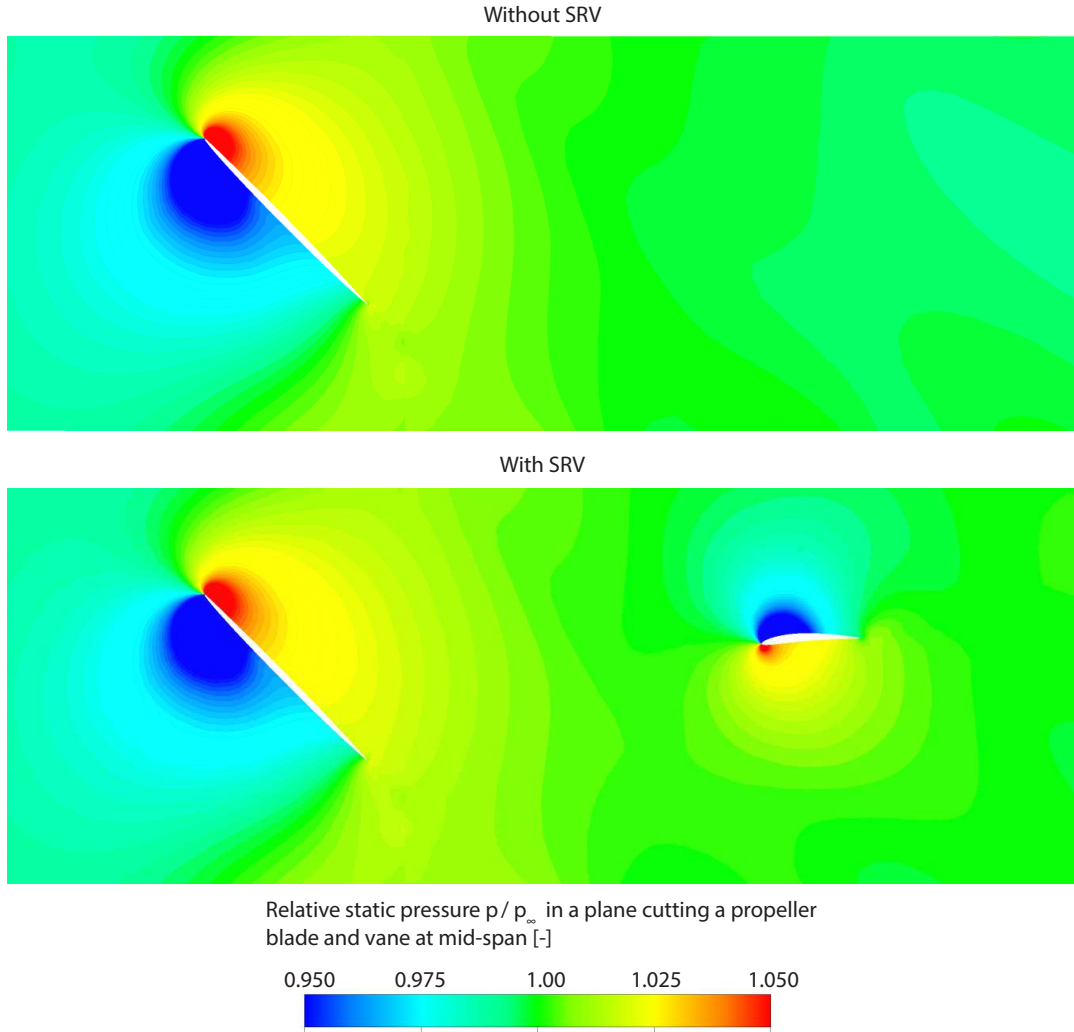


Figure 5.2: Relative static pressure without and with SRV in a plane cutting a blade of the propeller and a swirl recovery vane at an instance of the propeller blade passage at $J = 0.95$.

between two effects: The physical effect of the SRV and the effect of the modelling of SRV by means of the profile transformation. The first effect is assumed to be zero in the SRV analysis tool (Assumption **A1** in Subsection 4.2.4).

For the 360° domain simulations performed for the design for cruise, propeller performance differences can only be the result of the physical effect of the SRV on the propeller. These differences are very small and are likely the result of changes in the pressure field by the SRV: Observe Figure 5.2 which shows a comparison of the instantaneous pressure field at a plane at mid-span without and with SRV designed for cruise at $J = 0.95$. The static pressure behind the propeller slightly increases by the addition of the SRV, and this leads to an increase of static pressure at the pressure side of the propeller blade, increasing the loading. The resulting change in the propeller propulsive efficiency is a slight decrease in the order of 0.1 %.

The effect of the modelling of SRV by means of the profile transformation is in general larger. It leads to an increase in C_{Tp} and C_P , except for $J = 1.60$ for the design for high-thrust. This effect probably occurs as a result of the inherently wrong propeller vortex structure as discussed in Section 3.4.2 with corresponding wrong induced velocity field: Only five propeller tip and root vortices are present in the slipstream of the propeller. Furthermore, the changes in the pressure field due to the SRV behind the propeller are altered by the profile transformation and may also lead to deviations. Except at $J = 1.80$ for the design for high-thrust, the changes in the propeller propulsive efficiency by profile transformation are small, in the order of 0.1 %. At $J = 1.80$ for the design for high-thrust, the propeller propulsive efficiency increases by 0.4 % and this should be taken into account when interpreting the results in the next subsection.

5.3.2. SRV PERFORMANCE AND LOADING

The main quantity indicating the performance of SRV is the increase in propulsive efficiency by the addition of SRV $\Delta\eta_p$, which is given in Tables 5.2 and 5.3 for the two designs and defined in Equation (3.5). In Figure 5.3 $\Delta\eta_p$ is shown for (a) the design for cruise and (b) for the design for high-thrust. Both the prediction by the SRV analysis tool and the simulation results are plotted, from both the profile transformation and 360 ° domain simulations. In general, the prediction by the tool for $J = 1.30 - 1.60$ agrees well for both designs while deviations are present at lower and higher advance ratios. Despite the deviations, the propulsive performance benefit from the SRV is considerable for a large range of advance ratios for both designs.

As was already indicated in Subsection 3.4.2, the effect of the profile transformation on the SRV thrust is very small: The difference in combined propeller and SRV thrust coefficient ΔC_T between the 360 ° domain and profile transformation simulations is almost only the result of the difference in propeller thrust coefficient ΔC_{T_p} , and thus not the result of a difference in SRV thrust. The difference in $\Delta\eta_p$ between the 360 ° domain and profile transformation simulations results is only the result differences in the propeller thrust and power and is small. So, although the profile transformation results in slightly different propeller loading and power, these simulations give a good indication of the propulsive performance benefit of SRV.

$\Delta\eta_p$ as predicted by the tool assumes a constant propeller performance, that of the propeller without SRV. As seen in the previous subsection, the addition of SRV results in changes in propeller performance which influences $\Delta\eta_p$ directly from its definition and indirectly by altering the propeller slipstream and resulting SRV loading. It was concluded that these deviations were in the order of 0.1 % in the propeller propulsive efficiency except for the design for high-thrust at $J = 1.80$ which was 0.4 %. This latter deviation completely explains the difference between the prediction by the tool and the CFD result for the design for high-thrust at $J = 1.80$, and thus the prediction of the SRV performance agrees well for that point as well. In order to better investigate the differences between the tool and CFD simulation, the first direct influence can be removed by for instance only looking at the SRV thrust coefficient: Figure 5.4 shows the SRV thrust coefficient $C_{T_{SRV}}$ for both designs defined as:

$$C_{T_{SRV}} = \frac{T_{SRV}}{\rho_\infty n^2 D_p^4} \quad (5.1)$$

where T_{SRV} is the SRV thrust. In order to enhance the insight into the differences between the tool and CFD, for $J = 1.60$ and $J = 0.95$ the vane thrust spanwise distributions are plotted in Figures 5.6 and 5.7 respectively for the design for cruise and in Figures 5.8 and 5.9 respectively for the design for high-thrust. The thrust distribution is split in a pressure and shear force part. Both the average over the vanes at an instance in time as well as the individual loading of each vane is plotted alongside the prediction by the SRV analysis tool. The corresponding normal force distribution can be found in Figure 5.10 for the design for cruise and in Figure 5.11 for the design for high-thrust. At last, the wall shear lines on a vane are plotted for the whole range of advance ratios in Figure 5.12 for the design for cruise and in Figure 5.13 for the design for high-thrust. These results are discussed in the following paragraphs for each advance ratio.

J=1.60 / Cruise The predictions of $\Delta\eta_p$ in Figure 5.3 and $C_{T_{SRV}}$ in Figure 5.4 by the SRV analysis tool agree well with the CFD result for both SRV designs. In the spanwise thrust distribution in Figure 5.6 for the design for cruise a small overprediction by the tool of the pressure part of the average thrust is noticeable. The shear part of the average thrust is negative, so resulting in drag, and is considerably overpredicted by almost a factor two. Similar differences in the thrust distribution can be seen in Figure 5.8 for the design for high-thrust. These differences are fully explained by a difference in airfoil section properties as predicted by XFOIL and occurring in the CFD simulation. To show the effect of these different properties, the lift and drag coefficient of the airfoil root section for cruise have been calculated up to the maximum lift coefficient $C_{l_{max}}$ in CFD simulations with the same inlet conditions, mesh refinement and general CFD settings. Figure 4.10 in the previous chapter gives the $C_l - \alpha$ and $C_d - \alpha$ curve as predicted by XFOIL and occurring in the simulations: The lift curve slope is slightly overpredicted by XFOIL and $C_{l_{max}}$ as well as $\alpha_{C_{l_{max}}}$ are considerably overpredicted. Large differences in the drag coefficient are also present. Unreliable prediction of stall is not uncommon, as described by Rumsey and Ying [49] which presents a review of the prediction of high-lift by CFD and shows that under as well as overpredictions occur in maximum lift. Mesh issues tend to still remain very important and a far field mesh extent of 50 chords is recommended to accurately predict drag. Due to the presence of many aerodynamic surfaces, six propeller blades, five swirl recovery vanes, the nacelle and spinner and hub, the mesh size is very large, and possibly a compromise is made on the mesh refinement of the vanes that does affect maximum lift.

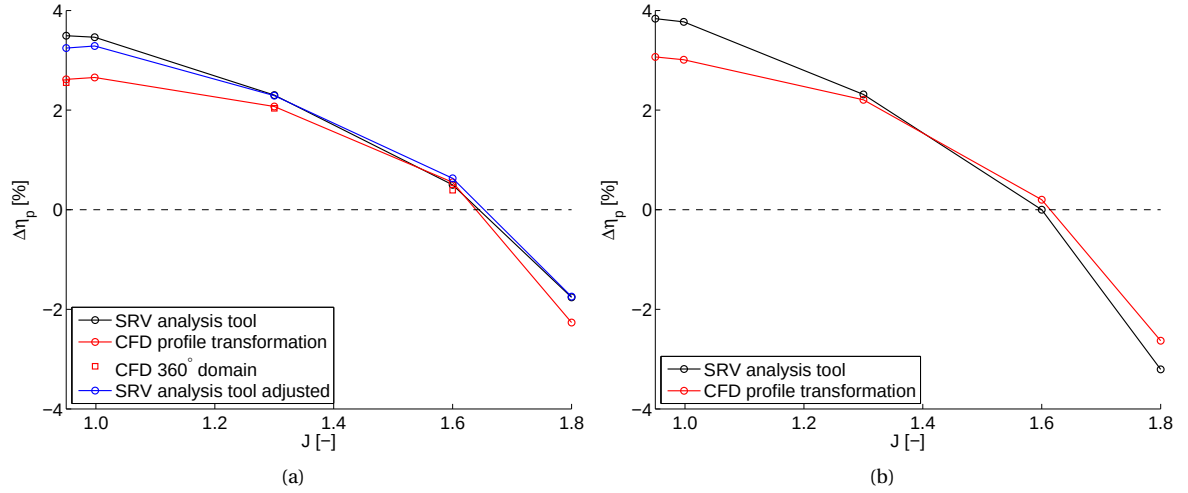


Figure 5.3: Comparison of SRV analysis tool and CFD of the increase in propulsive efficiency by the addition of SRV $\Delta\eta_p$ for (a) the design for cruise and (b) the design for high-thrust.

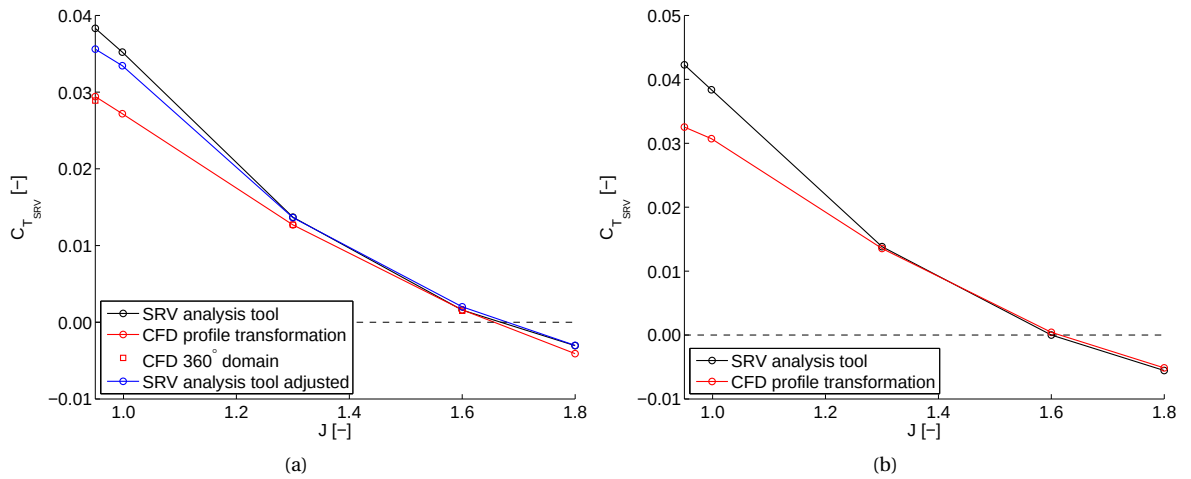


Figure 5.4: Comparison of SRV analysis tool and CFD of the SRV thrust coefficient $C_{T_{SRV}}$ for (a) the design for cruise and (b) the design for high-thrust.

When the CFD section properties are used in the SRV analysis tool, differences in SRV performance occur. These adjusted tool results are also plotted for the design for cruise (blue lines). The pressure and shear part of the average thrust distribution of the adjusted tool completely coincides with the CFD results. It is likely that similar agreement would be found for the design for high-thrust with adjusted section properties. One slight difference still occurs: For the design for cruise, near the root the pressure part of the average thrust is negative, resulting in net drag. This is the result of a small region of separated flow near the leading edge on the pressure side of the SRV, visible in the wall shear lines in Figure 5.12. At this location a low pressure results from the high flow velocity just outside of the separated region. The effect on the SRV thrust coefficient is small.

When observing the thrust-distribution of the individual vanes as indicated by the symbols, a wider spread in the pressure part is noticeable in the prediction by the tool compared to the CFD results. This is most likely the result of the assumption of quasi-steady flow: The loading on a vane is periodic with a frequency of the propeller rotational speed times the number of propeller blades. For $J = 1.60$ this is equal to 85 Hz and for lower advance ratios even higher. In the tool the loading is assumed quasi-steady, meaning that no transient effect occur in the loading of the vanes (Assumption A7 in Subsection 4.2.4). In Saxena et al. [50] the effect of sinusoidally oscillating velocity magnitude over a

stationary NACA0012 airfoil is tested at angles of attack close to stall in steady flow for $Re = 250,000$. Below stall, the force coefficient are found to be quasi-steady for a low frequency of 2.22 Hz and high-frequency of 9.66 Hz at an amplitude of 18% of the freestream velocity. The frequency of the inflow to the vanes is however much higher and not only the velocity magnitude but also the swirl angle changes periodically. Apparently, the periodic loading is damped with respect to the quasi-steady case and this assumption of quasi-steady loading may not be valid. Since the average vane loading agrees with the adjusted SRV analysis tool, the only result of this assumption is a conservative design with respect to stall in the constraint condition, because peaks in the loading are estimated too high.

J=1.80 For the design for cruise a region of flow separation occurs at $J = 1.60$ as shown in Figure 5.12. This separated region expands all the way to the trailing edge at $J = 1.80$ by the further reduction of swirl angle, resulting in a decrease in SRV thrust coefficient $C_{T_{SRV}}$ in Figure 5.4: Since the tool does not predict this root stall, it overpredicts the SRV thrust coefficient. For the design for high-thrust, no such root stall on the pressure side occurs and thus the SRV thrust coefficient is predicted well.

J=1.30 SRV thrust coefficient is predicted well for this condition. Only a slight overprediction for the design for cruise can be noticed. No regions of separation are present for both designs and considerable propulsive efficiency benefits are found. On the suction side, a radial flow component is visible outwards near the root and inwards near the tip, most likely towards the point of highest loading where the trailing vorticity switches sign. The opposite occurs on the pressure side as expected.

J=1.00 $\Delta\eta_p$ in Figure 5.3 and $C_{T_{SRV}}$ in Figure 5.4 are overpredicted by the SRV analysis tool to a large extent. When the airfoil section properties from the CFD simulations instead of from XFOIL are used for the design for cruise, the overprediction is less but still considerable. A small region of separated flow occurs at the trailing edge on the suction side for the design for cruise but not for the design for high-thrust, as can be seen in Figures 5.12 and 5.13 respectively. Despite this, the resulting propulsive efficiency benefit for both designs is large.

J=0.95 / High-thrust The same overprediction is present as for $J = 1.00$. This difference is also clearly visible in the pressure part of the average thrust distribution in Figures 5.7 and 5.9 for the design for cruise and high-thrust respectively. The region of separated flow at the trailing edge on the suction side for the design for cruise is slightly larger. However, since the design for high-thrust does not have a region of separation but $C_{T_{SRV}}$ is still overpredicted, other phenomena cause this overprediction. One of the phenomena is the difference in airfoil section properties as indicated for $J = 1.00$ already: Due to the lower lift curve slope and earlier deviation from the linear region, the loading on the swirl recovery vanes is lower and the resulting induced velocity field by all vanes results in a lower induced angle of attack. This reduction of induced angle of attack increases the effective angle of attack. Together with the reduced $\alpha_{C_{L_{max}}}$, a region of separation occurs for lower angles of attack or higher advance ratios than predicted with the airfoil section properties of XFOIL. Therefore it was necessary to optimise the design for cruise taking a 3° margin from stall into account as constraint in order to get a converged simulation.

For the phenomena that may explain the overprediction of the SRV thrust coefficient, one should look at the assumptions made in the SRV analysis tool:

- First of all, static pressure gradients in the propeller slipstream are not taken into account and may result in different SRV loading (Assumption A2 in Subsection 4.2.4). In Figure 5.2 one can see a small static pressure gradient at the location of the SRV in the result without SRV. Considering an axial distance of the average chord length, the average static pressure difference in axial direction between the leading and trailing edge is 114 Pa at $J = 0.95$. An estimation of the error can be made by multiplying this static pressure difference with the axial projected area of the SRV, which is $3.13 \cdot 10^{-3}\text{ m}^2$. The corresponding increase in $C_{T_{SRV}}$ is $2.31 \cdot 10^{-4}$ and only explains 3% of the difference between the adjusted prediction and CFD result.
- Lifting line theory in general and the non-linear lifting line theory of Phillips and Snyder [45] specific has a limit on the aspect ratio: Predictions below $\mathcal{R} = 4$ according to Phillips and Snyder [45] and below $\mathcal{R} = 3$ according to Prandtl [40] simply can not be represented by loading on a lifting line and the chordwise distribution of lift should be considered (Assumption A3 in Subsection 4.2.4). One of the effects of a reduction of aspect ratio is a reduction of wing lift curve slope and

reducing the aspect ratio below 3 results in an even greater reduction of lift curve slope than predicted. In lifting line theory, when one considers the aspect ratio of a half-wing on a symmetry plane, one considers the full span, including the half span of the imaginary half-wing. In the case of swirl recovery vanes mirroring takes place in the cylindrical nacelle surface and the imaginary vortex distribution is of greatly smaller length in spanwise direction than the actual vane. It is considered plausible to take the sum of the vane span and imaginary spanwise length as the distance to consider for the calculation of the aspect ratio. Then aspect ratios of $\mathcal{R} = 5.7$ and $\mathcal{R} = 3.6$ are found for the design for cruise and high-thrust respectively. Both aspect ratios do not exceed the limit as estimated by Prandtl [40]. However, Jones [51] introduces a correction of the lifting line theory for the effect of chord as and shows that such correction leads to a reduction of wing lift curve slope of about 4 % for $\mathcal{R} = 5.7$ and even 8 % for $\mathcal{R} = 3.6$ for an elliptical wing. Since in the high-thrust condition the SRV is highly loaded and thus the vane lift coefficient is large, the effect of a reduction of lift curve slope on the loading is also large. Furthermore, the loading of the vanes is much more outboard than that of an elliptical wing, and since especially near the tip a deviation from 2-D airfoil theory is present, this effect may be larger for SRV. Thus this may explain a large part of the overprediction of SRV loading.

- The trailing vortex filaments are assumed to flow downstream in axial direction $\mathbf{u}_t = (0, 0, 1)$ instead of in the flow direction $\mathbf{u}_t = \mathbf{v}$ (Assumption A4 in Subsection 4.2.4). This assumption simplifies the problem since the vortex filaments are straight instead of helical and their direction does not depend on the solution of the induced velocity field. This assumption is correct when the swirl velocity component is fully removed downstream of the SRV. In the next subsection it will be shown that for $J = 0.95$ the average swirl angle is reduced from $\phi = 17^\circ$ upstream to $\phi_d = 9^\circ$ downstream and thus the assumption cannot be fully justified. In Figure 5.5 this situation is sketched: When assuming that $\mathbf{u}_t = (0, 0, 1)$ as done in the SRV analysis tool, the downwash to be considered for the change in angle of attack is equal to $w = V_i \cos(\phi)$ where V_i is the velocity induced at the quarter chord line where the lifting line is located. When a vortex filament in flow direction is considered $\mathbf{u}_t = \mathbf{v}$ which is more representing reality, then $w = V_i \cos(\phi - \phi_d)$. When the vortex strength is equal and thus the induced velocity magnitude is equal, a decrease in downwash of 3.6 % is found by this assumption. A too low induced angle of attack and thus a too high effective angle of attack may explain an overprediction of the SRV loading. However, a component of the induced velocity is also in swirl direction, and this component is more negative due to this assumption which reduces loading. For a single vane of the design for cruise $\mathbf{u}_t$ has been changed to $\mathbf{u}_t = \mathbf{v}$ in the SRV analysis tool but only a reduction of 0.12 % in SRV thrust is found. Note that the vortex filaments were still straight and not helical.
- The airfoil section properties are calculated by XFOIL based on the undisturbed air speed and mean chord and variation of the chord and variation of the air speed over the vane span are not taken into account (Assumption A5 in Subsection 4.2.4). For the high-thrust case, especially the velocity varies considerably over the span as shown in Figure 2.15 and leads to differences in Reynolds number based on chord Re_c and local Mach number M for compressibility correction. When for $J = 0.95$ the average inflow velocity of $V = 103 \text{ m/s}$ is used to estimate these values, an increase in vane thrust of 2.0 % is found. This results in a further deviation of tool and CFD data, although the effect is small.
- The earlier mentioned periodic loading of the vane may result in transient effects that can explain part of the difference in SRV thrust between the quasi-steady prediction and transient CFD result (Assumption A7 in Subsection 4.2.4). According to McCroskey [52], ‘If the angle of attack oscillates around a mean value α_0 that is of the order of the static-stall angle, large hystereses develop in the fluid-dynamic forces and moments with respect to the instantaneous angle $\alpha(t)$.’ In general this leads to an increase of $C_{l_{max}}$ which is clearly not the case here. Gharali and Johnson [53] describes a case of a pitching airfoil under unsteady freestream velocity where in a specific case when the phase difference between the unsteady freestream and the pitching of the airfoil is high, a reduction in loading was found with respect to the static case. In order to test the effect of periodic loading of the vanes, a simulation with a mixing plane at the interface between the propeller and SRV is conducted: The mixing plane circumferentially averages all flow quantities and thus the inflow field to the SRV is steady. No significant deviation from the average loading with periodic loading is found however.

- In the vane wall shear line plots in Figures 5.12 and 5.13 for the design for cruise and design for high-thrust respectively, large radial components can be distinguished, especially for low advance ratios: On the suction side near the trailing edge a radial inward component at the tip and outward component at the root is present, and on the pressure side the reverse. This radial flow can be related to the derivative of the bound vortex strength which determines the local trailing vortex strength. Therefore, near the radial location of maximum loading, the radial flow direction changes sign. The sheet of trailing vortices allows for the discrete radial velocity difference at the trailing edge between the suction and pressure side according to Prandtl [40]. Because of the large variation in vane spanwise loading, the radial flow is so pronounced for low advance ratios. According to Abbott and Von Doenhoff [39]: ‘The wing characteristics may be predicted from the known aerodynamic characteristics of the wing section if [...] the chordwise component of the velocity is large compared with the spanwise component’. The question can be raised whether this radial velocity component is still small compared to the chordwise component and may lead to changes in loading as found from lifting-line theory with airfoil section data (Assumption A6 in Subsection 4.2.4). A lot of research on the effects of radial flow is available for rotating lifting surfaces. For these surfaces, not only the loading, but also the centrifugal forces determine the radial flow. This radial flow increases lift: According to Snel et al. [54] on the suction side, flow towards the tip develops which results in a Coriolis force in the flow direction, acting as a favourable pressure gradient. This leads to a cambering effect by reduction of the displacement thickness and thus an increase in lift. However, for the vanes, no Coriolis force is present, which makes comparison impossible. The radial flow is very large and an effect on the loading is therefore very likely. The effective chord length increases, in other words, the flow travels a longer distance from leading to trailing edge. This may result in additional boundary layer growth which would have a decambering effect. This is however speculation and it is unknown what the effect of the radial flow on the loading is.

Most likely, the difference in loading of the SRV between the prediction and CFD results is mainly caused by the combination of airfoil section properties, the chord effect as described by Jones [51] and the yet unknown effect of the radial flow.

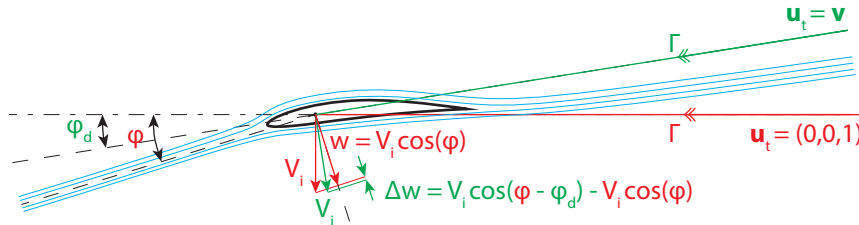


Figure 5.5: Sketch showing the effect of the assumption of the trailing vortex direction on the downwash at the lifting line.

In general the CFD results show that swirl recovery vanes lead to an increase in propulsive efficiency by increasing the system thrust over a wide range of advance ratios, with minor effect on the system power. The design for cruise proves that it is possible to increase the propulsive efficiency of an operating point close to the point of maximum propeller propulsive efficiency. The design for high-thrust shows that if a larger increase in propulsive efficiency at low advance ratios is desired, the design can be changed at the cost of propulsive efficiency benefit at higher advance ratios, for a fixed pitch design. Improvements to the design are possible, especially improvements to the airfoil section to postpone flow separation. It is recommended to also look into variable pitch designs as were presented in the previous chapter to avoid flow separation. Improvements to the SRV analysis tool are also possible, most importantly the estimation of airfoil section properties, the chord effect and correction for the radial flow. Regarding the chord effect, Jones [51] presents a correction factor to the lifting line theory which may be applied. Interchanging the lifting line method with a vortex lattice method may also improve the estimation.

For the SRV design for cruise the next subsection gives a description of the slipstream alongside results without SRV for comparison. In the next chapter simulation results are presented for this design with a trailing wing.

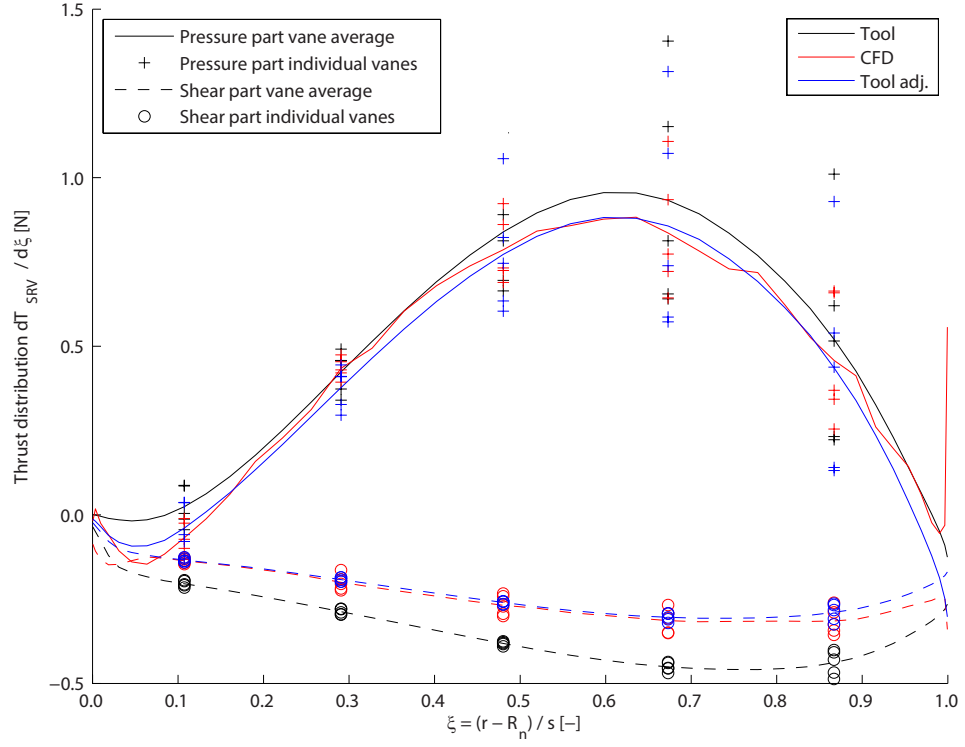


Figure 5.6: Average and individual thrust distribution along the span of the SRV for the design for cruise at $J = 1.60$ as predicted by the SRV analysis tool, resulting from the CFD simulation and predicted by the tool with adjusted airfoil section properties.

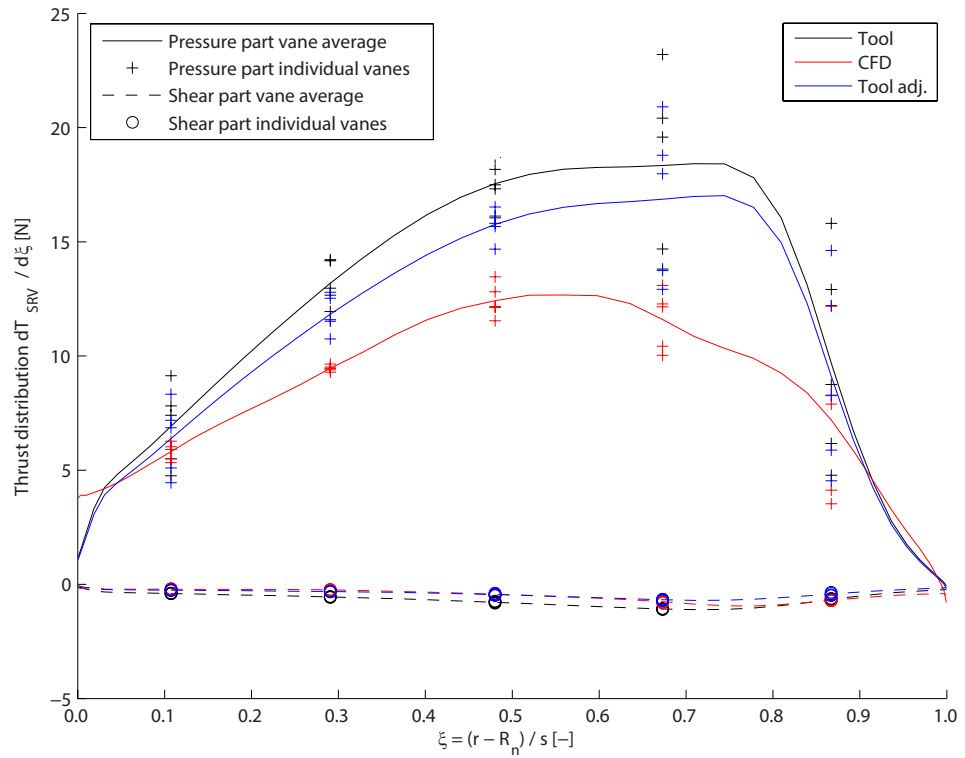


Figure 5.7: Average and individual thrust distribution along the span of the SRV for the design for cruise at $J = 0.95$ as predicted by the SRV analysis tool, resulting from the CFD simulation and predicted by the tool with adjusted airfoil section properties.

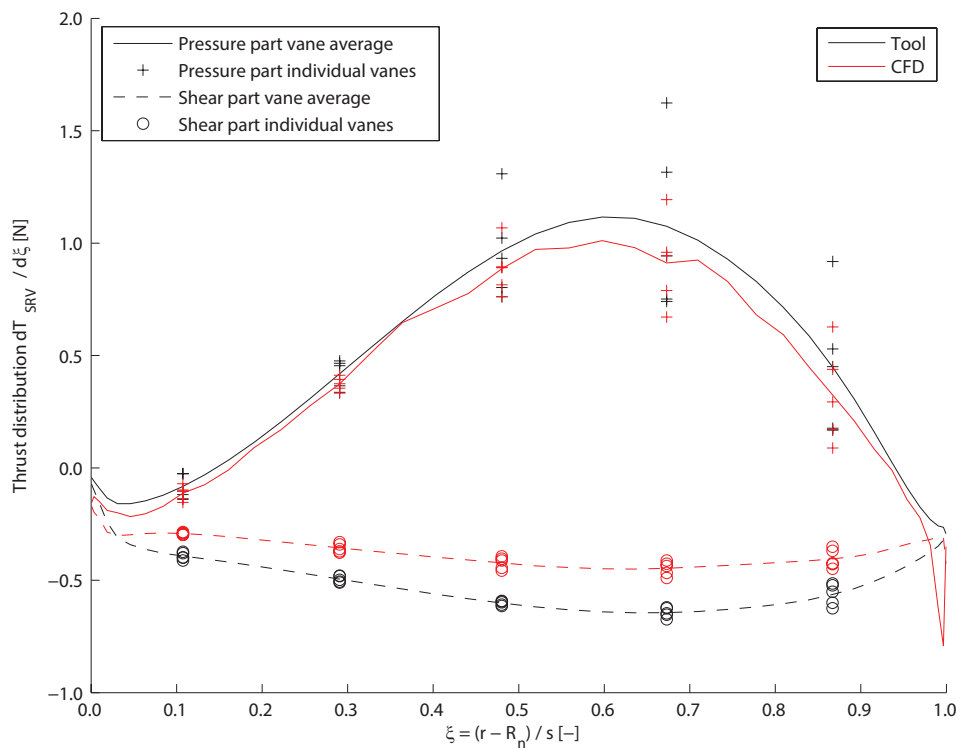


Figure 5.8: Average and individual thrust distribution along the span of the SRV for the design for high-thrust at $J = 1.60$ as predicted by the SRV analysis tool and resulting from the CFD simulation.

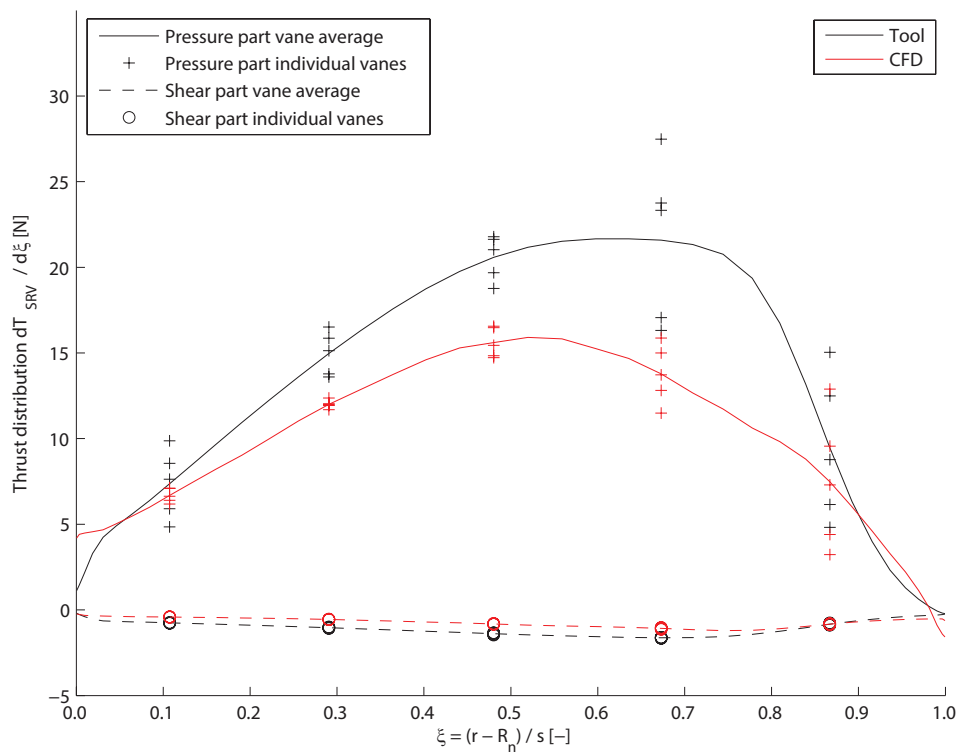


Figure 5.9: Average and individual thrust distribution along the span of the SRV for the design for high-thrust at $J = 0.95$ as predicted by the SRV analysis tool and resulting from the CFD simulation.

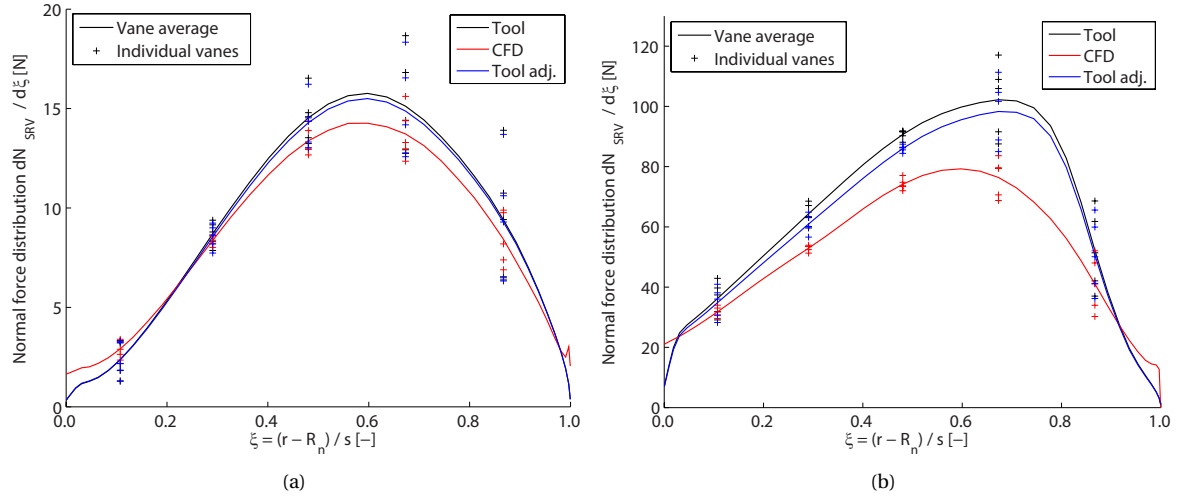


Figure 5.10: Average and individual normal force distribution along the span of the SRV for the design for cruise at (a) $J = 1.60$ and (b) $J = 0.95$ as predicted by the SRV analysis tool, resulting from the CFD simulation and predicted by the tool with adjusted airfoil section properties.

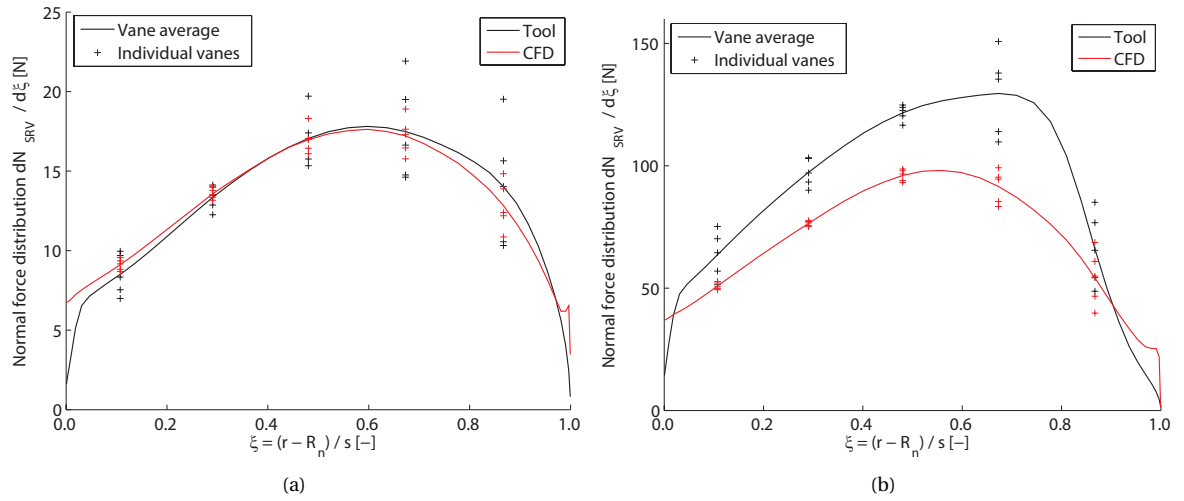


Figure 5.11: Average and individual normal force distribution along the span of the SRV for the design for high-thrust at (a) $J = 1.60$ and (b) $J = 0.95$ as predicted by the SRV analysis tool and resulting from the CFD simulation.

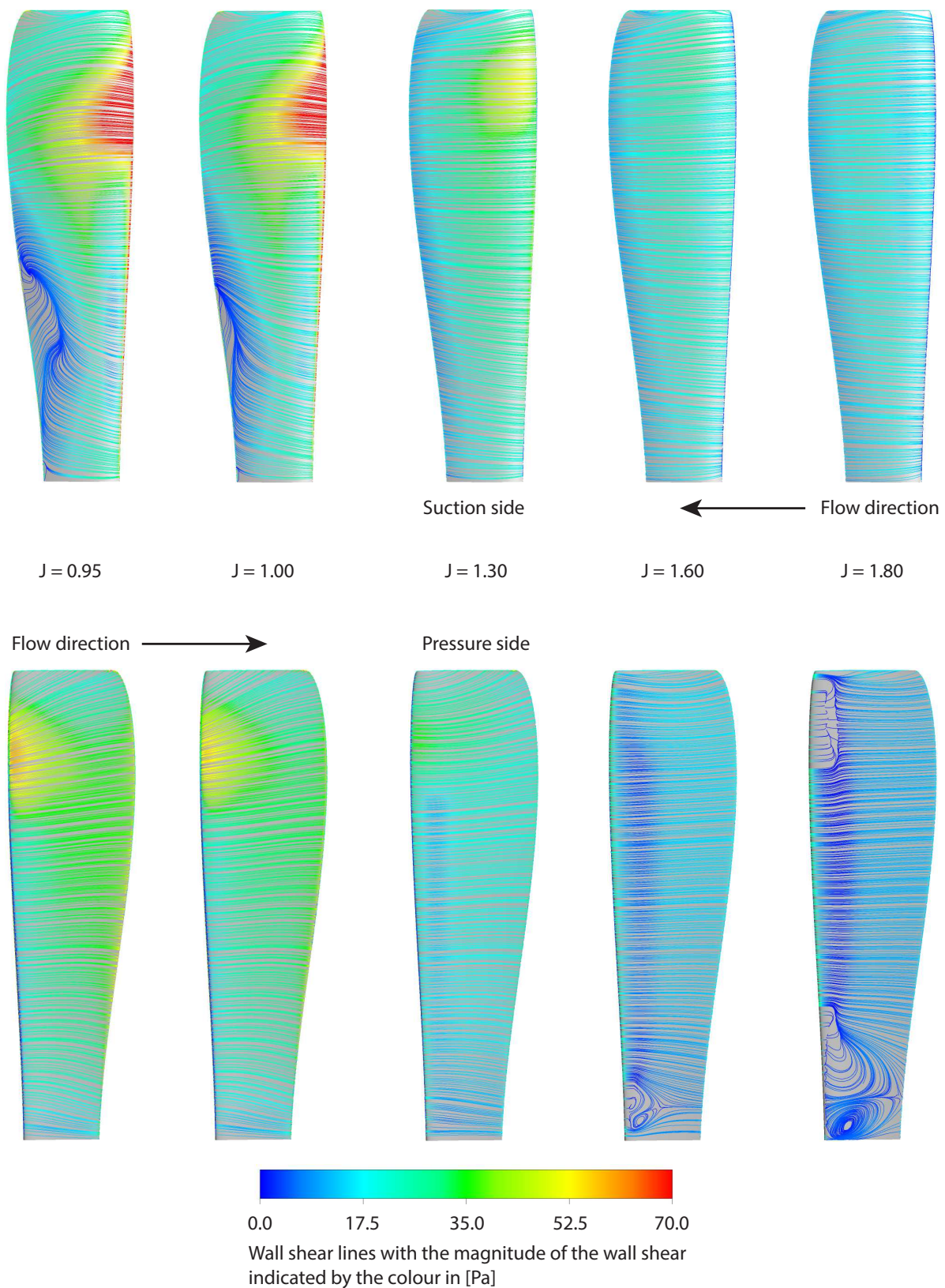


Figure 5.12: Shear lines on the suction and pressure side of a vane designed for cruise for a range of advance ratios including the magnitude of the shear stress at an instance in time.

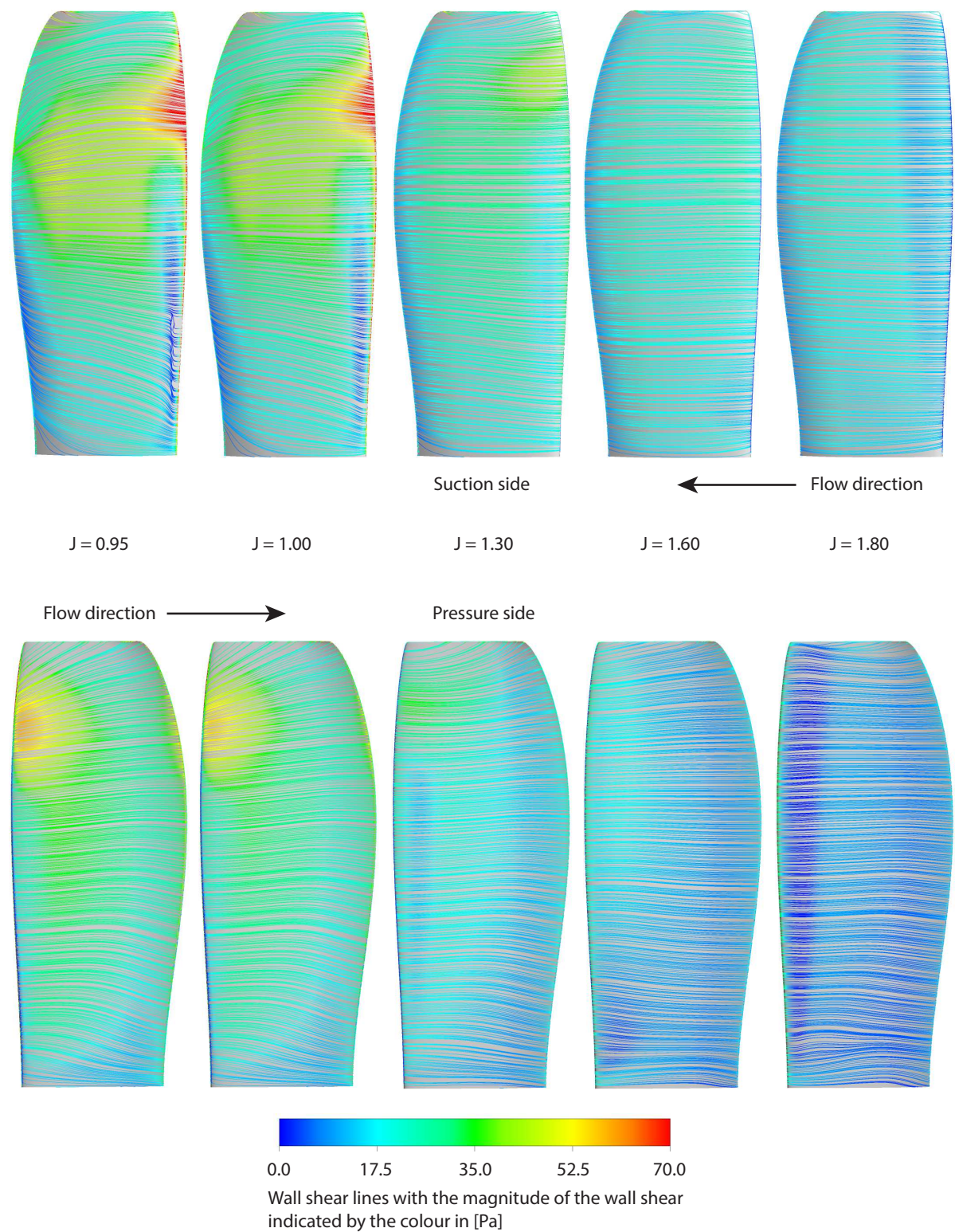


Figure 5.13: Shear lines on the suction and pressure side of a vane designed for high-thrust for a range of advance ratios including the magnitude of the shear stress at an instance in time.

5.3.3. DESCRIPTION OF THE SLIPSTREAM

In Chapter 2 the slipstream has been described of the APIAN propeller without swirl recovery vanes. This subsection describes the effect of SRV on the slipstream. The SRV designed for cruise are chosen for this analysis and only results from the 360° domain simulations will be shown. Figure 5.15 indicates four planes downstream of the propeller by different line styles. At these planes, the circumferential-average radial distribution of axial and tangential velocity component in Figures 5.16, 5.18 and 5.20 and swirl angle and total pressure in Figures 5.17, 5.19 and 5.21 are plotted at $J = 1.60$, $J = 1.30$ and $J = 0.95$ respectively. In these figures also the results of Chapter 2 are plotted without SRV for comparison. Figure 5.22 shows three isosurfaces of the vorticity behind the propeller at $J = 1.60$, $J = 1.30$ and $J = 0.95$ including a contour plot of the axial vorticity component. These results will be discussed for each advance ratio:

J=1.60 / Cruise First consider the slipstream results in Figures 5.16 and 5.17 for the case without SRV. The circumferential-average axial velocity increases slightly in downstream direction: the static pressure is still reducing to free-stream conditions. The tangential velocity only shows a minor change in downstream direction. As a result, the swirl angle decreases slightly. The total pressure changes very little. When the SRV is present, upstream of the SRV at $0.45R_p$ all flow quantities are completely in line with those without SRV. This is in line with the very minor upstream effect of the SRV on the propeller performance as shown in the previous subsection. At the plane $1.35R_p$ just behind the SRV the axial velocity is still very similar to that without SRV. The tangential velocity and resulting swirl angle reduces by almost a factor two and the maximum shifts outward, in line with results shown in Figure 1.3 from Yamamoto [11] for the SRV design from NASA Lewis Research Center's Advanced Turboprop Project. The total pressure is not affected by the SRV except for a slight reduction of the maximum. Further downstream the SRV causes a small expansion of the slipstream boundary, especially visible in the axial velocity and total pressure. The maximum in axial velocity and total pressure reduces by this expansion.

Figure 5.22 shows an isosurface of the vorticity magnitude with a contour plot of the axial vorticity component. The vortex structure is very similar to that described in Figure 3.13, except that since no cropping is applied, the SRV tip vortices are at the same radial position as the propeller tip vortices instead of below. The propeller root vortices are cut by the vanes and do not reattach downstream. SRV root vortices are also present. The SRV tip vortices and propeller blade tip vortices interact with each other. Observe the sketch of one SRV tip vortex and two propeller blade tip vortices in Figure 5.14. Since the vanes are stationary, the SRV tip vortices move downstream under an angle equal to the remaining swirl angle ϕ_d . The vortex strength of the blade tip vortices is higher than the vane tip vortices due to the higher loading on the blades. At the crossing of a vane and blade tip vortex, a deformation can be noticed of both vortices: The primary deformation of the vane tip vortex is that upstream of the blade tip vortex it moves downward and downstream upward. The blade tip vortex is deformed similarly by the vane tip vortex except that due to the difference in strength, the deformation is smaller: When looking along the vane tip vortex downstream, the blade tip vortex moves upward on the left and downward on the right. Since the primary deformation moves the vortices out of the plane they span, apart from a further primary deformation a secondary deformation takes place when they flow downstream. Two mechanism play a role in the secondary deformation: First, because now a vertical component is present for both tip vortices, in plane deformations occur. The vane tip vortex moves to the left upstream and to the right downstream of the blade tip vortex. The blade tip vortex moves downstream to the left and upstream to the right of the vane tip vortex. Second, since the blade and vane tip vortices are originally not at right angles to each other, the vane tip vortex is deformed to the left upstream and to the right downstream of the blade tip vortex, and the blade tip vortex is deformed upstream to the left and downstream to the right of the vane tip vortex. These two effects work against each other: For the deformation of the blade tip vortex, the first effect is dominant, since the vane vortex is deformed considerably by the primary deformation leading to a large vertical component. For the deformation of the vane tip vortex the second effect is dominant, since the blade tip vortex is less deformed by the primary deformation and thus a smaller vertical component is present. Another effect of not being at right angles to each other is that a component of the vane tip vortex is in the direction of the blade tip vortex, but of opposite sign: The vortices partly cancel each other. While moving downstream, the deformation of the tip vortices continues and becomes more complex with tertiary deformations. The increasingly out of plane movement of the tip vortices explains the increase of the slipstream boundary as was seen in the circumferential-average flow quantities. The structure of tip vortices remains intact and does not dissipate a lot in downstream direction for the shown range.

J=1.30 For this condition, all effects as described for $J = 1.60$ are present and more pronounced. The circumferential average flow quantities in the slipstream are shown in Figures 5.18 and 5.19. When no SRV are present, the circumferential-average axial velocity increases more in downstream direction and the tangential velocity shows some larger changes with similar changes in the swirl angle. The total pressure changes a little more a well. When the SRV is present, upstream of the SRV at $0.45R_p$ the flow quantities deviate slightly more from those without SRV, in line with the somewhat larger upstream effect of the SRV on the propeller performance. The tangential velocity and resulting swirl angle reduces again by almost a factor two and the maximum shifts outward. The expansion of the slipstream boundary further downstream is more pronounced, leading to larger reduction of the maximum in axial velocity and total pressure. This is caused by the larger deformation of the blade and vane tip vortices as can be seen in Figure 5.22.

J=0.95 / High-thrust The circumferential-average flow quantities in the slipstream are shown in Figures 5.20 and 5.21. When no SRV are present, the circumferential-average axial velocity increases again in downstream direction similarly as for the other two conditions. The tangential velocity at $0.45R_p$ shows a clear second peak which reduces downstream. This may be due to dissipation related to the mesh and temporal and spatial discretization. For the swirl angle a similar second peak is visible which reduces downstream. The total pressure changes considerably more as well and shows a slight expansion of the slipstream boundary. When SRV are present at the plane $0.45R_p$ again only slight differences can be noticed in all flow quantities compared to without SRV. Downstream of the SRV the swirl angle reduces almost by a factor two again. For this condition, the largest difference with respect to the other cases is the very large expansion of the slipstream boundary. In Figure 5.22 one can see that the primary effect of the vane tip vortices on the blade tip vortices is very large: the blade tip vortices are deformed completely vertical, expanding far outside the initial slipstream boundary. In between these sheets of vorticity the blade tip vortices dissipate early, much earlier than without SRV. Clearly only five structures remain. It is difficult to conclude if propeller vortex structure breakdown happens in reality or is solely due to a difference in numerical dissipation by a difference in distance from the plane to the propeller and to the SRV.

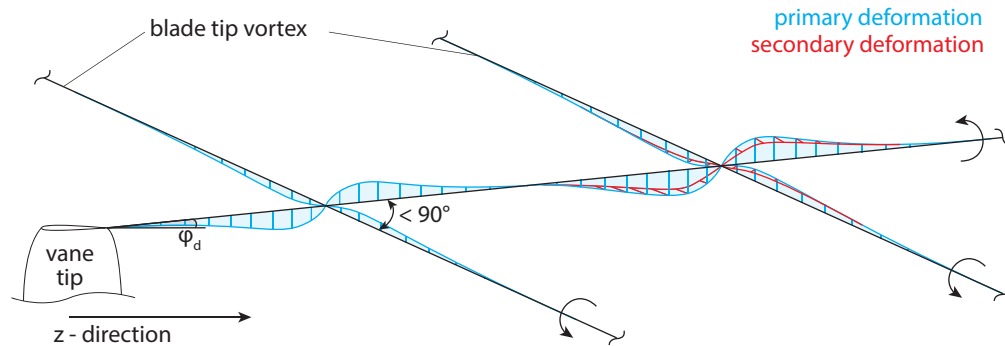


Figure 5.14: Sketch of propeller blade and SRV tip vortex interaction.

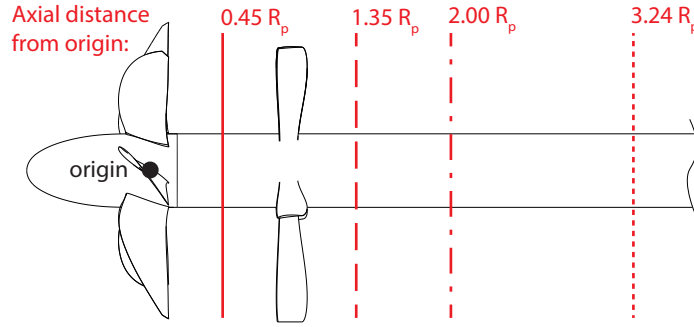


Figure 5.15: Definition of four planes downstream of the propeller in a side view of the propeller-SRV configuration.

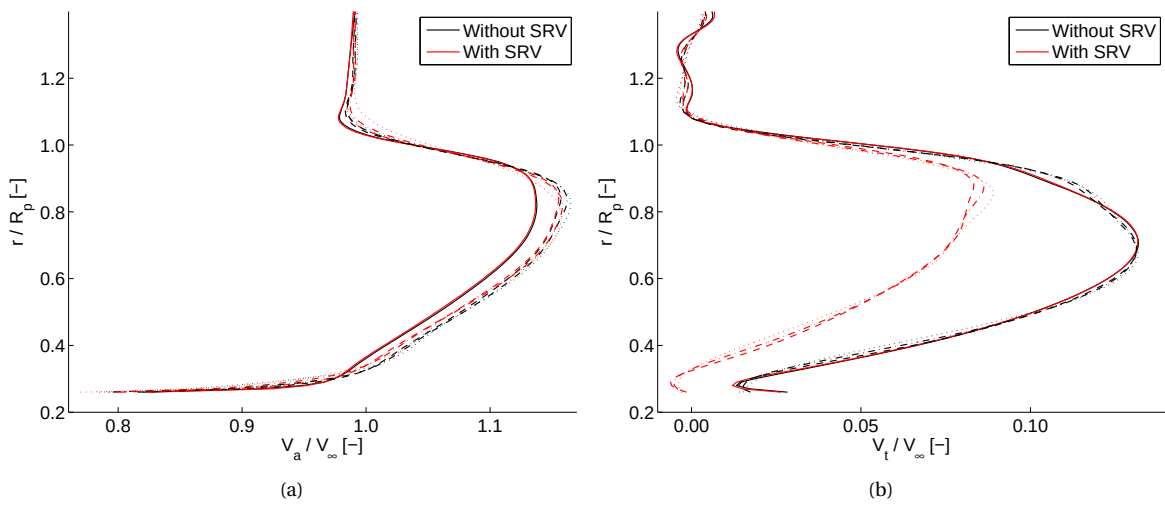


Figure 5.16: Radial distribution of the circumferential-average (a) axial velocity and (b) tangential velocity component without and with SRV in four planes downstream of the propeller as defined in Figure 5.15 for the SRV designed for cruise at $J = 1.60$.

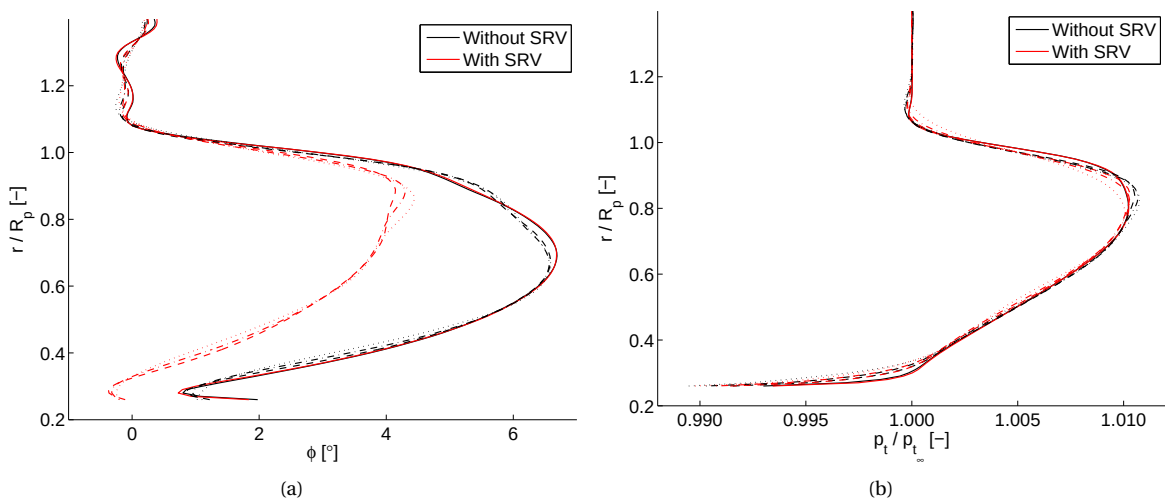


Figure 5.17: Radial distribution of the circumferential-average (a) swirl angle and (b) total pressure without and with SRV in four planes downstream of the propeller as defined in Figure 5.15 for the SRV designed for cruise at $J = 1.60$.

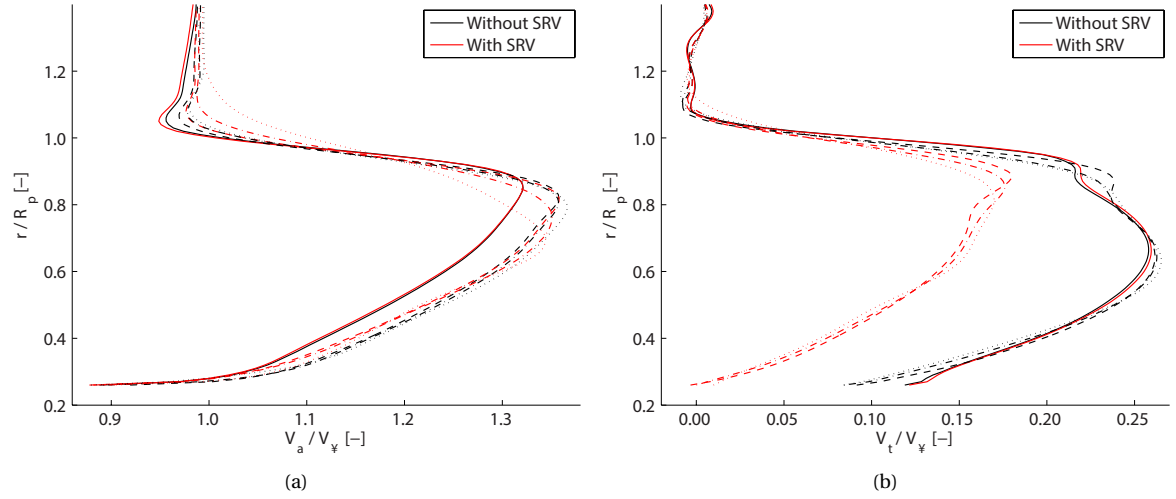


Figure 5.18: Radial distribution of the circumferential-average (a) axial velocity and (b) tangential velocity component without and with SRV in four planes downstream of the propeller as defined in Figure 5.15 for the SRV designed for cruise at $J = 1.30$.

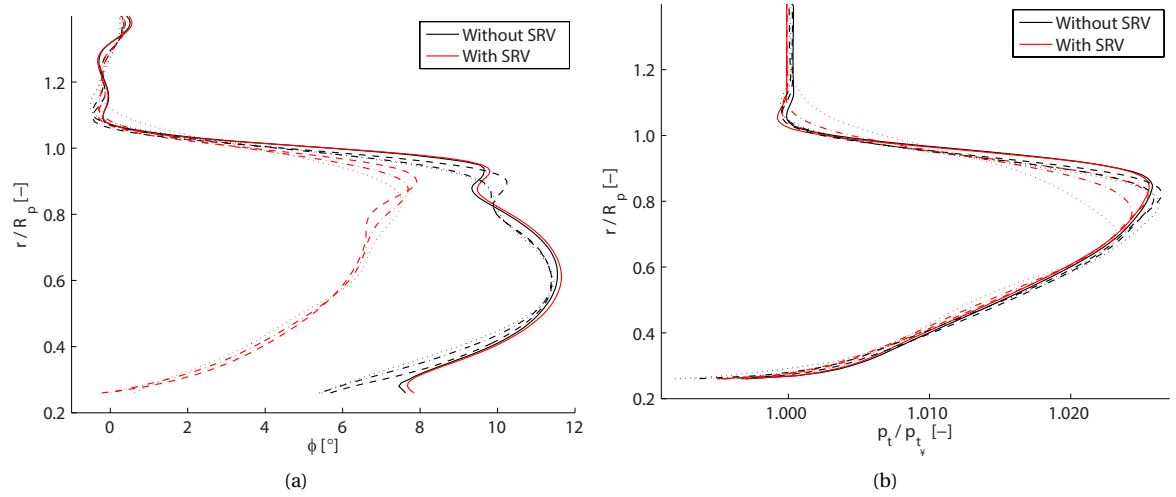


Figure 5.19: Radial distribution of the circumferential-average (a) swirl angle and (b) total pressure without and with SRV in four planes downstream of the propeller as defined in Figure 5.15 for the SRV designed for cruise at $J = 1.30$.

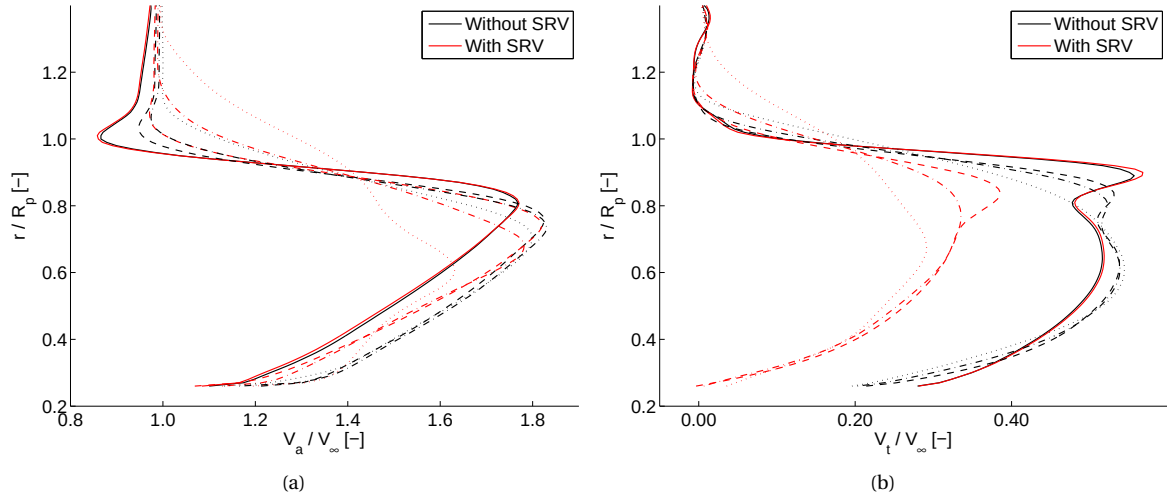


Figure 5.20: Radial distribution of the circumferential-average (a) axial velocity and (b) tangential velocity component without and with SRV in four planes downstream of the propeller as defined in Figure 5.15 for the SRV designed for cruise at $J = 0.95$.

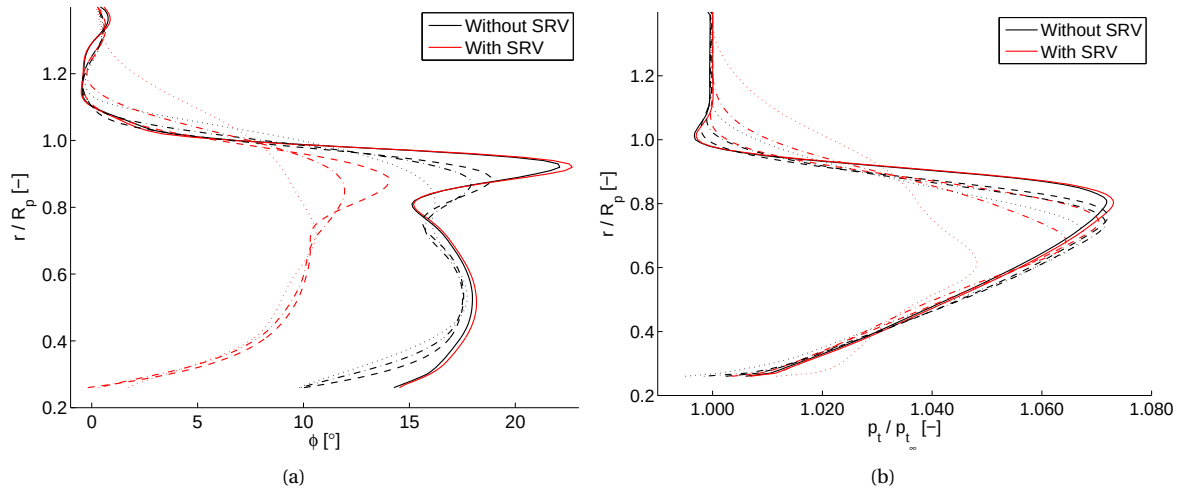


Figure 5.21: Radial distribution of the circumferential-average (a) swirl angle and (b) total pressure without and with SRV in four planes downstream of the propeller as defined in Figure 5.15 for the SRV designed for cruise at $J = 0.95$.

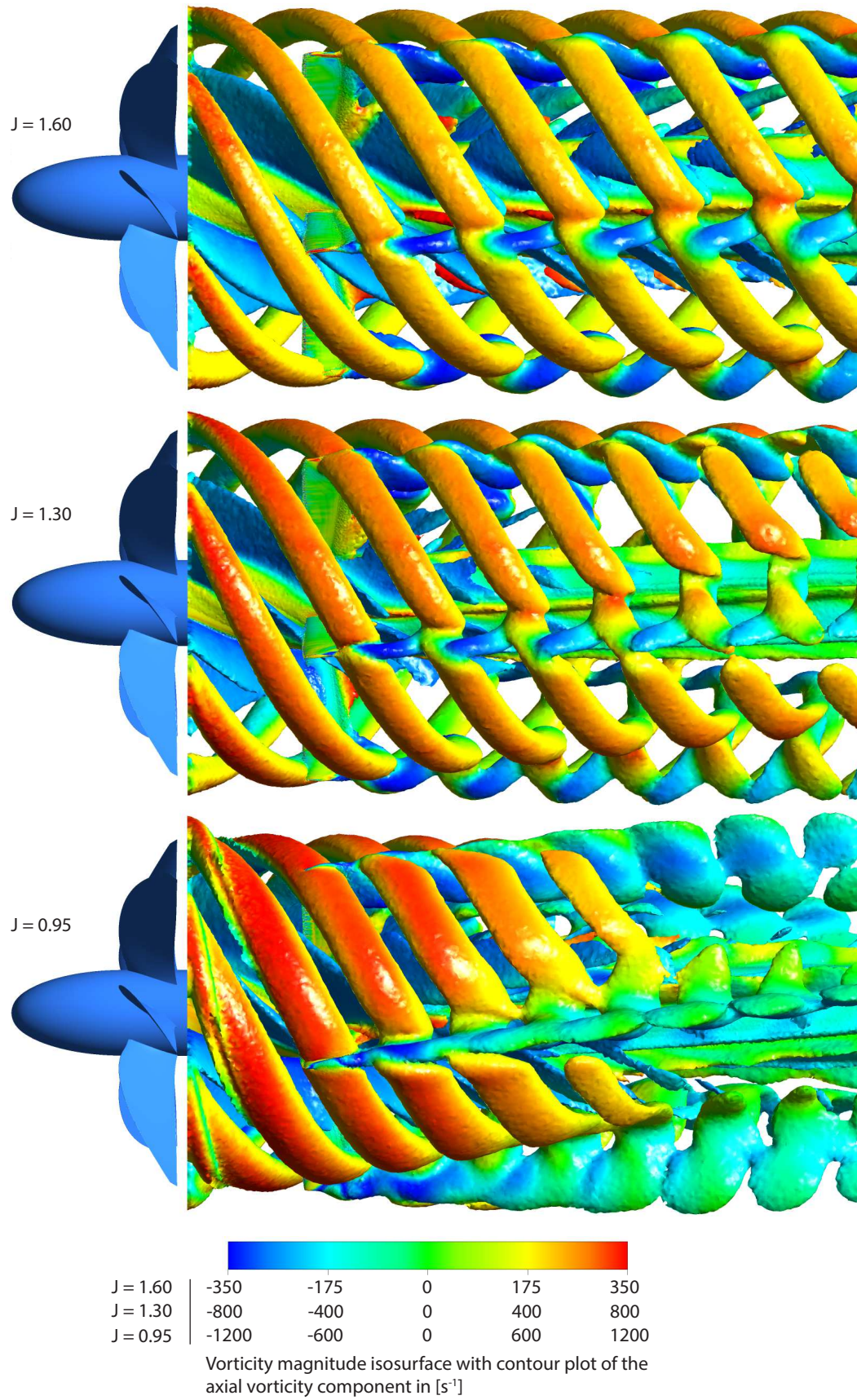


Figure 5.22: Isosurfaces of the vorticity magnitude behind the APIAN propeller with SRV designed for cruise including a contour plot of the axial vorticity component at an instance of the propeller blade passage at $J = 1.60$, $J = 1.30$ and $J = 0.95$.

6

PROPELLER-SRV-WING CFD MODEL

In this chapter a wing is introduced behind the propeller and swirl recovery vanes designed for cruise. In the previous chapter gains in the propulsive efficiency by the addition of these SRV were found for a large range of advance ratios. A trailing wing may affect conclusions drawn on the performance gain of SRV and the airplane in general. A number of aspects were discussed in Section 1.3:

- The wing swirl recovery may be reduced due to the reduction of the swirl by the SRV, resulting in a higher wing drag than for the propeller without SRV. In Section 5.3.3 indeed a downstream reduction of swirl by almost a factor two is found.
- The axial velocity may be increased by the SRV, leading to a local increase in lift and drag forces. However, in Section 5.3.3 downstream of the SRV a small reduction of axial velocity is found, mainly due to an increase of the slipstream boundary.
- The propeller and SRV operate in the induced upwash by the wing, resulting in azimuthal varying propeller blade loading and corresponding circumferentially non-uniform slipstream. The question can be raised whether the SRV performance is affected by this upwash and altered slipstream.
- Co-rotating propellers are associated with trimming drag due to the asymmetry of drag and lift between both wing-halves. SRV may reduce these asymmetric effects and the corresponding trimming drag.

All aspect except for the last will be discussed quantitatively in this chapter. On the last aspect about the reduction of asymmetric effects only qualitative statements can be made, since only a counter-rotating propeller-wing configuration is tested and also no quantitative statements can be made on trimming drag. The geometry of this configuration is presented in Section 6.1. The mesh including a mesh dependency study for the wing is discussed in Section 6.2. In Section 6.3 all results are presented and discussed.

6.1. GEOMETRY

This section describes the propeller-SRV-wing geometry and the corresponding CFD model domain geometry.

6.1.1. PROPELLER-SRV-WING GEOMETRY

In this chapter a wing is introduced behind the APIAN propeller and SRV designed for cruise. The wing is a slightly altered scaled Fokker F50 wing. Figure 6.1 presents a sketch of the geometry. The propeller is located $2R_p$ in front of the local wing-section quarter-chord position and the location of the SRV is unaltered with respect to the propeller. The wing properties are given in Table 6.1 and are derived from Veldhuis [16]. The wing is scaled such that the original propeller diameter matches that of APIAN propeller. This is considered a reasonable method for scaling since the original propeller is six bladed as well. It is chosen not to change the propeller blade angle since not only the SRV are designed with the slipstream of the current blade angle, but also validation of the propeller performance and slipstream is performed for this blade angle only. The consequence is that the ratio of propeller thrust and wing drag may deviate from an actual ratio found in cruise flight. A simplification with respect to the original wing is the negligence of dihedral of 2.5° for the

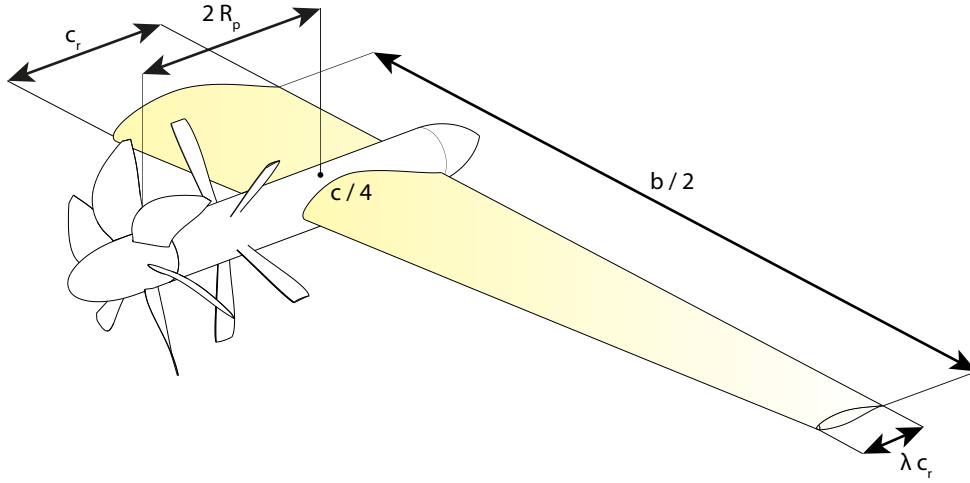


Figure 6.1: The geometry of the APIAN propeller, SRV designed for cruise and Fokker F50 wing.

outer 75% of the wing half-span to avoid this step change of the wing and corresponding required mesh refinement. A further deviation is that the original nacelle is of much larger diameter to store the landing gear as well as the gas turbine and is also not a body of revolution. At last, the fuselage is not considered and the half-wing is extended over the full half-span. Since the goal is not to represent the Fokker F50 as close as possible but just to have a representative wing for a tractor configuration, these deviations are allowed. For a realistic implementation of SRV to the Fokker 50, one would need to consider them.

Table 6.1: Overview of Fokker F50 scaled wing properties based on Veldhuis [16].

Variable	Value
Scale	1/7.32
Wing area S	1.352 m^2
Wing span b	4.098 m
Wing root chord c_r	472.7 mm
Taper ratio λ	0.396
Quarter chord sweep $\Lambda_{c/4}$	1.24 $^\circ$
Dihedral	0.00 $^\circ$
Root incidence angle θ_r	3.27 $^\circ$
Twist Θ	-2.00 $^\circ$
Root airfoil	NACA 64 ₂ 421
Tip airfoil	NACA 64 ₂ 415

6.1.2. DOMAIN GEOMETRY

The fluid domain is shown in Figure 6.2. It consists of an outer, rotating, wing and wake region. The rotating region and the part of the wake region containing the SRV are unaltered with respect to the propeller-SRV CFD model of the previous chapter. The wake region extends downstream of the nacelle to capture the wake of the propeller over the SRV, nacelle and wing. The wing as well as the rotating and wake region are contained in the wing region, extending considerably downstream of the wing to capture the wake of the wing. The outer domain is of much larger size than the outer domain for the CFD models without wing as shown in Figure 2.2, since the larger size of the model geometry requires a larger outer domain in order to minimize influencing the flow properties near the model by the boundary conditions. The length of the nacelle including spinner and hub l_n is used for dimensioning the outer domain.

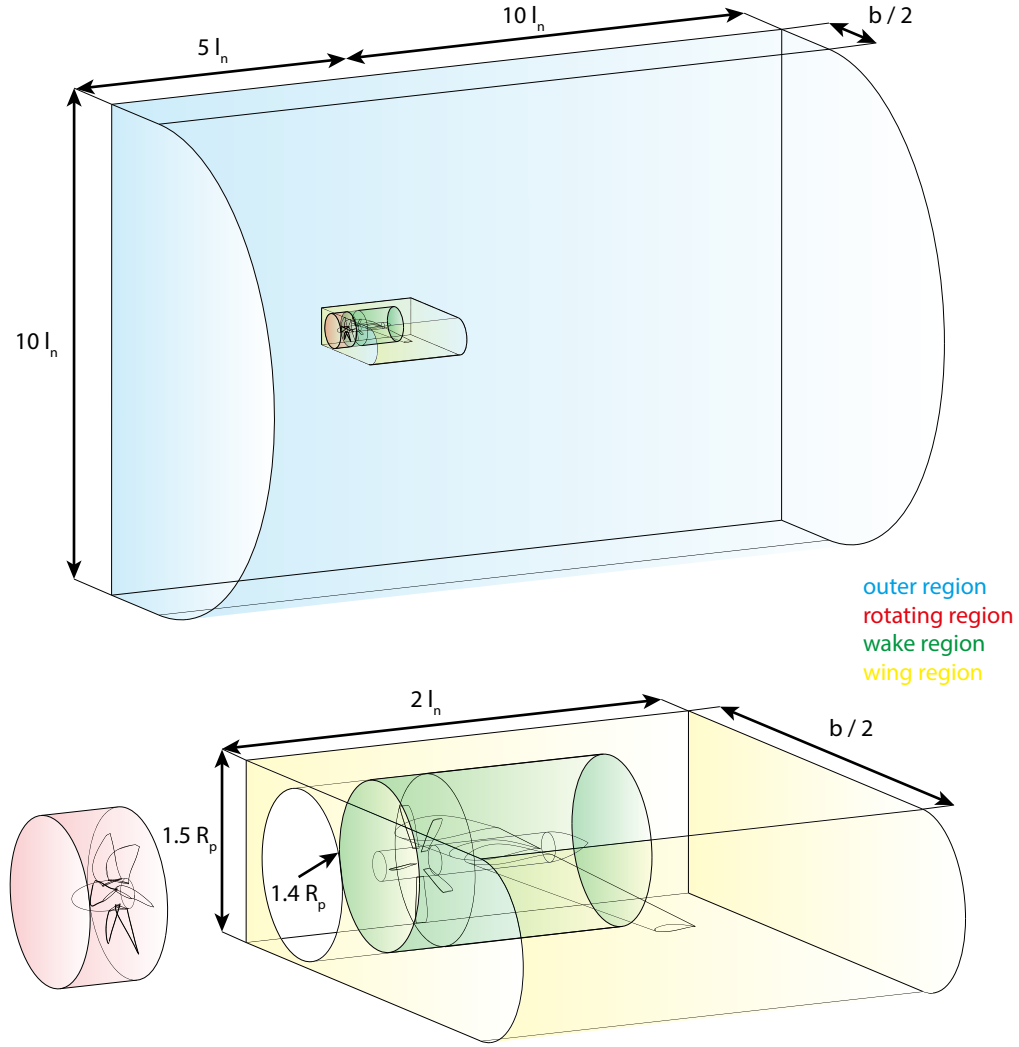


Figure 6.2: Detailed view of the fluid domains of the propeller-SRV-wing CFD model.

6.2. MESH

Apart from the wing wall and wing region volume refinement, the properties of the mesh are the same as for the propeller-SRV CFD model, which are on its turn based on those of the propeller and APIAN-INF CFD models introduced in Sections 2.2 and 3.2 respectively. On the wing wall two different wall refinements are specified in order to investigate mesh dependency of the results. The mesh dependency study is performed with an isolated wing. The mesh size and refinement dimensions of this coarse and fine wing wall refinement for the mesh dependency study are given in Table 6.2. A notable change with respect to the mesh properties on the propeller and SRV is the increased number of inflation layers. This is necessary in order to capture the boundary layer in the inflation layer over the full wing chord. The results of these two refinements are discussed in Subsection 6.3.1. The coarse mesh is chosen for the propeller-SRV-wing CFD model and the corresponding mesh size is given in Table 6.3, split in the rotating region and all other regions combined in the stationary region with and without SRV.

Table 6.2: Overview of mesh size and refinement dimensions of the mesh dependency study for the propeller-SRV-wing CFD model.

refinement	# nodes	nr. of inflation layers	volume size [mm]	wall size [mm]	l.e. and tip size [mm]	t.e. size [mm]
Coarse	3,077,181	20	20	7.00	2.00	0.75
Fine	4,980,374	20	20	5.00	1.25	0.50

Table 6.3: Overview of mesh size of the propeller-SRV-wing CFD model.

Case	# nodes stationary region	# nodes rotating region
Without SRV	4,852,022	4,575,306
With SRV	7,512,281	4,575,306

6.3. RESULTS

A broad range of results are generated for the propeller-SRV-wing CFD model. First the results of the mesh dependency study are given in Subsection 6.3.1, which are simulations of the isolated wing. These simulations give insight into the wing wall mesh refinement. By means of simulations of the propeller, nacelle and wing without SRV the effect of the wing on the SRV inflow field is investigated and conclusions on the design of SRV are drawn in Subsection 6.3.2. Based on these findings the design of the SRV is slightly adjusted. In Subsection 6.3.3 the effect of the wing on the propeller and SRV loading is discussed and in Subsection 6.3.4 the effect of the addition of SRV on the overall thrust and lift force balance is investigated. The corresponding effect on the slipstream is described in Subsection 6.3.5. The idea was to generate these results at $J = 1.60$, $J = 1.30$ and $J = 0.95$, in line with the conditions for which a description of the slipstream is given in Subsection 5.3.3 without wing. In this way the effect of the wing can be better investigated. Due to conclusions drawn on the SRV design in Subsection 6.3.2, the high-thrust condition $J = 0.95$ is left out of this discussion. Many of the results for $J = 1.30$ are given in Appendix B to enhance readability of this chapter.

6.3.1. MESH DEPENDENCY STUDY

For the mesh dependency study two wall mesh refinements of the wing are considered, a coarse and fine mesh. The fine mesh contains 62 % more nodes, leading to a considerable increase in computational time. In Figure 6.3 the lift and drag distribution are shown along the half-span of the wing. Both the results with

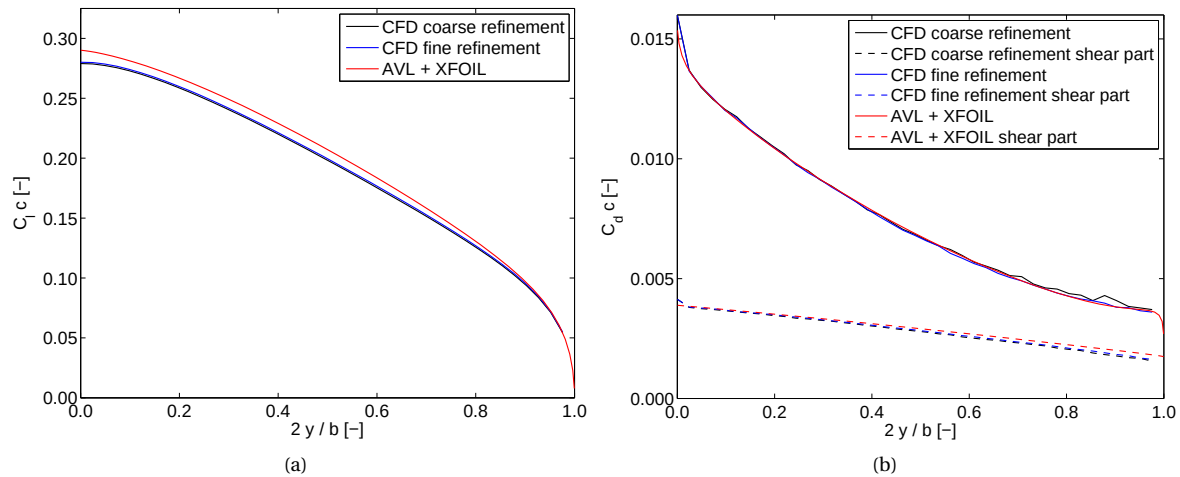


Figure 6.3: Wing (a) lift and (b) drag distribution along the span of the F50 wing without propeller, SRV and nacelle for the coarse and fine mesh of the mesh dependency study including a prediction by the vortex lattice program AVL [55] and XFOIL [41] for verification.

the coarse and fine mesh refinement are plotted as well as a prediction by the vortex lattice program AVL [55] combined with XFOIL [41] for the section properties. By visual inspection almost no difference between the

two wall refinements can be noticed. The fine mesh refinement leads to a 0.55 % increased wing lift coefficient and 0.88 % decreased wing drag coefficient. By AVL and XFOIL a 3.57 % higher wing lift coefficient and -1.50 % lower wing drag coefficient are predicted with respect to the coarse mesh refinement. The agreement of the shape of the lift and drag and shear part of the drag distribution is very good. It is unlikely that further mesh refinement explains the differences between the prediction and simulation and they are most likely the result of a combination of differences in section properties and spanwise flow as was the case for the SRV loading explained in Subsection 5.3.2. The coarse mesh is chosen for the remainder of the simulations in this chapter, since the effect of mesh refinement is small and does not weigh up against the increased computational time.

6.3.2. EFFECT OF WING ON SRV DESIGN

The swirl recovery vanes in this research are designed for the slipstream of the isolated propeller and nacelle. The loading on a wing results in an upwash upstream of the wing. This results in a change of the slipstream and thus in a deviation from the design condition. Initially, it was thought that this effect would be small enough such that a SRV design for an isolated propeller would still work well when a wing is present. However, it is found that due to the upwash the performance of the SRV design for cruise is heavily degraded. Observe Figures 6.5 (a) and 6.6 (a) which show the time-average swirl angle radial distribution at the location of the vanes without and with wing for the cruise and high-thrust condition respectively. Note that these results are from simulations without SRV and the location of the SRV corresponds to their quarter chord, in the wake rake as defined in Figure 2.6. Figure 6.4 defines the symbol belonging to each vane. When no wing is present, the time-average swirl angle distribution is the same for each vane. When the wing is present the inflow field for each vane is different: A shift of radial distribution of swirl angle can be noticed that is different for each vane location.

Simulations with the original SRV design for cruise have been performed: At $J = 1.60$ the swirl angle locally changes such that on the pressure side flow separation over the full chord for part of the span of vane $+$ occurs. At $J = 0.95$ the simulation did not converge due to flow separation on the suction side of vane $\circ$ and $\square$. Both occurrences of stall lead to a large decrease of the net SRV thrust.

The question is how the SRV design should be changed to perform well when a wing is present. Since each vane experiences a different time-average flow field, ideally one would find a different design for each vane. In an optimisation procedure the number of variables would increase by a factor equal to the number of vanes N_{SRV} . Since the SRV analysis tool does not allow for individual vane design, this option is not chosen but is recommended for future work. A different option is that since the shape of the swirl radial distribution does not change a lot, each vane is turned individually over the temporal- and radial-average change in swirl angle. This results in similar vane loading as without wing, avoiding large flow separation. Although the loading may be similar, due to the differences in swirl angle, the orientation of the lift and drag vectors of each vane change, and thus the corresponding vane thrust changes. A third option is to change the downstream location of the SRV: Figures 6.5 (b) and 6.6 (b) show the time-average swirl angle radial distribution at half-distance from the propeller. Since the upwash decreases with upstream distance from the wing, the upwash effect on the swirl angle is less, although still present. Due to the large computational cost of the simulations, only one of the options can be explored within the set time-frame and it is chosen to leave the SRV at its original location to maintain a better comparison with the simulations without wing.

Table 6.4: Required change of vane root pitch $\Delta\theta_r$ to maintain temporal- and radial-average design inflow angle for each vane as defined in Figure 6.4.

location	J [-]	$\Delta\theta_r$ [°] $+$	$\circ$	$\square$	$\diamond$	Δ
design	1.60	+2.4	-1.7	-2.4	-0.9	+1.0
half-distance	1.60	+1.2	-1.1	-1.8	-0.7	+0.8
design	0.95	+0.9	-1.6	-2.1	-0.5	0
half-distance	0.95	+0.3	-1.4	-1.7	-0.5	+0.3

Table 6.4 gives the required change in vane root pitch angle for each vane to maintain the design temporal- and radial-average inflow angle. From this table another problem can be noticed: There is a considerable difference in required $\Delta\theta_r$ for $J = 1.60$ than for $J = 0.95$. The influence of the upwash on the slipstream swirl decreases with decreasing advance ratio. The SRV design is fixed pitch but from this observation one could conclude that a variable pitch SRV design as presented in Section 4.4 would be better when a trailing wing is

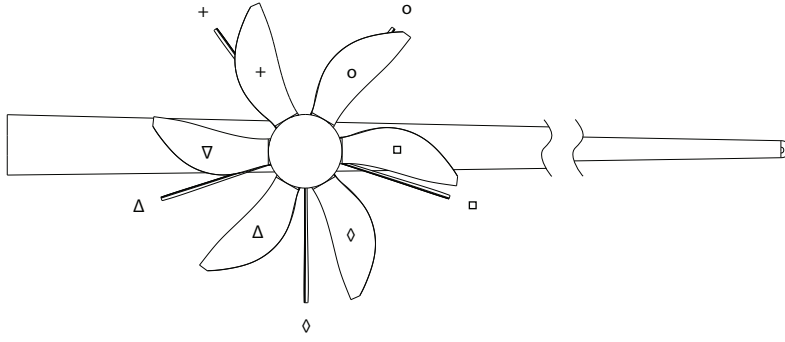


Figure 6.4: Definition of the propeller blades and swirl recovery vanes for $t/t_{bp} = 0$ in a front view of the propeller-SRV-wing configuration.

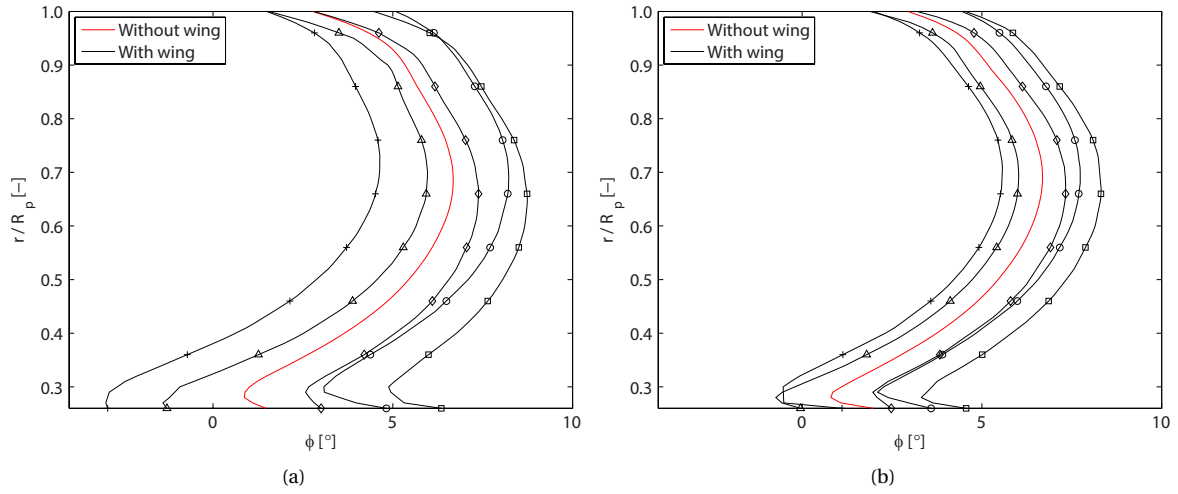


Figure 6.5: The effect of the wing on the radial distribution of time-average swirl angle at the location of the swirl recovery vanes as defined in Figure 6.4 for $J = 1.60$ at (a) the design SRV location $0.9R_p$ downstream of the propeller and (b) half distance $0.45R_p$ downstream of the propeller simulated without SRV.

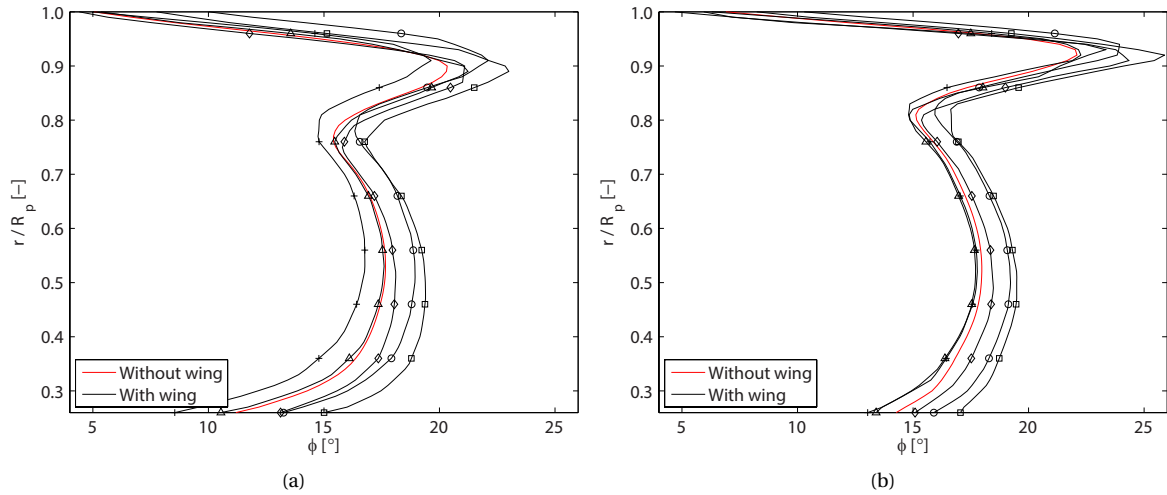


Figure 6.6: The effect of the wing on the radial distribution of time-average swirl angle at the location of the swirl recovery vanes as defined in Figure 6.4 for $J = 0.95$ at (a) the design SRV location $0.9R_p$ downstream of the propeller and (b) half distance $0.45R_p$ downstream of the propeller simulated without SRV.

present. Ideally the pitch would have to be variable for each vane individually. Since the SRV design for cruise as simulated in the previous chapter is not designed with variable pitch in mind but with a constraint on stall in the high-thrust condition, it is not chosen to use this design for variable pitch. Only a change in the design is made by turning the vanes over the angle $\Delta\theta_r$ corresponding to their design condition, so for $J = 1.60$, and this is left constant for all other operating conditions. With this design for $J = 0.95$ still no converged solution is found due to flow separation and thus $J = 0.95$ is omitted and only simulations for $J = 1.60$ and $J = 1.30$ are performed. It is recommended for future research to look into variable pitch design for each vane individually since from these results clear advantages can be expected.

6.3.3. EFFECT OF WING ON PROPELLER AND SRV PERFORMANCE

In this section the effect of the wing on the performance of the propeller and SRV is investigated. The root pitch angle of each of the vanes of the SRV designed for cruise is adjusted individually as described in Subsection 6.3.2. The time-average performance quantities are presented in Tables 6.5 and 6.6 without and with SRV respectively, both relative to the propeller CFD model from Chapter 2, so without wing and SRV. The given performance quantities are defined in Equation 3.1 through 3.5. So in these equations, the subscript *without* denote the results without SRV and wing. For the case without SRV, the wing leads to a considerable increase in propeller thrust and power coefficient. This can be explained by two effects: First, the blade angle of attack is increased by a high pressure region in front of the wing according to Veldhuis et al. [56]. Second, the upwash by the loading on the wing results in an increase in propeller thrust and power just like for a propeller under an incidence angle as is shown by Ortun et al. [23] and Stuermer and Rakowitz [57]. The resulting propeller propulsive efficiency increases for $J = 1.60$ and decreases for $J = 1.30$.

Table 6.5: Time-average performance quantities of propeller-SRV-wing CFD model without SRV, relative to the performance without wing and SRV from Table 2.7 as defined in Equations (3.1) to (3.4) where the subscript *without* refers to the results without SRV and without wing and the subscript *with* to the results without SRV and with wing.

J [-]	ΔC_T [%]	ΔC_P [%]	$\Delta \eta_{p_p}$ [%]
1.30	+2.39	+2.49	-0.07
1.60	+3.90	+3.57	+0.24

When SRV are present, the propeller thrust and power coefficient increase slightly more. Apart from the effect of the wing, the SRV affect the propeller performance directly as described in Subsection 5.3.1 and indirectly through an effect on the wing loading. The effect of the SRV on the propeller propulsive efficiency $\Delta \eta_{p_p}$ is small and almost solely resulting from the wing. For $J = 1.60$ the increase in propulsive efficiency $\Delta \eta_p$ is considerably higher than without wing as shown in Table 5.2. This is mainly due to the increased propeller propulsive efficiency, but partly by increased SRV thrust as well. A comparison between the SRV performance with and without wing is not straight forward due to the change in root pitch angles. The increase in propulsive efficiency for $J = 1.30$ is very similar to the one without wing. So in general, the SRV still perform as well or even better than without wing.

Table 6.6: Time-average performance quantities of propeller-SRV-wing CFD model with adjusted SRV designed for cruise, relative to the performance without wing and SRV from Table 2.7 as defined in Equations (3.1) to (3.5) where the subscript *without* refers to the results without SRV and without wing and the subscript *with* to the results with SRV and with wing.

J [-]	ΔC_{T_p} [%]	ΔC_T [%]	ΔC_P [%]	$\Delta \eta_{p_p}$ [%]	$\Delta \eta_p$ [%]
1.30	+2.51	+5.89	+2.59	-0.07	+2.14
1.60	+4.12	+5.04	+3.78	+0.26	+0.93

Apart from differences in time-average performance, the wing results in periodic propeller blade loading. In Figure 6.7 the individual and average blade thrust are plotted for a single blade passage starting at the orientation defined in Figure 6.4 at $J = 1.60$. Both the blade thrust without and with SRV are plotted with respect to the time-average propeller thrust without SRV and a similar plot at $J = 1.30$ can be found in Figure B.3. While the individual blade thrust is sinusoidal with a period equal to a full propeller rotation, the average blade thrust is constant in time. The addition of SRV lead to a small increase of blade thrust almost only when the individual blade thrust is higher than the average. This corresponds to a blade position from $\dagger$ to $\diamond$ as defined in Figure 6.4. The result is a slight increase in average blade thrust.

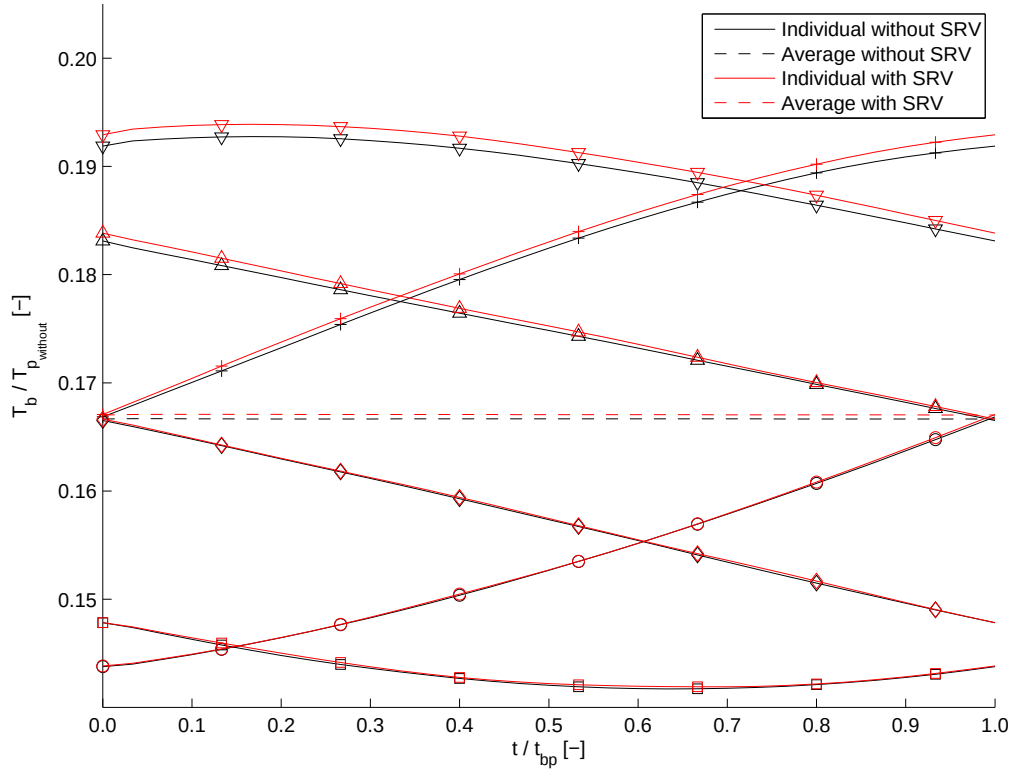


Figure 6.7: Propeller individual and average blade thrust T_b relative to the time-average propeller thrust without SRV $T_{p_{without}}$ for the duration of one blade passage period t_{bp} as defined in Figure 6.4 at $J = 1.60$.

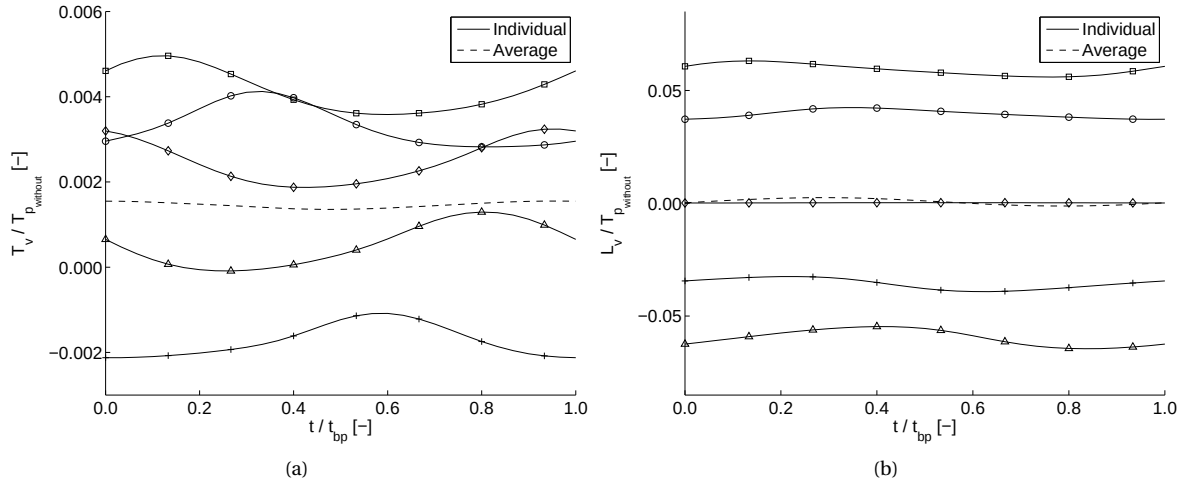


Figure 6.8: SRV individual and average vane (a) thrust T_v and (b) lift L_v relative to the time-average propeller thrust without SRV $T_{p_{without}}$ for the duration of one blade passage period t_{bp} as defined in Figure 6.4 at $J = 1.60$.

Figure 6.8 gives the individual and average vane thrust and lift relative to the time-average propeller thrust without SRV during the same propeller blade passage at $J = 1.60$. The same results are given for $J = 1.30$ in Figure B.4. Although the time-average SRV thrust is equal or higher than without wing, the individual vane thrust varies considerably from net drag to net thrust well above the average at $J = 1.60$. At $J = 1.30$ the vane thrust is positive for all vanes. The thrust on each vane is periodic with the period of the propeller blade passage. The average vane lift is slightly positive at $J = 1.60$ and negative at $J = 1.30$ when averaged over time. The individual vane lift is also periodic with the period of the propeller blade passage. One can conclude that both a better SRV thrust and lift would have been obtained at $J = 1.60$ when vane + on the down-going blade side would have been removed.

A number of conclusions can be drawn from these results on the design of SRV for a wing-mounted tractor-propeller. First of all, for SRV designed without wing the vane root pitch needs to be adjusted to prevent stall and obtain the design inflow angle. For this design the time-average vane thrust is dependent on the azimuthal position, leading to very effective and less effective or even ineffective vanes. The azimuthal variation is a result of the upwash of the wing, which depends on the wing loading. Since the wing loading is not constant in flight, the effectiveness of the vanes varies as well. Possibly for this design the vane root pitch needs to be adjusted again to prevent stall with a different wing loading. The best overall SRV performance can only be obtained when the individual vanes can be turned depending on wing loading and advance ratio. For certain combinations of advance ratio and wing loading, vanes on the down-going blade side might be turned to remove any loading and reduce their corresponding drag. In general it is recommended to look into a design optimisation of each vane individually depending on the azimuthal position. Since the upwash of the wing does not result in a large change in the shape of the radial distribution of swirl angle as was seen in Subsection 6.3.2, this may not result in better SRV performance and individual vane turning may be sufficient.

6.3.4. EFFECT OF SRV ON THRUST AND LIFT FORCE BALANCE

For a wing-mounted tractor-propeller conclusions on SRV performance can only be drawn from the complete force balance of thrust and lift. Tables 6.7 and 6.8 present this force balance at $J = 1.60$ and $J = 1.30$ respectively relative to the propeller thrust without SRV $T_{p_{\text{without}}}$ for the thrust balance and relative to the wing lift without SRV $L_{w_{\text{without}}}$ for the lift balance. The ratio of $T_{p_{\text{without}}}$ and $L_{w_{\text{without}}}$ is mentioned in the respective table caption. At $J = 1.60$ the addition of SRV results in a slight increase in propeller thrust, an increase in wing drag and a slight decrease in nacelle drag. Together with the net SRV thrust, the overall thrust balance is slightly lower than without SRV and thus it is better from a thrust point of view to have no SRV. However, the difference is only 1/4 of the drag resulting from vane + and thus when this vane would be removed or turned, the SRV are already beneficial. At $J = 1.30$ the net thrust is 1.84 % higher due to the SRV, although the wing drag is higher. For the configuration with wing the net increase in propulsive efficiency by the addition of SRV can be defined as:

$$\Delta\eta_{p_{\text{net}}} = \frac{J}{\rho_{\infty} n^2 D_p^4} \left(\left(\frac{T_{\text{net}}}{C_p} \right)_{\text{with}} - \left(\frac{T_{\text{net}}}{C_p} \right)_{\text{without}} \right) \quad (6.1)$$

where T_{net} is the net thrust and the subscripts *with* and *without* denote the properties with and without SRV respectively. The resulting increase in propulsive efficiency for $J = 1.60$ and $J = 1.30$ are $\Delta\eta_{p_{\text{net}}} = -0.14 \%$ and $\Delta\eta_{p_{\text{net}}} = +1.00 \%$ respectively.

Table 6.7: Balance of forces of all aerodynamic surfaces in thrust and lift direction relative to the propeller thrust and wing lift without SRV for $J = 1.60$ ($T_{p_{\text{without}}} = 13.59 \% L_{w_{\text{without}}}$).

item	Thrust without [% $T_{p_{\text{without}}}$]	Thrust with [% $T_{p_{\text{without}}}$]	Lift without [% $L_{w_{\text{without}}}$]	Lift with [% $L_{w_{\text{without}}}$]
Propeller	100.00	100.24	0.85	0.86
SRV	-	0.88	-	0.05
Wing	-31.40	-32.69	100.00	100.20
Nacelle	-1.34	-1.22	6.30	6.41
Net	67.26	67.21	107.15	107.52

At $J = 1.60$ the lift of the propeller, wing and nacelle increases slightly and the lift of the SRV is very small but positive, leading to a small increase in overall lift of 0.35 % by the addition of SRV. At $J = 1.30$ the SRV lift is small but negative and the SRV results in a decrease in wing lift, resulting in a overall decrease in lift of 0.55 %.

It is difficult to draw definite conclusions on SRV for a wing-mounted propeller from these results: In general it is seen that the wing results in a negative shift of the net propulsive efficiency increase by the addition

Table 6.8: Balance of forces of all aerodynamic surfaces in thrust and lift direction relative to the propeller thrust and wing lift without SRV for $J = 1.30$ ($T_{p_{\text{without}}} = 31.07 \% L_{w_{\text{without}}}$).

item	Thrust without [% $T_{p_{\text{without}}}$]	Thrust with [% $T_{p_{\text{without}}}$]	Lift without [% $L_{w_{\text{without}}}$]	Lift with [% $L_{w_{\text{without}}}$]
Propeller	100	100.12	0.98	0.99
SRV	-	3.32	-	-0.15
Wing	-12.64	-14.84	100.00	99.48
Nacelle	-0.98	-0.63	6.54	6.61
Net	86.38	87.97	107.52	106.93

of SRV. Careful optimisation of SRV taking the wing into account as well as the lift as a constraint will most likely result in a performance benefit, since already with this non-optimised design an increase in thrust or lift can already be found depending on the advance ratio. However, two caveats should be mentioned: First, the ratio of propeller thrust to wing and nacelle drag is far from a realistic ratio in cruise. Based on Obert [58] it is estimated that the sum of wing and nacelle drag accounts for 67 % of the total drag of the Fokker F50. Note that the nacelle drag is corrected for its smaller size compared to the original nacelle. At $J = 1.60$, which represents the cruise condition, the wing and nacelle account for 32.74 % of the propeller thrust when no SRV are present. The low undisturbed speed V_∞ combined with the blade angle β_{75} are the main factors contributing to this unrealistic ratio of propeller thrust and wing and nacelle drag. V_∞ and β_{75} are chosen because for these values the propeller performance and slipstream are validated and because they allow for easier experimental testing. The second caveat is the propeller turning direction: This outboard-up configuration is worse for wing drag than an inboard-up configuration when no SRV are present as discussed in Section 1.3, and thus conclusions drawn for an inboard-up configuration will likely be worse for the performance benefit of SRV. The inboard-up configuration is not tested since it is though better to first look into optimisation of the SRV design for the configuration with wing, taking into account all discovered effects of the wing on the SRV and visa versa.

Not only the total wing lift and drag changes by the addition of SRV, also interesting changes in the lift and drag spanwise distribution occur. Figure 6.9 shows the time-average lift and drag distribution of the wing only (without propeller and nacelle) as well as the whole configuration without and with SRV at $J = 1.60$. A similar result for $J = 1.30$ is given in Figure B.5. First consider the effect of the propeller slipstream: The tangential velocity component in the propeller slipstream leads to a reduction of angle of attack and thus lift inboard and an increase in angle of attack and thus lift outboard of the propeller. The increased axial velocity in the slipstream increases the lift on both sides. The net result is a small decrease inboard and large increase of lift outboard of the propeller. This is in line with experimental results of a wing-mounted tractor-propeller configuration of Veldhuis [16]. In line with the discussion in Section 1.3, the drag increases inboard and decreases outboard due to the tangential velocity component when lift is present. The axial velocity component leads to an increase in drag on both sides. The net result is a large increase in drag inboard and smaller decrease in drag outboard of the propeller. The viscous part of the drag increases on both sides, since its dependence on the angle of attack is small and thus it is almost only influenced by the increased velocity.

Now consider the effect of the SRV: These reduce the tangential velocity component resulting in a smaller reduction of angle of attack inboard and smaller increase in angle of attack outboard of the propeller, resulting in a less decreased and increased lift respectively. SRV may also affect the axial velocity component but in the next section it will be shown that for this advance ratio the axial velocity is not affected much. In the drag distribution, one would expect a less increased drag inboard and less decreased drag outboard of the propeller. Only the latter occurs and the inboard peak in drag remains almost unaltered. Apparently the increase in resultant force on the inboard side leads to an increase in drag that exactly compensates the decrease in drag due to the turning of the resultant force vector. This is also the case for $J = 1.30$.

In general, one can conclude that SRV reduce some of the effects of the propeller on the wing lift and drag distribution, resulting in a smaller deviation from the wing loading without propeller. If the same conclusions can be made for an inboard-up configuration, then it could be concluded that SRV reduce but not completely remove the asymmetric effects and the corresponding trimming drag associated with co-rotating propellers.

Figure 6.10 presents the pressure coefficient on the surface of the wing and aft nacelle without and with SRV at an instance in time. The shear lines are indicated as well. On the inboard side, one can clearly see the aft position of the suction peak, resulting in the large peak in drag. The effect of the SRV is not well visible. On the outboard side the suction peak is very close to the leading edge, resulting in the minimum in drag. When the SRV are present, this suction peak reduces considerably, resulting in a smaller minimum in drag.

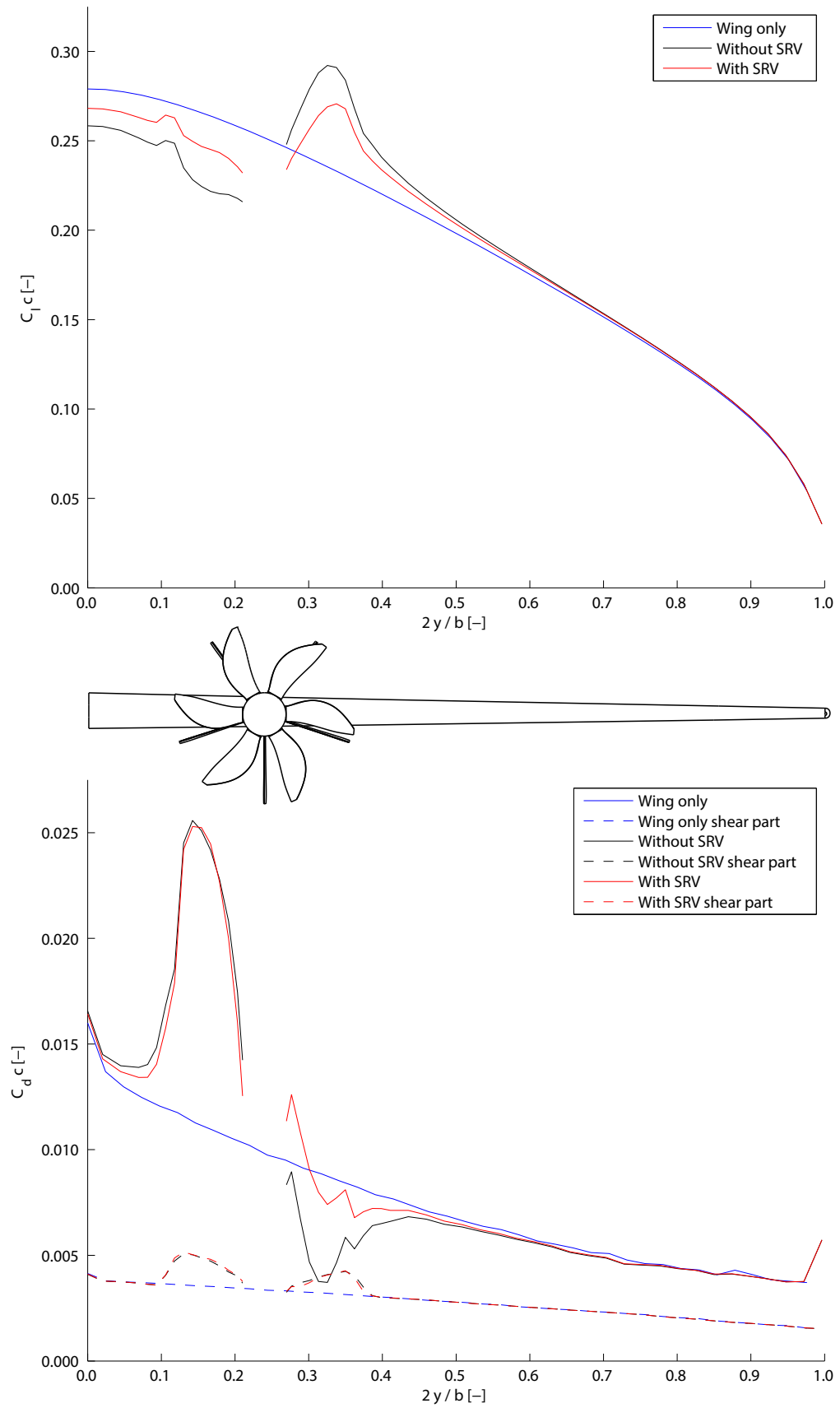


Figure 6.9: Wing lift and drag distribution defined with undisturbed conditions for the wing only and the wing with propeller and nacelle with and without SRV at $J = 1.60$.

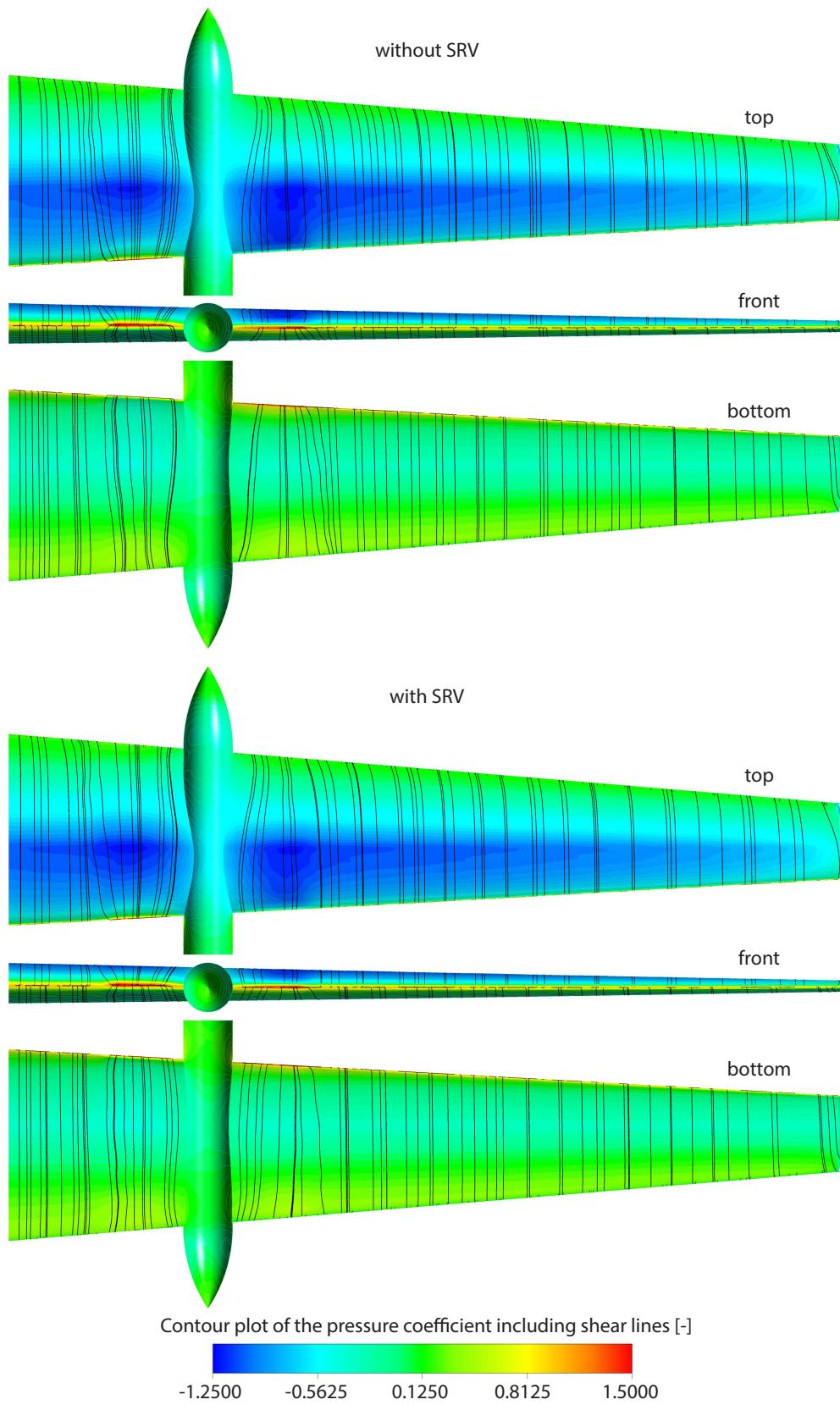


Figure 6.10: Contour plot of the pressure coefficient $C_p = \frac{p - p_\infty}{\frac{1}{2} \rho_\infty V_\infty^2}$ defined with undisturbed conditions on the wing and aft part of the nacelle with and without SRV at $J = 1.60$.

6.3.5. DESCRIPTION OF THE SLIPSTREAM

This subsection describes the effect of SRV on the slipstream of the wing-mounted propeller. Figure 6.11 indicates the planes behind the propeller where flow quantities are extracted. These correspond to those in Subsection 5.3.3. Figures 6.12, 6.13, 6.14 and 6.15 give the time-average distribution of axial velocity, tangential velocity, swirl angle and total pressure over four azimuthal varying radial lines in these planes at $J = 1.60$. The corresponding figures at $J = 1.30$ are given in Appendix B.

When no SRV are present, the axial velocity at $0.45R_p$ shows little azimuthal variation. At $1.35R_p$ the axial velocity increases for the radial lines above the wing and decreases for the radial lines below the wing. On the plane cutting the wing at $2.00R_p$ the axial velocity increases for all azimuthal positions, more above than below the wing and more inboard than outboard. The propeller slipstream is displaced by the wing leading to an increase in axial velocity, and the loading on the wing causes a larger flow acceleration on the upper side of the wing. The larger chord and corresponding thickness on the inboard side than on the outboard side of the wing explains the spanwise difference of axial velocity. At $3.24R_p$ the axial velocity is reduced again to slightly higher values than at $1.35R_p$. The addition of SRV does not lead to large changes in the axial velocity development, in agreement with the results without wing of Subsection 5.3.3. Only minor changes close to the nacelle and slipstream boundary are visible, most probably caused by deformation of the propeller root and tip vortices.

The tangential velocity is shown in Figure 6.13. When no SRV are present, at $0.45R_p$ an azimuthal variation is clearly visible: Due to the upwash induced by the wing, the tangential velocity is reduced inboard and increased outboard of the propeller. Closer to the wing at $1.35R_p$ the local effect of the presence of the wing results in an upward velocity component above and downward velocity component below the wing, reducing the tangential velocity inboard above and outboard below and increasing the tangential velocity outboard above and inboard below the wing. The large flow acceleration above and below the wing at $2.00R_p$ results in a reduction of tangential velocity, especially near the wall. Behind the wing the combined local effect of the curvature of the airfoil and the downwash is clearly visible above the wing, leading to a large increase and decrease inboard and outboard of the propeller respectively. The addition of SRV does not result in changes in tangential velocity upstream of the SRV. Downstream of the SRV at $1.35R_p$ the reduction of the tangential velocity is clearly visible, in line with the results without wing. More downstream, the reduction of the tangential velocity remains presents but the difference becomes less, mainly due to the more effective swirl recovery of the wing when no SRV are present.

The development of the swirl angle in Figure 6.14 is the result of the combined effects on the axial and tangential velocity components. The swirl angle is mainly interesting to investigate the effect of the SRV. Upstream of the SRV no effect can be noticed while downstream at $1.35R_p$ the swirl recovery is clearly visible. Since the effect of the SRV on the axial velocity is small, the effect on the swirl angle is very similar to that on the tangential velocity component. Again the more effective swirl recovery by the wing can be noticed when no SRV are present, resulting in a decreased difference of swirl angle with and without SRV behind the wing.

Only minor changes in total pressure distribution are visible in Figure 6.15. The main effects are near the wall and near the slipstream boundary, resulting from the reduction of total pressure in the boundary layer and the displacement of the slipstream boundary by the wing. The effect of the SRV is very little, except for behind the wing at $3.24R_p$ where larger variations are visible especially on the inboard side. These are related to the local deformation of the vortex structure discussed in the next paragraph.

Figure 6.16 shows a side and rear view of an isosurface of the vorticity magnitude behind the propeller. The axial vorticity component is indicated by the colour and both the isosurface for the configuration without and with SRV are shown. When no SRV are present, the only vortex structures that can be seen are the propeller blade tip and root vortices, the wing tip vortex and a sheet of vorticity trailing from the wing. The difference in axial velocity above and below the wing is clearly visible in the more downstream displaced tip vortices above the wing compared to their corresponding vortices below the wing. In the side view an upward motion of the vortex structure upstream of the wing and a downward motion downstream of the wing is visible. This is due to the upwash and downwash induced by the wing respectively. In the rear view, the vortex structure above the wing is clearly sheared more inboard with respect to the vortex structure below the wing. This shearing of the slipstream boundary is also indicated by Samuelsson [18] and is a result of the spanwise flow over the wing by its loading. This is for instance also clearly visible in the surface shear lines in Figure 6.10. The effect of the SRV on the vortex structure is very similar to the effect when no wing is present and is already described in Subsection 5.3.3. The effect of the wing and SRV combined lead to a very large deformation of the original propeller vortex structure.

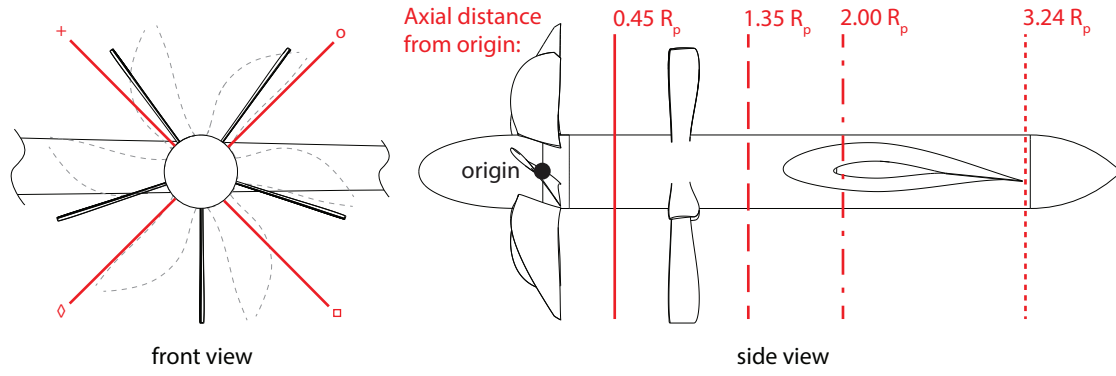


Figure 6.11: Definition of radial lines in four planes downstream of the propeller in a front and side view of the propeller-SRV-wing configuration.

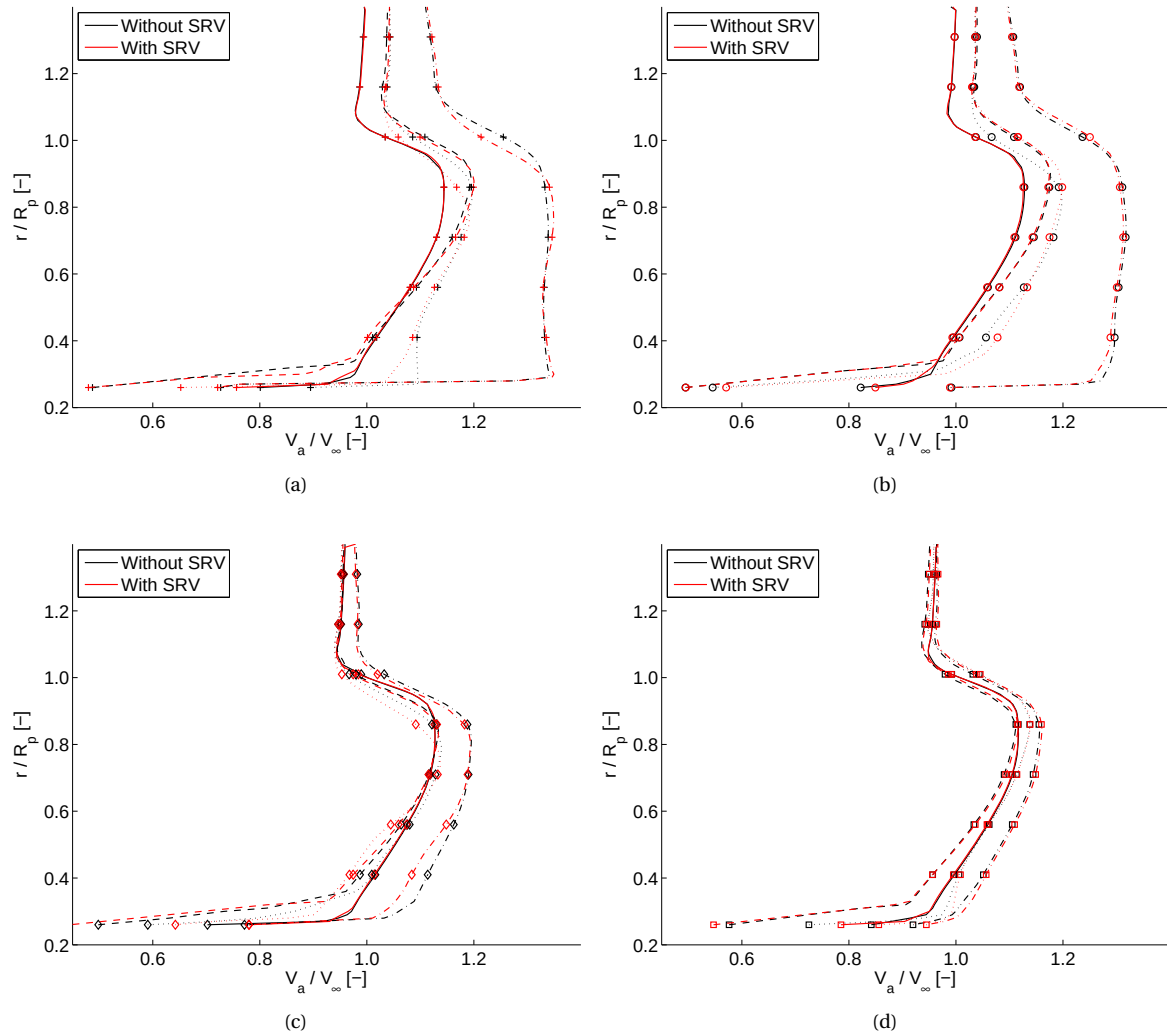


Figure 6.12: Time-average axial velocity component without and with SRV on four radial lines in four planes downstream of the propeller as defined in Figure 6.11 for the propeller-SRV-wing configuration.

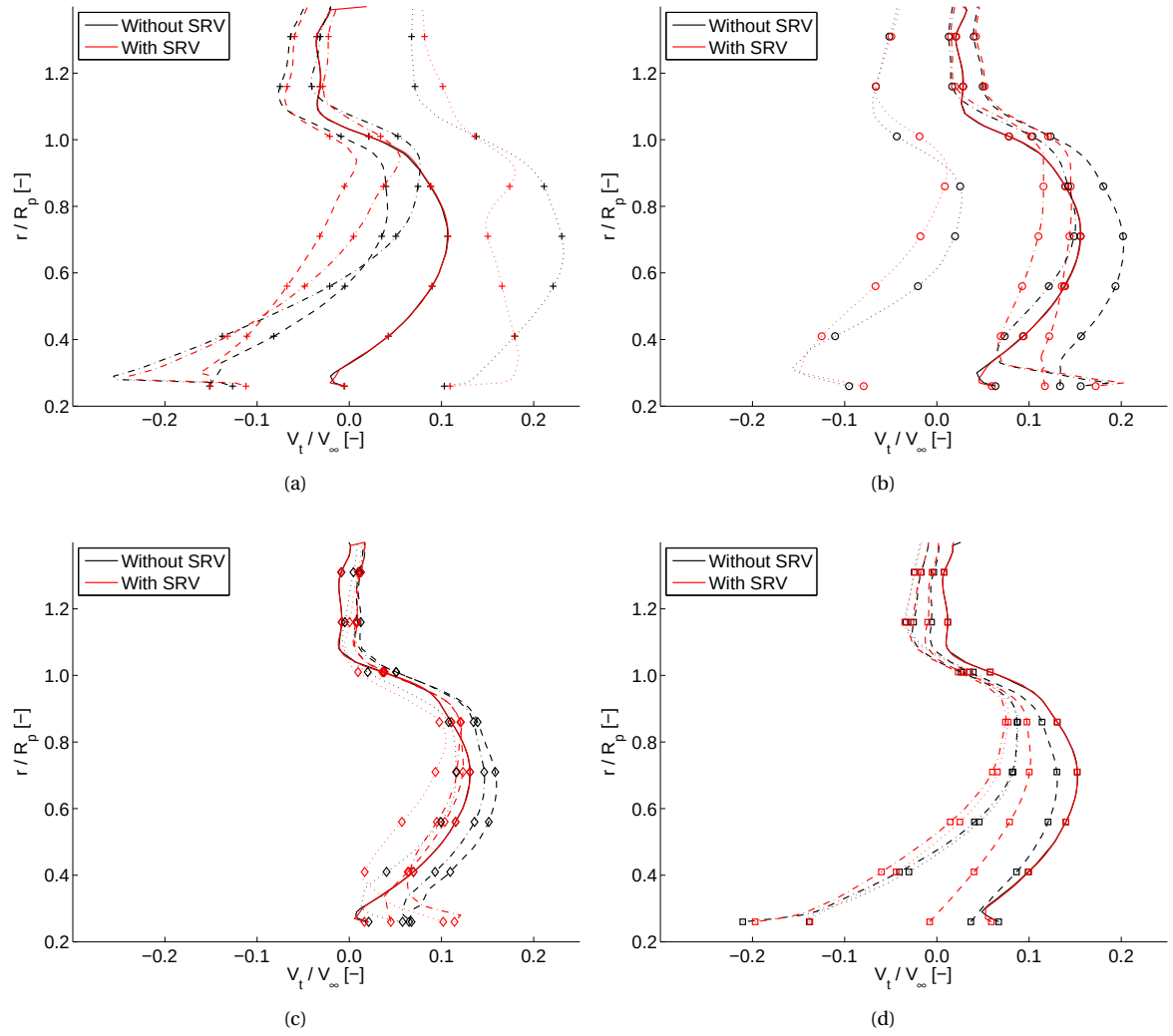


Figure 6.13: Time-average tangential velocity component without and with SRV on four radial lines in four planes downstream of the propeller as defined in Figure 6.11 for the propeller-SRV-wing configuration.

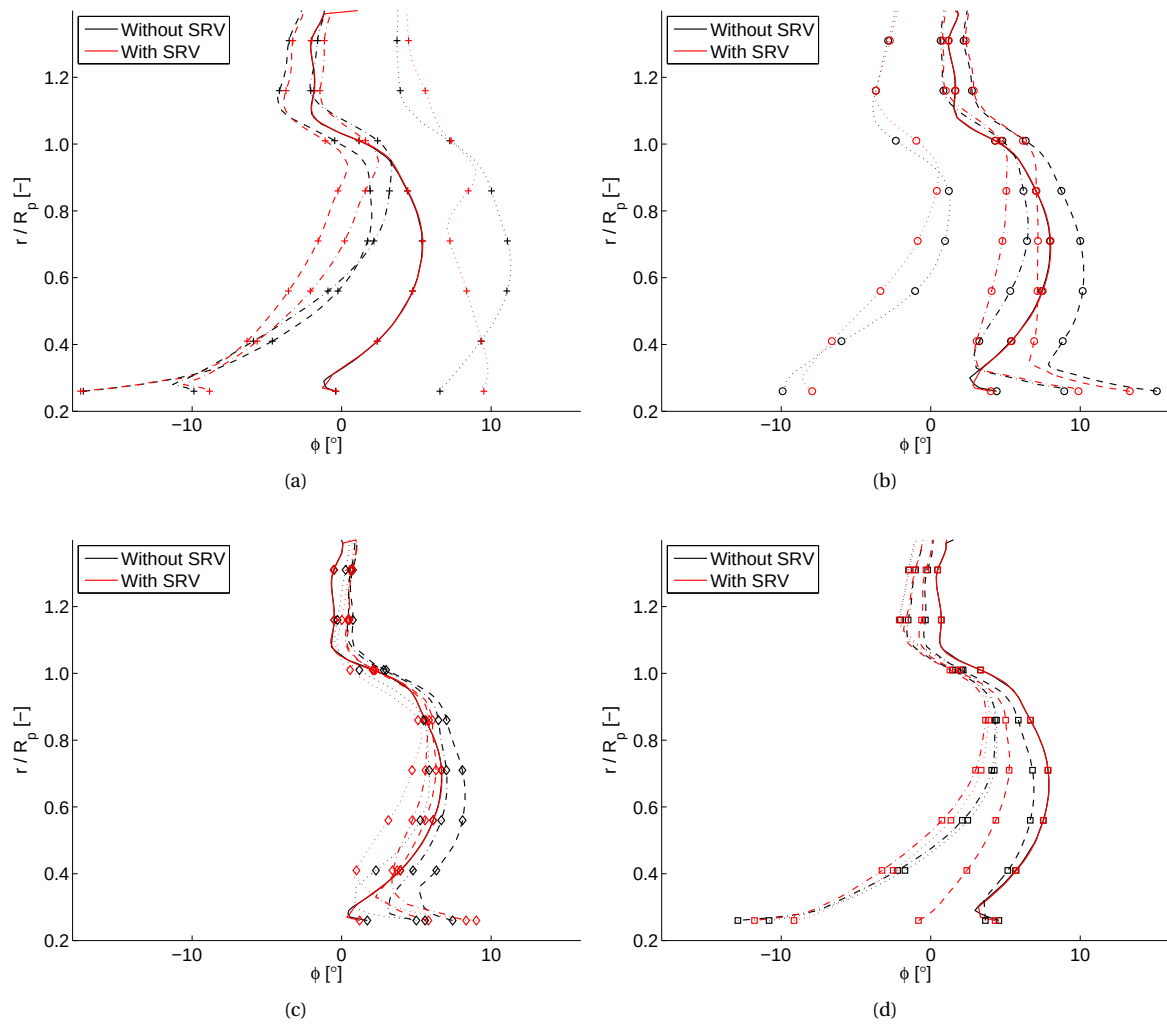


Figure 6.14: Time-average swirl angle without and with SRV on four radial lines in four planes downstream of the propeller as defined in Figure 6.11 for the propeller-SRV-wing configuration.

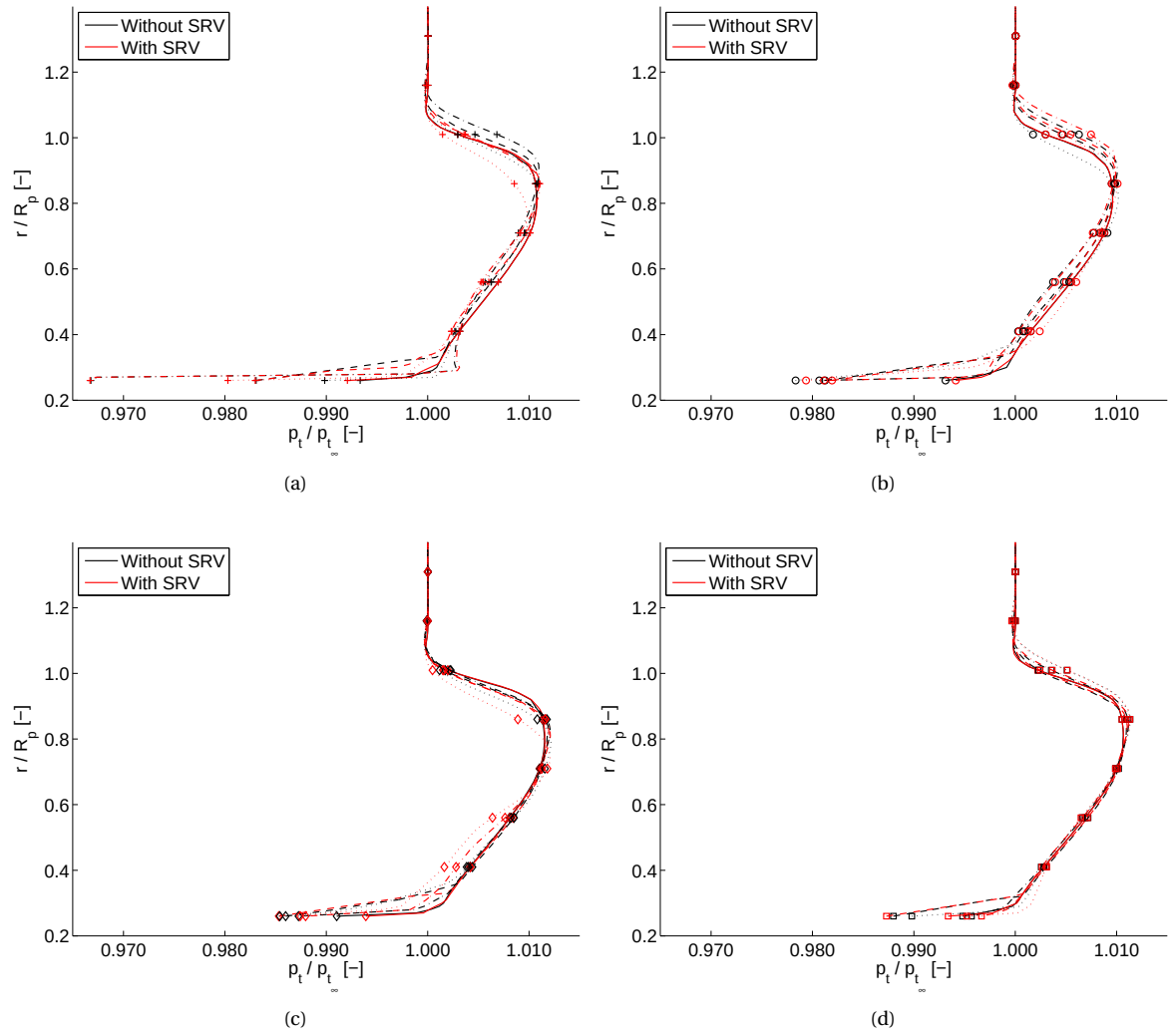


Figure 6.15: Time-average total pressure without and with SRV on four radial lines in four planes downstream of the propeller as defined in Figure 6.11 for the propeller-SRV-wing configuration.

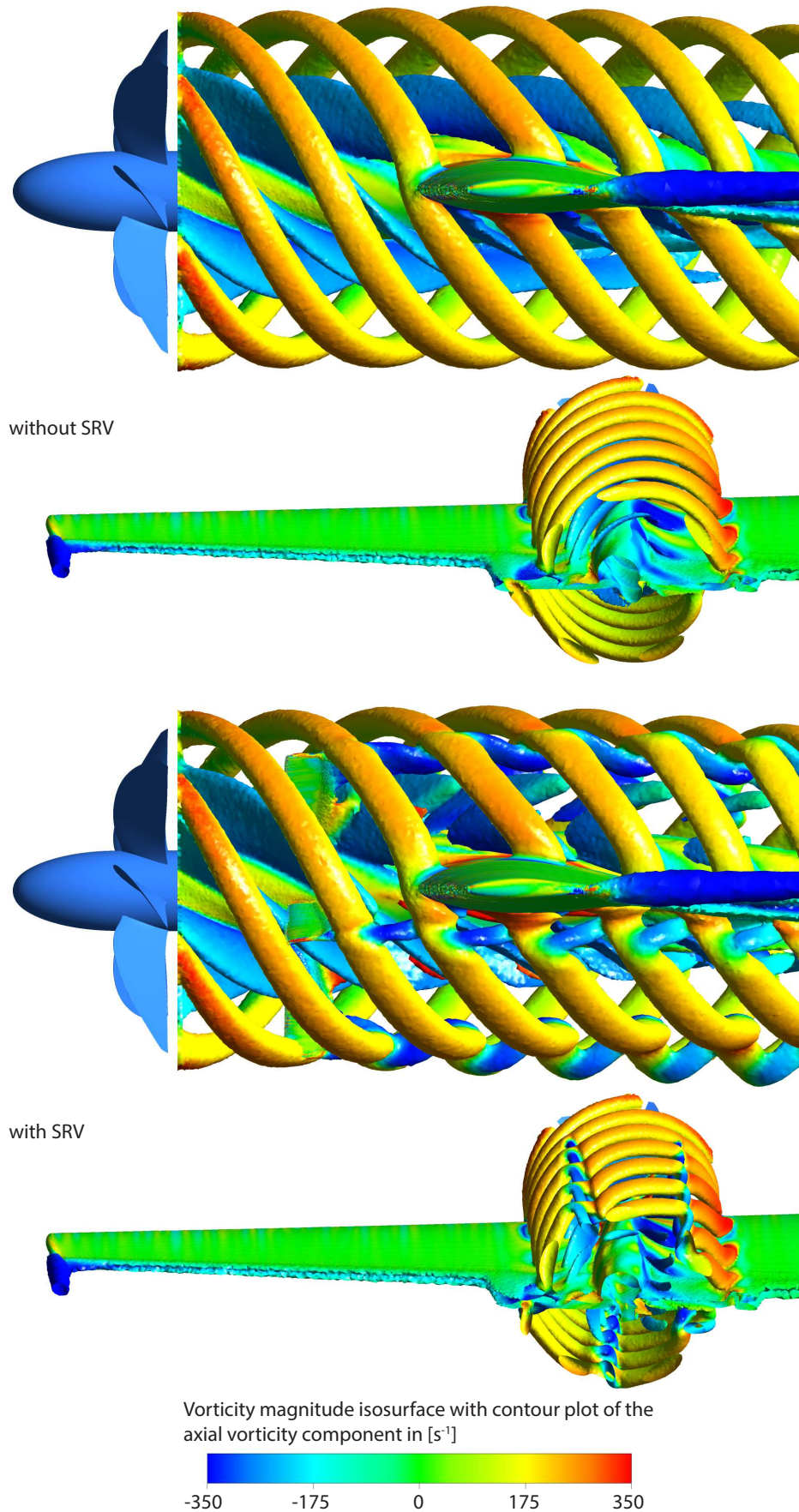


Figure 6.16: Side and rear view isosurfaces of the vorticity magnitude behind the wing-mounted APIAN propeller without and with SRV designed for cruise including a contour plot of the axial vorticity component at an instance of the propeller blade passage at $J = 1.60$.

CONCLUSIONS AND RECOMMENDATIONS

This final chapter presents the conclusions drawn throughout this thesis and provides recommendations for future work on swirl recovery vanes (SRV). The conclusions are stated in Section 7.1 and the recommendations are given in Section 7.2.

7.1. CONCLUSIONS

This work deals with the design and analysis of SRV for an isolated and a wing mounted tractor propeller. The objective of this research is to gain an improved understanding of the aerodynamic interaction between the propeller and swirl recovery vanes in an isolated configuration and wing-mounted tractor arrangement in the cruise condition and in a high-thrust condition. This study is realized by performing a series of transient Reynolds-averaged Navier-Stokes (RANS) CFD simulations and providing a comparison with existing experimental results of Custers and Elsenaar [22] for the isolated propeller and of Sinnige et al. [7] for the isolated propeller with SRV.

First a propeller CFD model of the six-bladed APIAN propeller is constructed for which appropriate solver settings are established and a mesh is selected by means of a mesh dependency study. In this study the effect of mesh volume and wall refinement is investigated independently. Volume refinement has a very small effect on the propeller performance and a considerable effect on the flow quantities in the propeller slipstream. Especially in regions where large gradients are present, i.e. the blade tip vortex region, differences occur by volume refinement resulting in larger temporal and spatial gradients. This is especially noticeable in an improved capturing of the propeller blade tip and root vortex structure. The effect of wall refinement on the propeller performance is considerable while the effect on the slipstream is negligible. Considering that the focus of this work is on SRV which are placed in the slipstream of the propeller, a compromise is found: For this mesh the slipstream agrees well with time-average experimental rake measurements and the propeller performance is adequate, with a deviation in thrust coefficient of -3.59% and power coefficient of $+2.95\%$ from experimental results. These deviations can be explained by the chosen wall refinement and the decambering effect by the fully turbulent modelling of the boundary layer. A representative advance ratio for the cruise and high-thrust condition is chosen and for these conditions the slipstream flow quantities are determined, which are necessary for the SRV design routine. An interesting flow phenomenon is discovered for this propeller: It is found that at low advance ratios a 3-D conical separation vortex originates at the leading edge of the propeller blades near the spanwise location where the blade sweep changes sign. This phenomenon described by Schüle in et al. [34] is most likely also present in the experiment, visible in a second suction peak in the chordwise pressure coefficient distribution near the blade tip.

For this research an SRV design is required. One of the options is an existing SRV design from the APIAN-INF (APIAN In Non-uniform Flow) test program in the DNW-LLF as described in Sinnige et al. [7]. This design is investigated by means of the APIAN-INF CFD model. For this design an SRV wall refinement is performed which shows a mesh dependency of the SRV thrust: A compromise is found between computational time and accuracy. Two different approaches are used for connecting the rotating region of the domain containing the propeller and stationary region containing the SRV. For the first approach only one propeller blade and vane are simulated by means of periodic boundary conditions and when the total number of blades and vanes are dissimilar, a profile transformation is performed at the interface of these two regions: In the profile

transformation method on the pitchwise boundaries of each region periodic boundary conditions are applied and across the interface the flow profile is stretched or compressed to connect the two dissimilar pitch regions of the domain. For the second approach the full propeller and SRV are modelled in a 360° domain and no special transformation technique is required across the interface. The effect of the profile transformation is investigated in a comparison with 360° domain performance results and it is found that the time-average SRV thrust is not affected and small differences in propeller thrust and power lead to small differences in the increase in propulsive benefit by the addition of SRV. Overall, it can be concluded that this method provides a fast first estimate of the SRV performance which can be used to test new designs fairly rapidly compared to the full 360° domain simulation. This latter simulation is necessary for an accurate description of the slipstream. For a condition in between the cruise and high-thrust, it is found that the propulsive efficiency increase by the addition of SRV is only $\Delta\eta_p = 0.57\%$ which is much lower than the design prediction of $\Delta\eta_p = 1.8\%$ by van Kuijk [21]. Therefore this design is not used in the remainder of the research and new SRV designs are proposed. PIV measurements in a plane spanned by the radial and rotation axis during a full propeller blade passage provide a comparison of the slipstream velocity components and vorticity and enables conclusions on the quality of resolving the flow in the CFD simulation for these specific solver settings and mesh. When no SRV are present, the wake structure is very similar, however the features in the PIV measurement are more distinct with sharper gradients and larger extrema. When SRV are present, the same conclusion can be drawn, but also qualitative differences occur further downstream, most probably due to a stronger interaction of the vortex structure trailing from the propeller and SRV. This simulation combined with the PIV measurements enables an extensive description of the structure of root and tip vortices induced by the propeller blades and swirl recovery vanes.

In this research an SRV design procedure is introduced for the design of SRV behind an isolated propeller with nacelle. For such a configuration, the SRV thrust should be maximised in order to maximise $\Delta\eta_p$. The question is raised whether it is better to optimise the SRV design for maximum SRV thrust or zero downstream swirl, or whether these are the same objectives. This question is investigated in a simplified analytic analysis of a single zero-twist elliptic vane mounted on an infinite wall under an inflow with constant swirl angle. It is found that if no profile drag and only induced drag of the vane is considered, the maximum in vane thrust by changing the vane pitch angle also removes all downstream swirl. When profile drag is considered, the maximum shifts such that some downstream swirl remains for a given symmetric airfoil. When camber is considered, the largest maximum in vane thrust is found when the airfoil is at zero effective angle of attack, assuming the minimum in profile drag is found at this condition as well. For this condition the downstream swirl is zero again. However, for large swirl angles this requires unrealistic airfoil properties. Therefore it is better to optimise for maximum SRV thrust since it is likely that this most optimum SRV which also removes the downstream swirl may not be realistically found. An SRV design procedure is introduced consisting of an SRV analysis tool and optimisation routine. The SRV analysis tool is based on non-linear lifting line theory of Phillips and Snyder [45] adapted by the author for non-uniform inflow and for the effect of the nacelle and makes use of the airfoil analysis tool XFOIL by Drela [42] for the airfoil properties. In an optimisation routine the SRV planform and airfoil parameters are optimised for maximum SRV thrust for a velocity inflow field extracted from an isolated propeller CFD simulation. Four designs are obtained for the inflow field extracted from the propeller CFD model: Design 1 is optimised for the cruise condition with a constraint on stall for the high-thrust condition. Design 2 is optimised for the high-thrust condition with a constraint on the cruise condition for zero or positive SRV thrust. Design 1 and 2 represent designs where the swirl recovery vanes have a fixed pitch in flight. Design 3 is optimised for the cruise condition without constraint condition. The performance in the high-thrust condition is investigated by rotating the vanes to reach the highest efficiency benefit. Design 4 is similar to design 3 but now optimised for the high-thrust condition. All designs consist of five vanes with the same radius as the propeller. The effect of cropping and of the number of vanes on the propulsive efficiency is investigated as well for the objective of design 1. Table 7.1 summarises the predicted propulsive efficiency benefit for each design in the cruise and high-thrust condition. Cropping the SRV to avoid the largest velocity gradients in the propeller tip vortex region of the slipstream leads to a reduction of $\Delta\eta_p$ of almost a factor two, while increasing the number of vanes to seven or nine leads to a considerable increase. Although for at least up to nine vanes, more vanes is better, the choice for five vanes has been made for a number of reasons: First, preliminary investigations suggested a lesser increase in propulsive efficiency. Second, the APIAN-INF SRV consist of five vanes and is cropped, and thus keeping the number of vanes equal allows for a better qualitative investigation into the effect of cropping. This design consists of five vanes and not the same as the number of propeller blades to avoid noise by aerodynamic interference. Third, mesh size and thus computational cost increases with the number of vanes for 360° domain simulations. In retrospect

Table 7.1: Predicted and time-average simulated propulsive efficiency increase by the addition of SRV $\Delta\eta_p$ including the effect of cropping and number of swirl recovery vanes.

SRV design	predicted $\Delta\eta_p$ [%]		simulated $\Delta\eta_p$ [%]	
	cruise	high-thrust	cruise	high-thrust
1	0.50	3.49	0.39	2.62
1 - cropped	0.28	1.83	-	-
1 - 7 vanes	0.52	4.18	-	-
1 - 9 vanes	0.60	4.53	-	-
2	0.00	3.84	0.20	3.07
3	1.14	1.77	-	-
4	0.00	3.84	-	-

a design of more than five vanes might have been a better choice.

In the propeller-SRV CFD model which consists of the APIAN propeller and nacelle with SRV, design 1 and 2 are used in simulations by means of profile transformation for a range of advance ratios to validate the predictions from the SRV analysis tool. For design 1 also full 360° domain simulations are performed for the cruise, high-thrust and intermediate medium-thrust condition to estimate the error introduced by the profile transformation and enable an extensive description of the slipstream. The effect of the SRV on the propeller performance is found to be small, influencing the time-average propeller propulsive efficiency by a reduction in the order of 0.1 %. Similar to the APIAN-INF CFD model, the effect of the profile transformation on the time-average SRV thrust is negligible and the effect on the time-average propeller performance is small. For SRV design 1 and 2, Table 7.1 summarises the simulated propulsive efficiency gain in the cruise and high-thrust condition. The differences between the simulation and prediction in the cruise condition can be explained by the effect of the profile transformation and differences between the simulation and prediction by XFOIL of the airfoil lift and drag curve. The overprediction of $\Delta\eta_p$ by the SRV analysis tool for the high-thrust condition is most likely caused by the combination of the differences in airfoil properties, the chord effect for which Jones [51] introduces a correction of the lifting line theory and the yet unknown effect of the radial flow. In general the simulation results show that SRV lead to an increase in propulsive efficiency by increasing the system thrust over a wide range of advance ratios, with minor effect on the system power. Design 1 proves that it is possible to increase the propulsive efficiency of an operating point close to the point of maximum propeller propulsive efficiency. Design 2 shows that if a larger increase in propulsive efficiency at low advance ratios is desired, the design can be changed at the cost of propulsive efficiency benefit at higher advance ratios, for a fixed SRV pitch design. The effect of SRV on the slipstream is investigated by means of a comparison of the 360° domain simulation results and the results from the propeller CFD model. It is found that upstream of the SRV the effect on the velocity and total pressure in the slipstream is very small, in line with the small effect of the SRV on the propeller performance. Downstream of the SRV, somewhat less than half of the swirl is recovered on average, in line with results from Yamamoto [11] for the SRV design from NASA Lewis Research Center's Advanced Turboprop Project. Downstream of the SRV the axial velocity and total pressure are only slightly affected by the SRV for the cruise condition and considerably affected for the high-thrust condition: An expansion of the slipstream boundary can be noticed in the radial distribution of circumferential-average axial velocity and total pressure, which is the result of the interaction of propeller blade and vane tip vortices. An extensive description of this interaction is given, leading to deformation of the regular helical structure of propeller blade tip vortices without SRV, resulting in radial displacement of the vortex structure outside of the slipstream boundary without SRV.

The conclusions on SRV thus far are only applicable to isolated propeller nacelle configurations, i.e. fuselage mounted propellers. In the last part of this research a slightly altered scaled Fokker 50 wing is introduced behind the propeller and SRV for the propeller-SRV-wing CFD model. A wing wall mesh is obtained by a mesh dependency study on the isolated wing, verified by a comparison of the lift and drag spanwise distribution with a prediction by the vortex lattice program AVL [55] combined with XFOIL [41]. The SRV in this research are designed for the slipstream of the isolated propeller. The loading on a wing results in an upwash upstream of the wing. This results in a change of the slipstream and thus in a deviation from the design condition. Initially, it was thought that this effect would be small enough such that an SRV designed for an isolated propeller would still work well when a wing is present. However, it is found that the addition of the upwash to the propeller slipstream results in an inflow field that is different for each vane, decreasing the swirl angle on the propeller blade downgoing side and increasing the swirl angle on the propeller blade upgoing side. Be-

cause of this the performance of SRV design 1 is heavily degraded due to separation on the pressure side of a vane on the propeller blade downgoing side in the cruise condition and separation on the suction side of two vanes on the propeller blade upgoing side in the high-thrust condition. For the remainder of this research a change in the SRV design is made by turning each vane over an angle to obtain the time- and radial-average design inflow angle in the cruise condition and this design is kept constant for the other conditions. In the high-thrust condition still no converged solution is found due to flow separation and thus this condition is omitted and only simulations for the cruise and the medium-thrust condition are performed. For the propeller without SRV, the wing leads to a considerable increase in propeller thrust and power coefficient which affects the propeller propulsive efficiency. This can be explained by two effects: First, the blade angle of attack is increased by a high pressure region in front of the wing according to Veldhuis et al. [56]. Second, the upwash by the loading on the wing results in an increase in propeller thrust and power just like for a propeller under an incidence angle as is shown by Ortun et al. [23] and Stuermer and Rakowitz [57]. When SRV are present, the propeller thrust and power coefficient increase slightly more. The SRV affect the propeller performance directly as shown for the propeller-SRV CFD model without wing and indirectly through an effect of the SRV on the wing loading and corresponding upwash. For the cruise condition the increase in propulsive efficiency by the addition of SRV without considering differences in wing drag is found to be $\Delta\eta_p = 0.93\%$, which is considerably higher than without wing, mainly due to the increased propeller propulsive efficiency, but partly by increased SRV thrust as well. $\Delta\eta_p$ is equal to 2.14 % for the medium-thrust condition, which is very similar to the value without wing. The individual vane thrust varies considerably though, including net drag for one of the vanes on the propeller downgoing side in the cruise condition. For a wing-mounted tractor-propeller conclusions on SRV performance can only be drawn from the complete force balance of thrust and lift of the propeller, SRV, wing and nacelle. Considering the drag of all components, the net increase in propulsive efficiency by the addition of SRV is found to be $\Delta\eta_{p_{net}} = -0.14\%$ for the cruise and $\Delta\eta_{p_{net}} = 1.00\%$ for the medium-thrust condition with a net increase in lift of 0.35 % and net decrease in lift of 0.55 % respectively. However, when the drag producing vane is removed in the cruise condition, already a positive value for $\Delta\eta_{p_{net}}$ is found. In general, the wing results in a negative shift of propulsive efficiency benefit of SRV because of the reduced swirl recovery by the wing due to the reduced swirl behind the SRV. Careful optimisation of SRV taking the wing into account as well as the lift as a constraint will most likely result in a performance benefit, since already with this non-optimised design an increase in thrust or lift can be found depending on the advance ratio. The propeller slipstream greatly affects the wing lift and drag distribution by its increased axial velocity and introduced swirl. Only an outboard-up configuration is considered. It is concluded that SRV reduce some of the effects of the propeller on the wing lift and drag distribution by a reduction of the swirl, resulting in a smaller deviation from the wing loading without propeller. If the same conclusions can be made for an inboard-up configuration, then it could be concluded that SRV reduce but not completely remove the asymmetric effects and the corresponding trimming drag associated with co-rotating propellers. The wing greatly affects the slipstream and vortex structure downstream of the SRV. These effects are very similar whether SRV are present or not, taking into account the differences in slipstream and vortex structure by the SRV without wing.

7.2. RECOMMENDATIONS

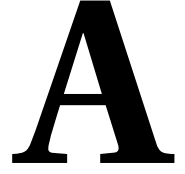
From this research a number of recommendations result for future research. Since the wing has a large effect on the performance of SRV, the recommendations are split it two. First a number of recommendations are given on the design of SRV for propeller configurations without a wing:

- For the SRV analysis tool, it is recommended to look into the chord effect for which Jones [51] introduces a correction of the lifting line theory, or apply vortex lattice theory with similar adaptations for non-uniform inflow and the nacelle effect.
- Two variable pitch SRV designs have been proposed, SRV design 3 and 4, optimised for the cruise and high-thrust condition. For a variable pitch design it is recommended to use a certain weighting factor to take both the loading in the cruise and the high-thrust condition into account, to arrive at a more feasible design with better high-thrust performance and of larger chord than the design for cruise, and with better cruise performance than the design for high-thrust.
- The fixed pitch SRV design 1 encountered problems with slight flow separation for both low and high propeller thrust for the propeller-SRV CFD model due to the large variation of swirl angle. These problems can be solved in two ways: First, it is recommended to look into variable pitch designs in order to

avoid flow separation. Second, it is recommended to look into airfoils with better stall characteristics. The range of airfoils considered in this research only consists of the whole NACA 4 series and thus improvements are most likely possible for a fixed-pitch SRV design. The application of leading edge droop may be considered.

In this thesis the SRV are designed without a wing in mind. The effect of the wing on such a design is investigated. From this investigation recommendations can be made on the design of SRV for a wing-mounted tractor-propeller:

- The wing induces an upwash at the location of the SRV. Since each vane experiences a different time-average flow field due to this upwash, ideally one would find a different design for each vane. This may improve the propulsive efficiency gain compared to the simpler option that is applied in this research to turn each vane individually over the temporal- and radial-average change in swirl angle.
- Since the effect of the wing upwash on the SRV inflow field varies with advance ratio and with wing loading and thus varies in flight, a variable pitch SRV design is recommended where the pitch of each vane is adjusted individually. The necessity of this recommendation can be seen for the high-thrust condition for which still no converged solution is found due to flow separation after turning the vanes individually for the cruise condition.
- One should also look into the effect of the downstream location on the SRV performance in order to place the SRV closer to the propeller and reduce the upwash effect of the wing on the SRV inflow.
- Since it was found that the vane thrust is very much dependent on the azimuthal position, and depending on the condition and azimuthal position a vane may produce net drag, one should look into SRV designs of which the vanes are not placed rotationally periodic.
- In general, a design procedure for SRV should include the wing for instance by an additional lifting line and optimise for combined SRV and wing maximum thrust with a constraint on the net lift. This may lead to SRV designs that deviate from designs for isolated propellers, more focussed on providing the optimal inflow for the wing in order to reduce the wing drag. For the investigated outboard-up configuration, the maximum in the wing drag distribution on the inboard side of the propeller that remains unaltered with or without SRV, may be reduced by an improved inflow.



NEWTON'S METHOD AND LINEARIZED INITIAL ESTIMATE

In order to solve the system of non-linear equations from Equation (4.35) Newton's method is used. This appendix explains the implementation of this method in the SRV Analysis tool. The method is slightly modified from Phillips and Snyder [45] to allow for non-uniform inflow. The local angle of attack of panel k can be found from:

$$\alpha_k = \tan^{-1} \left[\frac{(\mathbf{v}_{\text{in}_k} + \sum_{j=1}^N G_j (\mathbf{v}_{jk} + (\mathbf{v}_{jk})_{\text{im}})) \cdot \mathbf{u}_{n_k}}{(\mathbf{v}_{\text{in}_k} + \sum_{j=1}^N G_j (\mathbf{v}_{jk} + (\mathbf{v}_{jk})_{\text{im}})) \cdot \mathbf{u}_{a_k}} \right] \quad (\text{A.1})$$

where $\mathbf{u}_{n_k}$ and $\mathbf{u}_{a_k}$ are unit vectors normal and tangent to the chord of panel k . For the initial estimate the following approximation for small angles of attack is made to find the local lift coefficient:

$$C_{l_k} = C_{l_{\alpha k}} (\alpha_k - \alpha_{L0k}) = C_{l_{\alpha k}} \left(\mathbf{v}_{\text{in}_k} \cdot \mathbf{u}_{n_k} + \sum_{j=1}^N G_j (\mathbf{v}_{jk} + (\mathbf{v}_{jk})_{\text{im}}) \cdot \mathbf{u}_{n_k} - \alpha_{L0k} \right) \quad (\text{A.2})$$

where $C_{l_{\alpha k}}$ is the section lift curve slope and α_{0k} the section zero-lift angle of attack of panel k . Inserting this equation in Equation (4.35) and ignoring second order terms results in the following linear system:

$$2|\mathbf{v}_{\text{in}_k} \times \boldsymbol{\zeta}_k| G_k - C_{l_{\alpha k}} \sum_{j=1}^N G_j (\mathbf{v}_{jk} + (\mathbf{v}_{jk})_{\text{im}}) \cdot \mathbf{u}_{n_k} = C_{l_{\alpha k}} (\mathbf{v}_{\text{in}_k} \cdot \mathbf{u}_{n_k} - \alpha_{0k}) \quad (\text{A.3})$$

The resulting dimensionless vortex strength vector $\mathbf{G}$ is the initial condition for Newton's method. The goal is to find a vector $\mathbf{G}$ such that the residual vector $\mathbf{R}$ goes to zero:

$$\mathcal{F}(\mathbf{G}) = \mathbf{R} \quad (\text{A.4})$$

where

$$\mathcal{F}_k(\mathbf{G}) = 2 \left| \left(\mathbf{v}_{\text{in}_k} + \sum_{j=1}^N G_j (\mathbf{v}_{jk} + (\mathbf{v}_{jk})_{\text{im}}) \right) \times \boldsymbol{\zeta}_k \right| G_k - C_{l_{\alpha k}} (\alpha_k) \quad (\text{A.5})$$

The Newton corrector equation iteratively reduces the residual:

$$[\mathbf{J}] \Delta \mathbf{G} = -\mathbf{R} \quad (\text{A.6})$$

In this equation $\Delta \mathbf{G}$ is the correction vector and $[\mathbf{J}]$ is the matrix of partial derivatives:

$$J_{kj} = \frac{\partial \mathcal{F}_k}{\partial G_j} = \begin{cases} \left[\frac{2\mathbf{w}_k \cdot (\mathbf{v}_{jk} \times \boldsymbol{\zeta}_k)}{|\mathbf{w}_k|} G_k - \frac{\partial C_{l_k}}{\partial \alpha_k} \frac{v_{a_k} (\mathbf{v}_{jk} \cdot \mathbf{u}_{n_k}) - v_{n_k} (\mathbf{v}_{jk} \cdot \mathbf{u}_{a_k})}{v_{a_k}^2 + v_{n_k}^2} \right], & j \neq k \\ \left[2|\mathbf{w}_k| + \frac{2\mathbf{w}_k \cdot (\mathbf{v}_{jk} \times \boldsymbol{\zeta}_k)}{|\mathbf{w}_k|} G_k - \frac{\partial C_{l_k}}{\partial \alpha_k} \frac{v_{a_k} (\mathbf{v}_{jk} \cdot \mathbf{u}_{n_k}) - v_{n_k} (\mathbf{v}_{jk} \cdot \mathbf{u}_{a_k})}{v_{a_k}^2 + v_{n_k}^2} \right], & j = k \end{cases} \quad (\text{A.7})$$

$$\mathbf{w}_k \equiv \left(\mathbf{v}_{\text{in}_k} + \sum_{j=1}^N G_j (v_{jk} + (v_{jk})_{\text{im}}) \right) \times \zeta_k \quad (\text{A.8})$$

$$v_{n_k} \equiv \left(\mathbf{v}_{\text{in}_k} + \sum_{j=1}^N G_j (v_{jk} + (v_{jk})_{\text{im}}) \right) \cdot \mathbf{u}_{n_k} \quad (\text{A.9})$$

$$v_{a_k} \equiv \left(\mathbf{v}_{\text{in}_k} + \sum_{j=1}^N G_j (v_{jk} + (v_{jk})_{\text{im}}) \right) \cdot \mathbf{u}_{a_k} \quad (\text{A.10})$$

After solving Equation (A.6) for the correction vector $\Delta \mathbf{G}$ a new estimate of the dimensionless vortex strength $\mathbf{G}$ can be found from:

$$\mathbf{G} = \mathbf{G} + \Omega \Delta \mathbf{G} \quad (\text{A.11})$$

where Ω is a relaxation factor. This method is repeated until a satisfactory convergence criterion on the magnitude of $\mathbf{R}$ is reached.

B

ADDITIONAL RESULTS OF PROPELLER-SRV-WING CFD MODEL

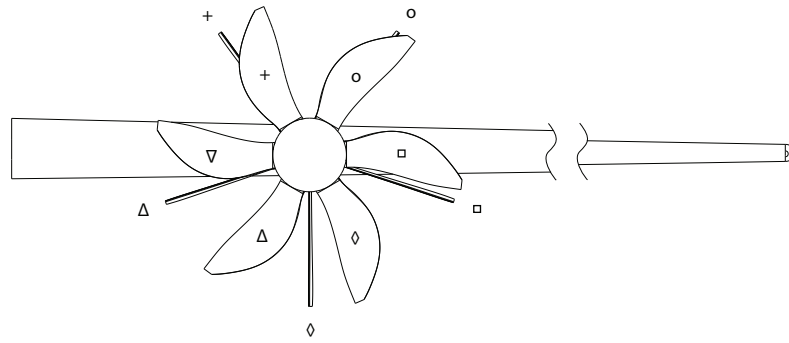


Figure B.1: Repetition of Figure 6.4: Definition of the propeller blades and swirl recovery vanes for $t/t_{bp} = 0$ in a front view of the propeller-SRV-wing configuration.

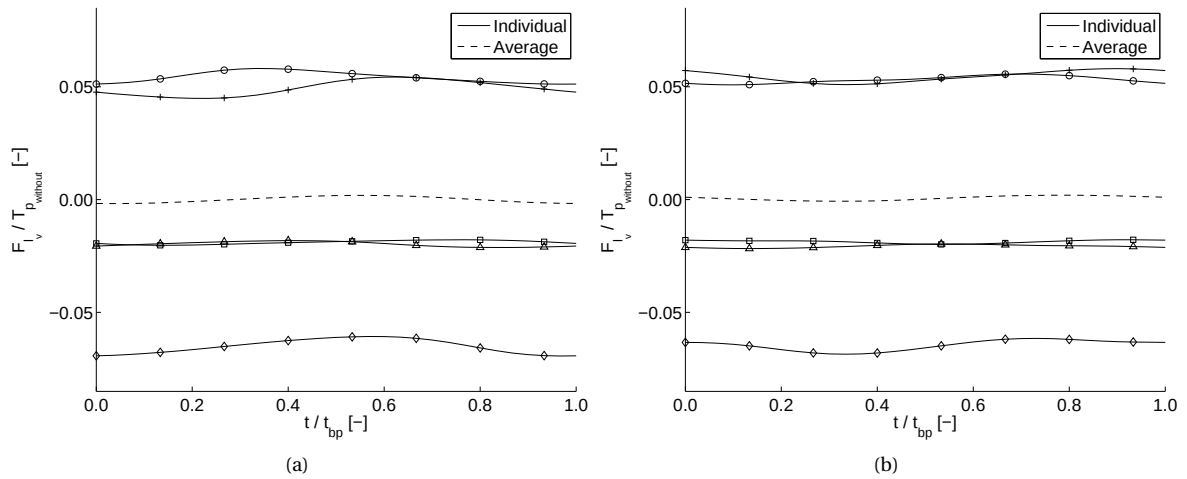


Figure B.2: SRV individual and average vane lateral force L_{l_v} (inboard positive) relative to the time-average propeller thrust without SRV $T_{p_{without}}$ for the duration of one blade passage period t_{bp} as defined in Figure B.1 at (a) $J = 1.60$ and (b) $J = 1.30$.

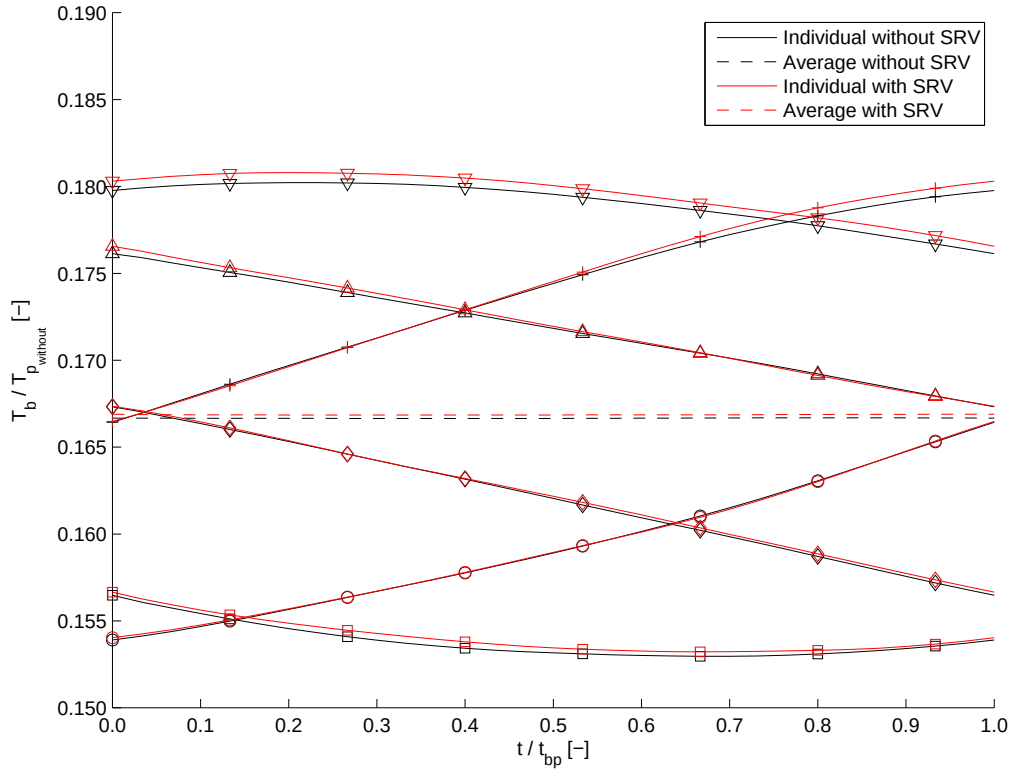


Figure B.3: Propeller individual and average blade thrust T_b relative to the time-average propeller thrust without SRV $T_{p_{without}}$ for the duration of one blade passage period t_{bp} as defined in Figure B.1 at $J = 1.30$.

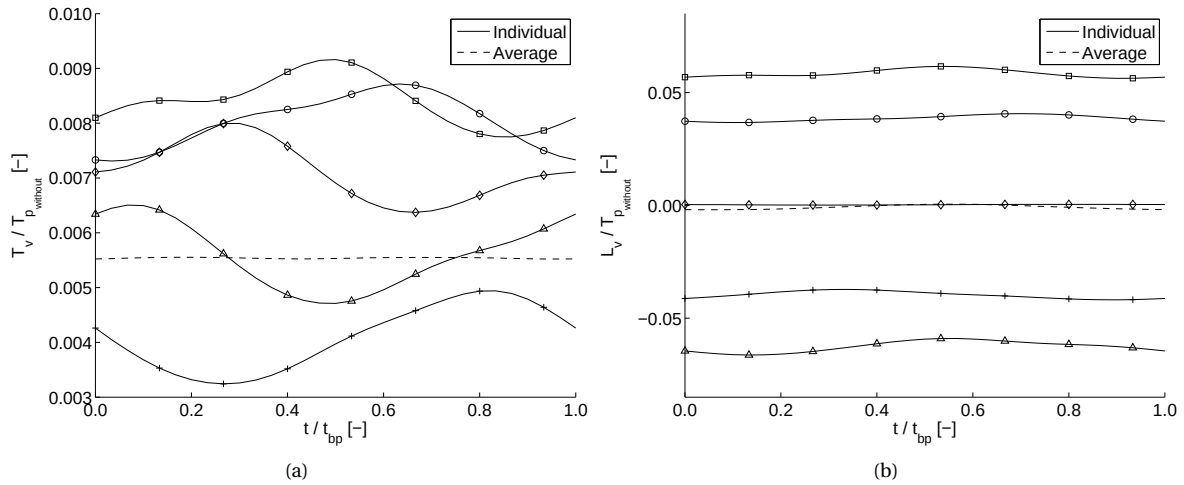


Figure B.4: SRV individual and average vane (a) thrust T_v and (b) lift L_v relative to the time-average propeller thrust without SRV $T_{p_{without}}$ for the duration of one blade passage period t_{bp} as defined in Figure B.1 at $J = 1.30$.

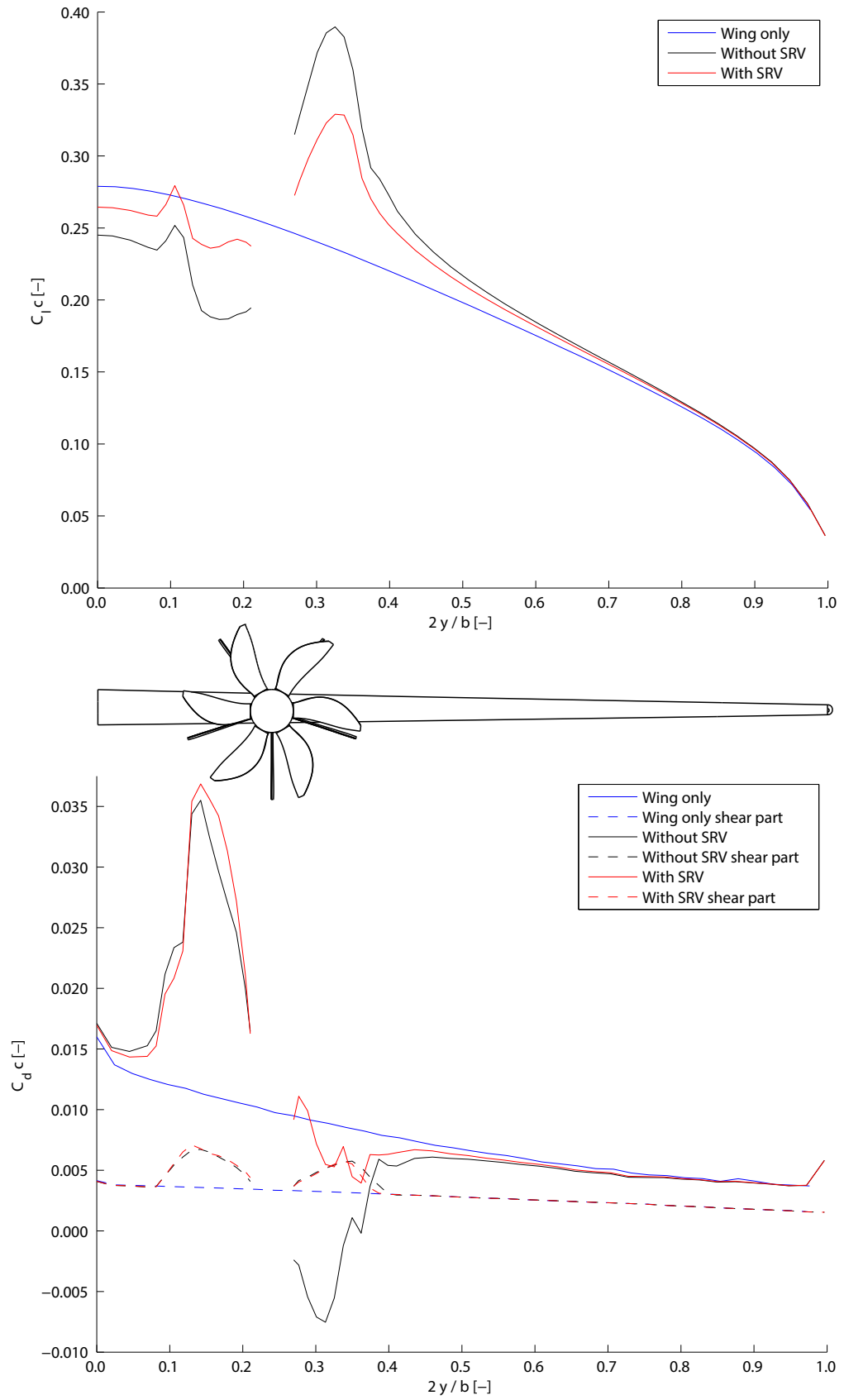


Figure B.5: Wing lift and drag distribution defined with undisturbed conditions for the wing only and with propeller and nacelle with and without SRV at $J = 1.30$.

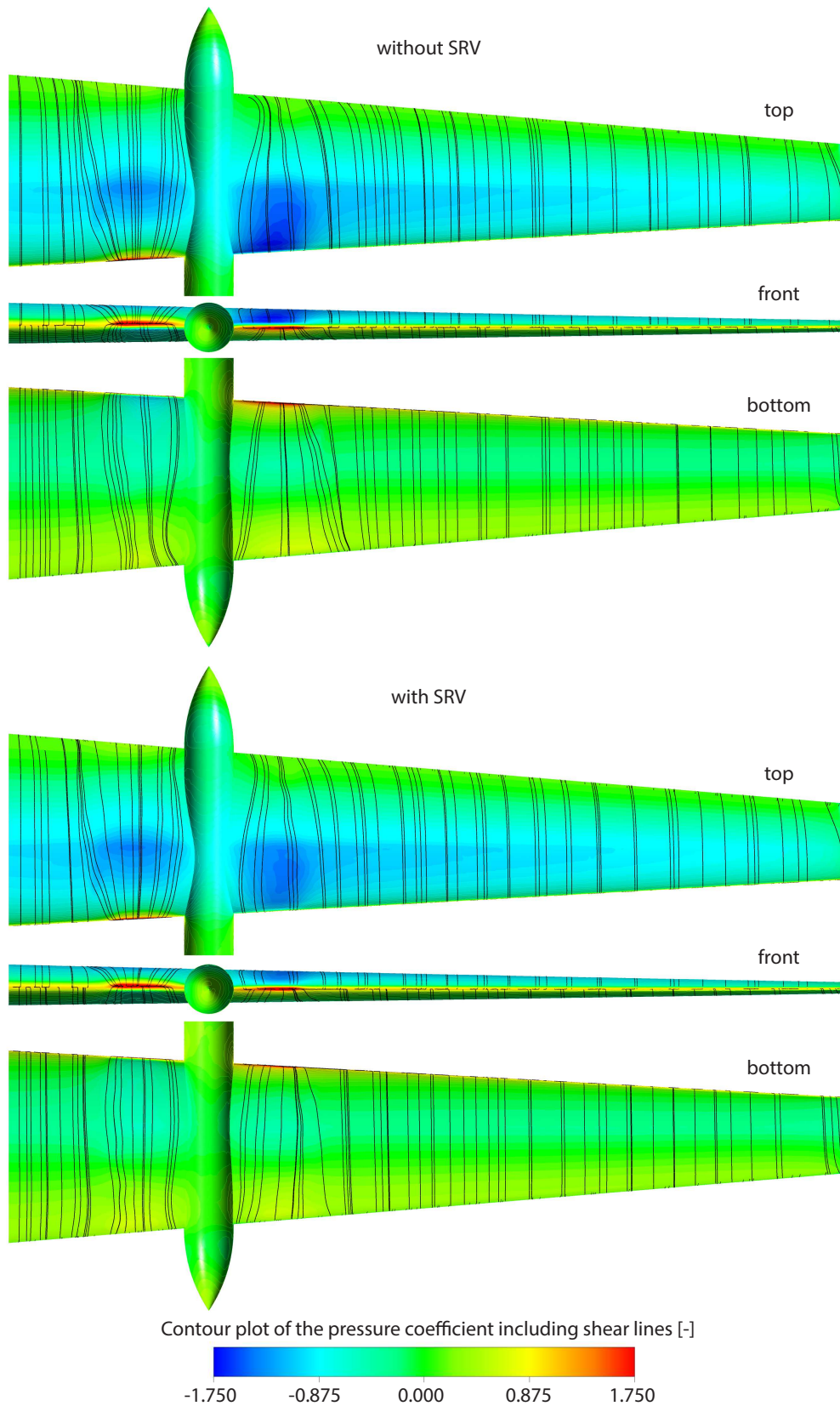


Figure B.6: Contour plot of the pressure coefficient defined with undisturbed conditions on the wing and aft part of the nacelle with and without SRV at $J = 1.30$.

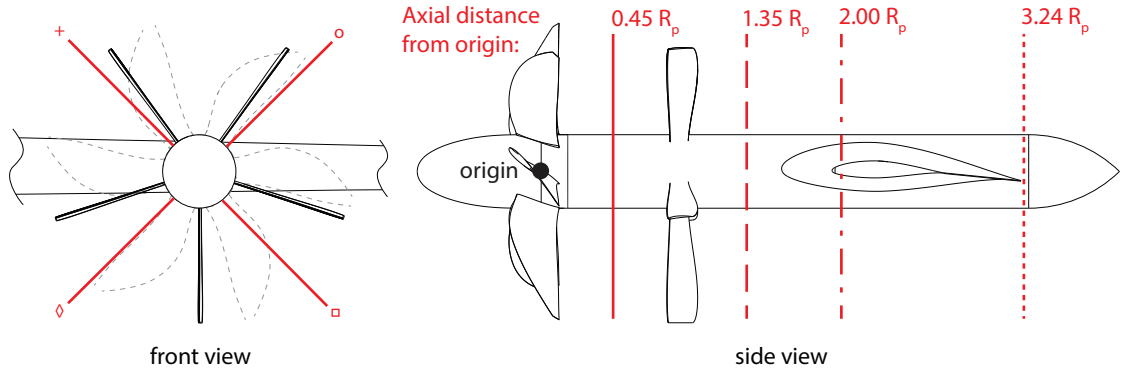


Figure B.7: Repetition of Figure 6.11: Definition of radial lines in four planes downstream of the propeller in a front and side view of the propeller-SRV-wing configuration.

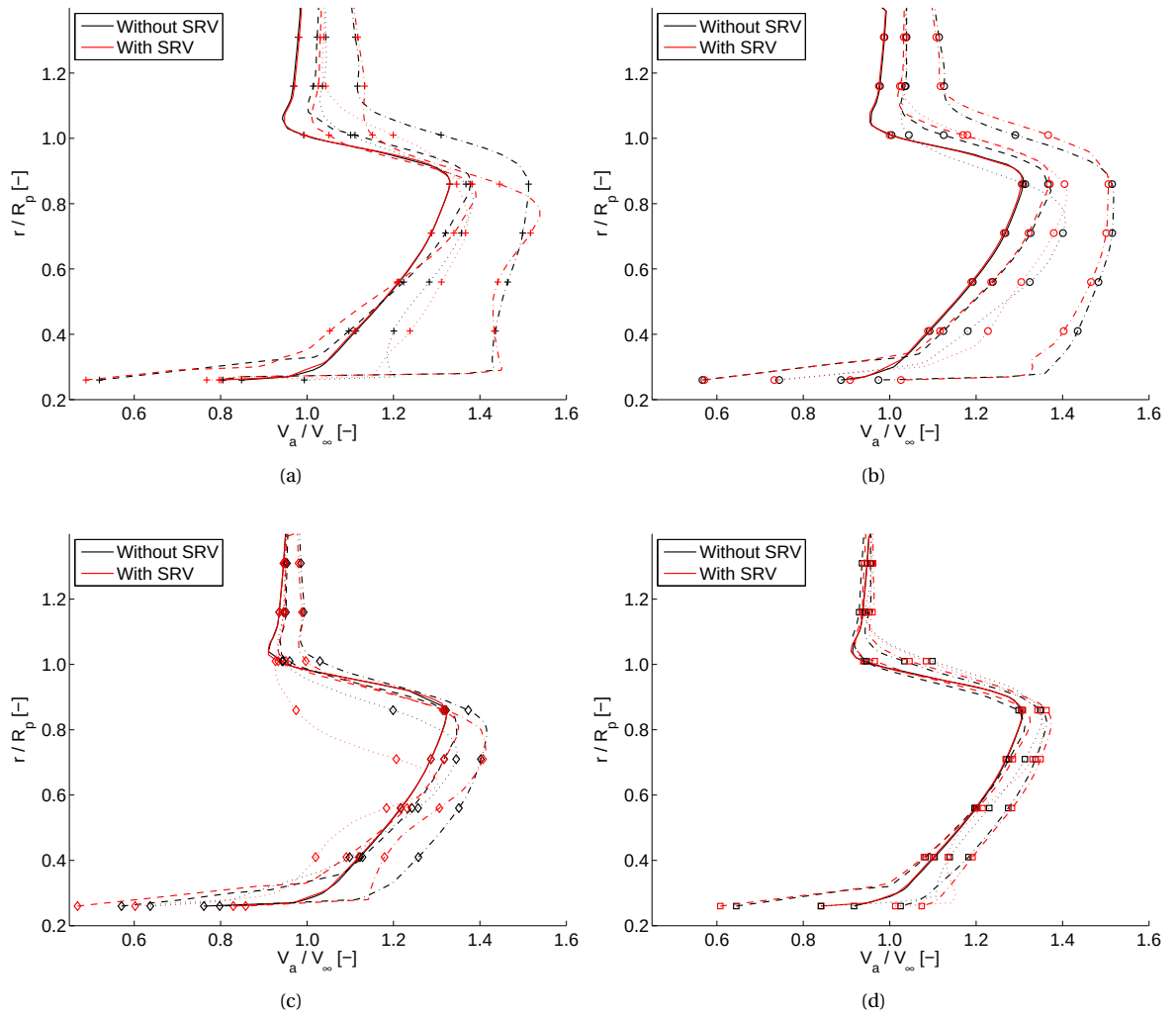


Figure B.8: Time-average axial velocity component without and with SRV on four radial lines in four planes downstream of the propeller as defined in Figure B.7 for the propeller-SRV-wing configuration.

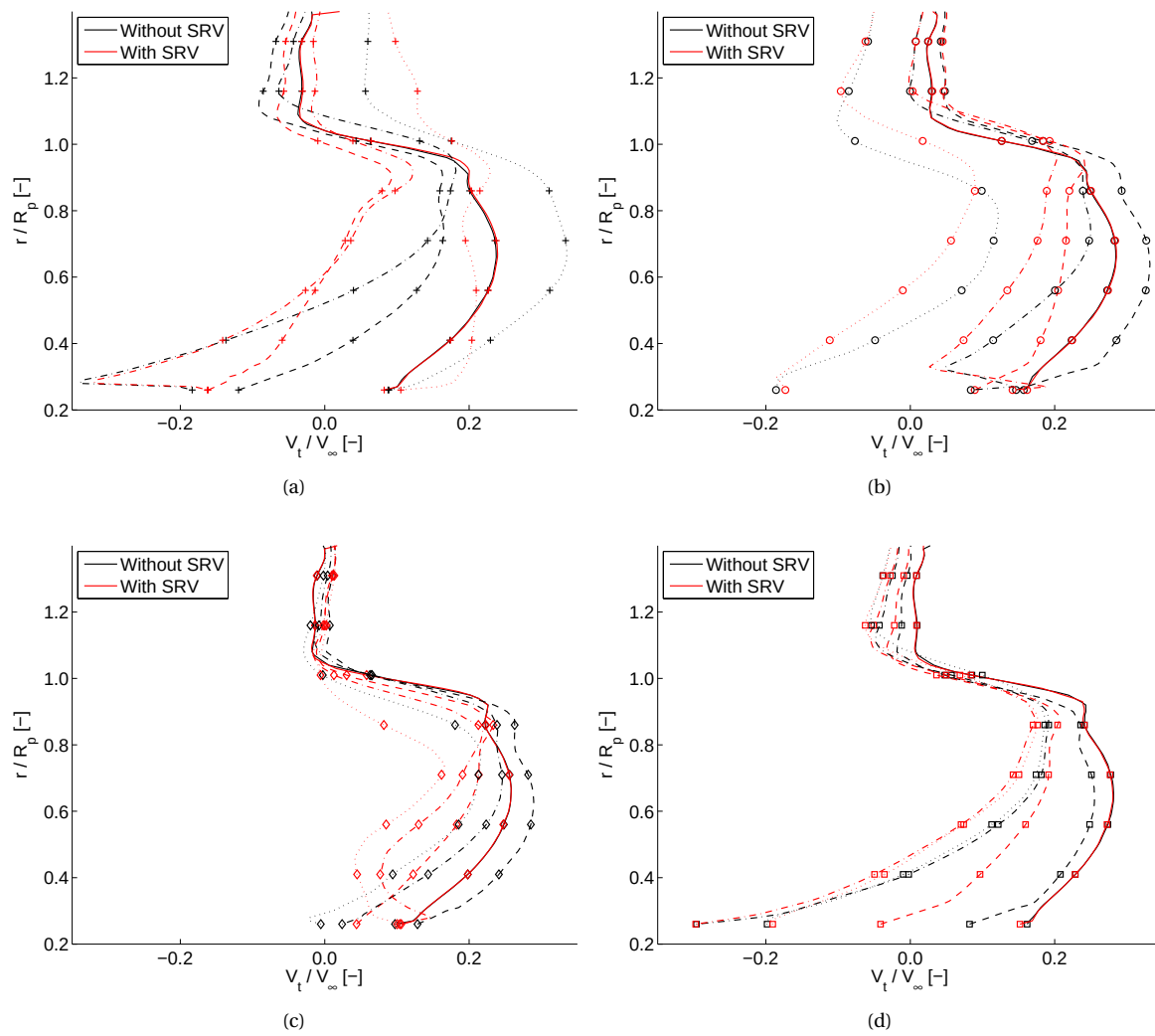


Figure B.9: Time-average tangential velocity component without and with SRV on four radial lines in four planes downstream of the propeller as defined in Figure B.7 for the propeller-SRV-wing configuration.

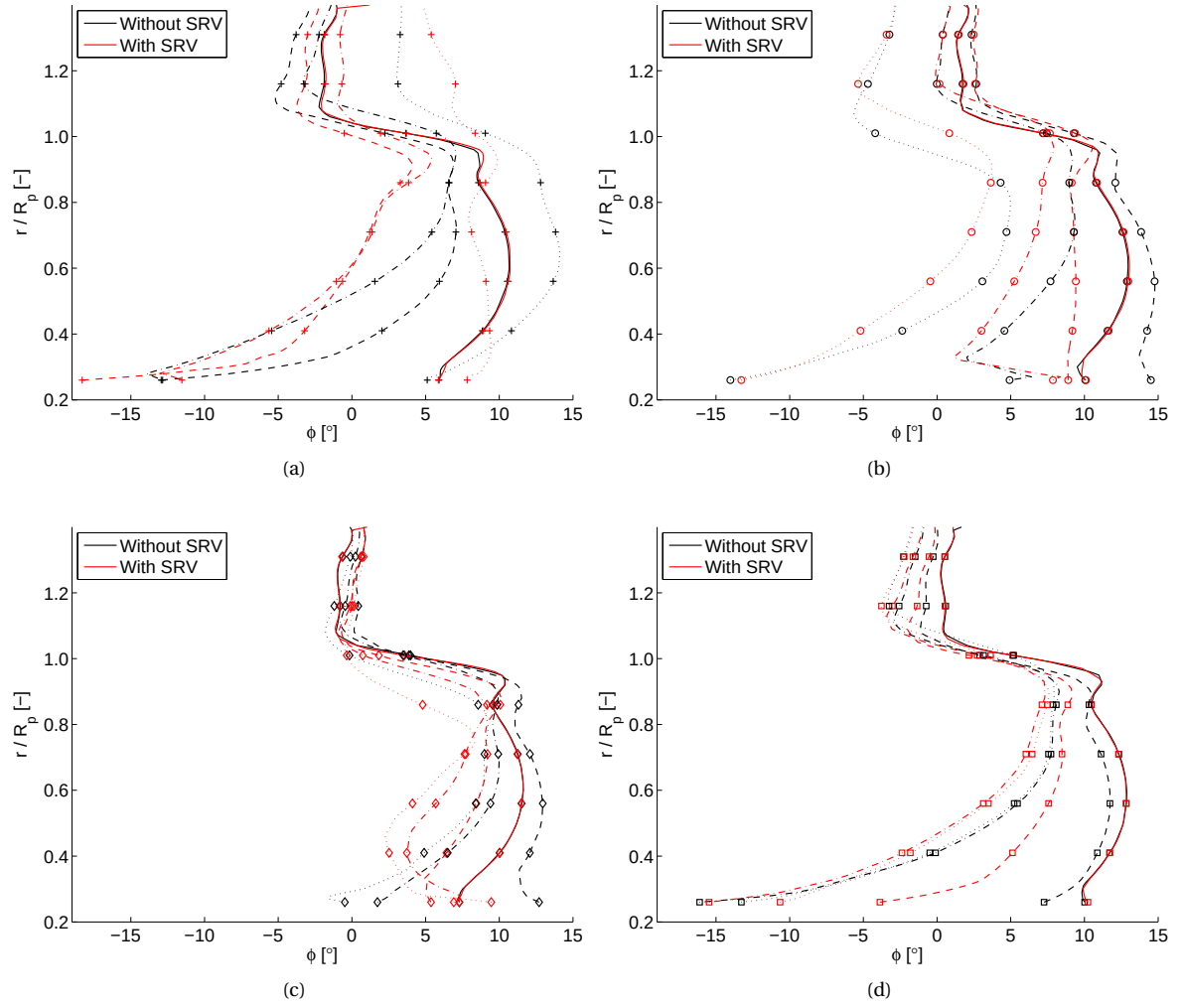


Figure B.10: Time-average swirl angle without and with SRV on four radial lines in four planes downstream of the propeller as defined in Figure B.7 for the propeller-SRV-wing configuration.

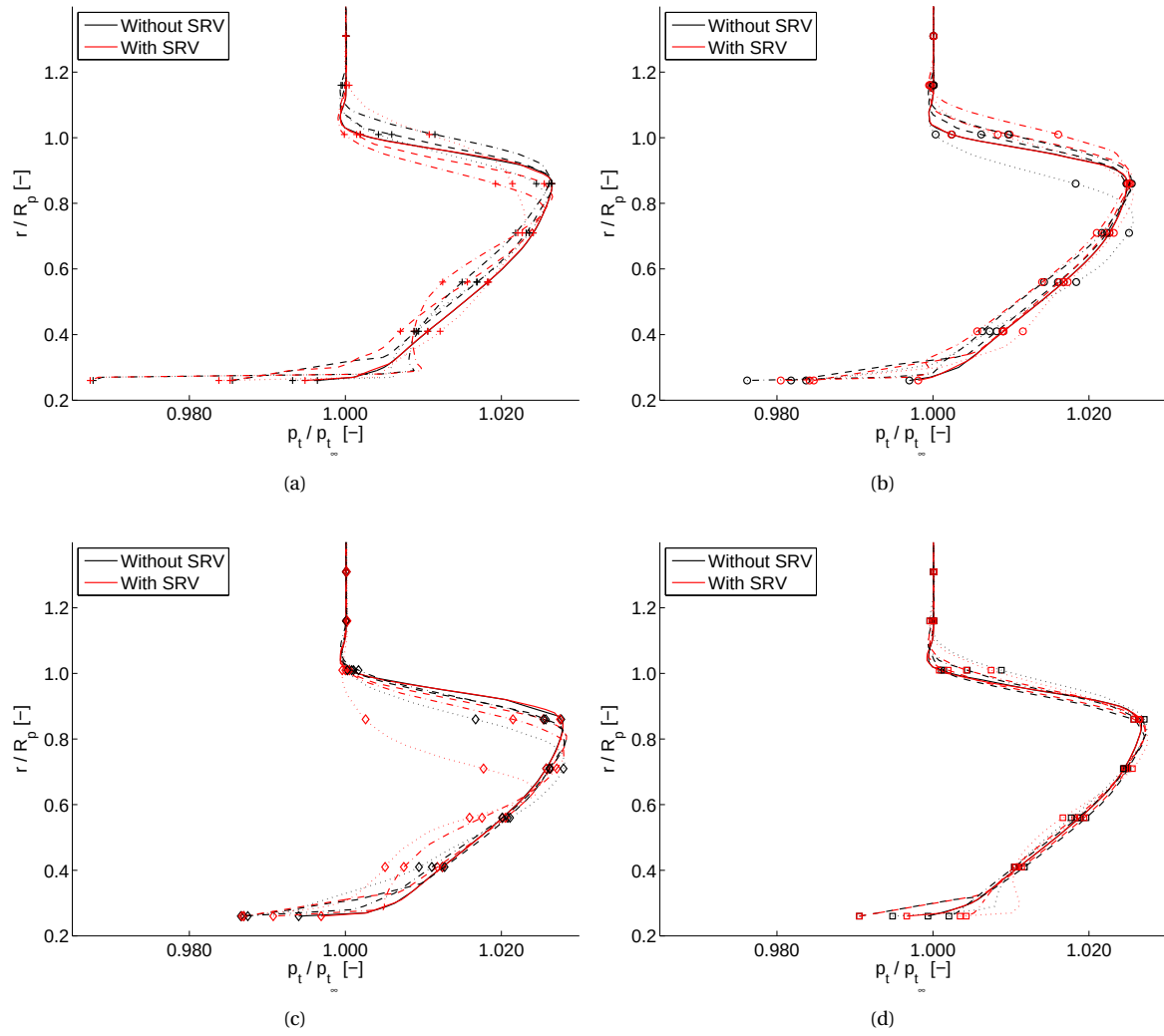


Figure B.11: Time-average total pressure without and with SRV on four radial lines in four planes downstream of the propeller as defined in Figure B.7 for the propeller-SRV-wing configuration.

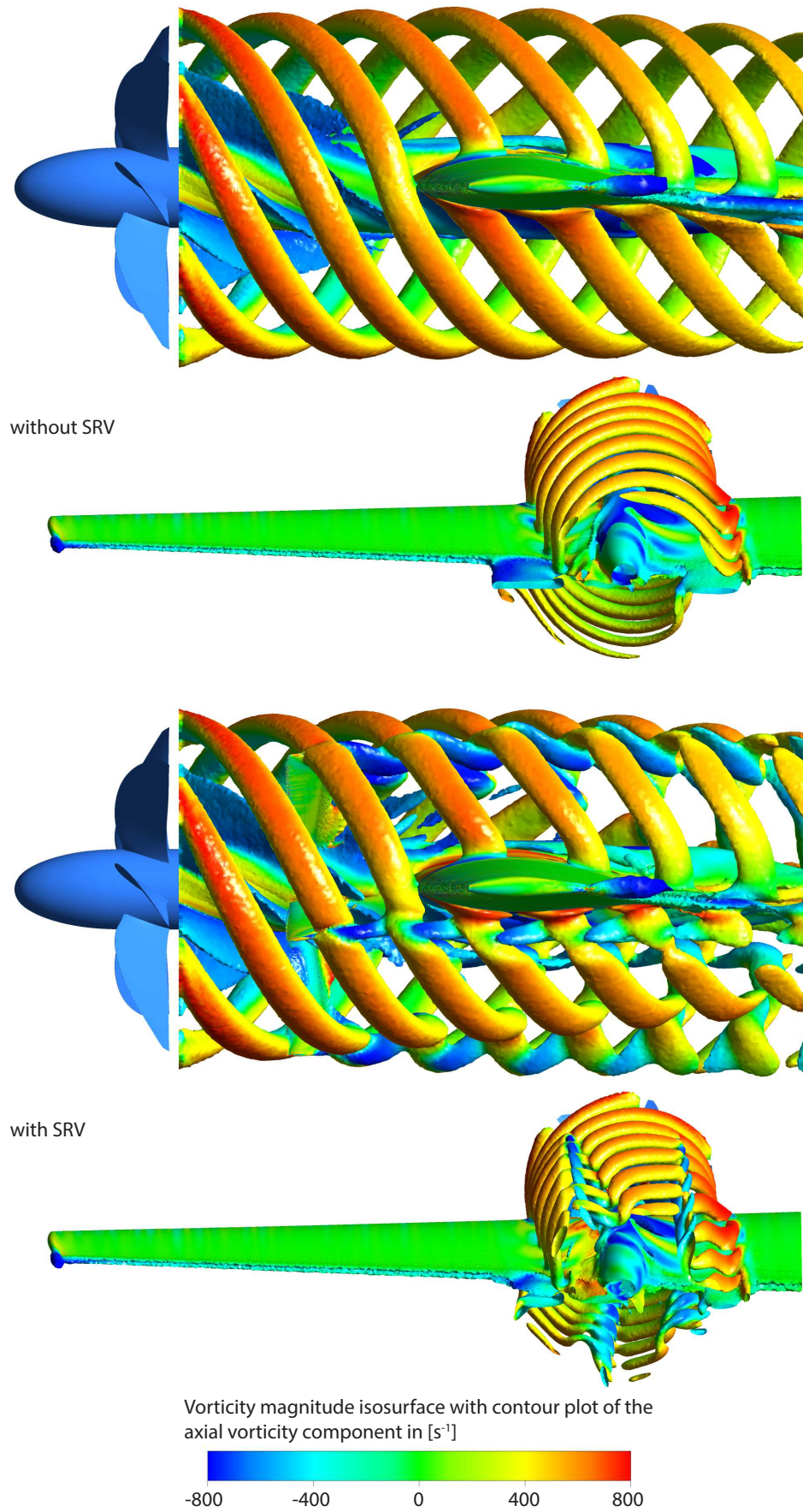


Figure B.12: Side and rear view isosurfaces of the vorticity magnitude behind the wing-mounted APIAN propeller without and with SRV designed for cruise including a contour plot of the axial vorticity component at an instance of the propeller blade passage at $J = 1.30$.

BIBLIOGRAPHY

- [1] A. Scott Berg. *Lindbergh*. Putnam Publishing Group, 1998.
- [2] C. G. Curtis. Marine propeller., January 1903. US Patent 803,671.
- [3] F. Weinig. *Aerodynamik der Luftschraube*. J. Springer, 1940.
- [4] M. D. Bowles and V. P. Dawson. The Advanced Turboprop Project: Radical Innovation in a Conservative Environment. <http://history.nasa.gov/SP-4219/Chapter14.html>, August 2014.
- [5] W. C. Strack, G. Knip, and A. Weisbrich. Technology and benefits of aircraft counter rotation propellers. Technical report, National Aeronautics and Space Administration, Cleveland, OH (USA). Lewis Research Center, 1981.
- [6] Y. Wang, Q. Li, G. Eitelberg, L. L. M. Veldhuis, and M. Kotsonis. Design and numerical investigation of swirl recovery vanes for the Fokker 29 propeller. *Chinese Journal of Aeronautics*, 27(5):1128–1136, 2014.
- [7] T. Sinnige, J. J. A. van Kuijk, K. P. Lynch, D. Ragni, G. Eitelberg, and L. L. M. Veldhuis. The Effects of Swirl Recovery Vanes on Single-Rotation Propeller Aerodynamics and Aeroacoustics. 21st AIAA/CEAS Aeroacoustics Conference, Dallas, Texas, 22 – 26 June 2015, American Institute of Aeronautics and Astronautics, 2015.
- [8] W. F. Phillips. *Mechanics of Flight*. Wiley, 2010.
- [9] J. A. Gazzaniga and G. E. Rose. *Wind tunnel performance results of swirl recovery vanes as tested with an advanced high speed propeller*. Joint Propulsion Conferences. American Institute of Aeronautics and Astronautics, 1992. doi:10.2514/6.1992-3770.
- [10] J. H. Dittmar and D. G. Hall. Cruise Noise of an Advanced Propeller with Swirl Recovery Vanes. *Journal of Aircraft*, 30(2):221–226, 1993.
- [11] O. Yamamoto. *Numerical calculations of propfan/swirl recovery vane flow field*. Joint Propulsion Conferences. American Institute of Aeronautics and Astronautics, 1992. doi:10.2514/6.1992-3771.
- [12] C. J. Miller. *Euler analysis of a swirl recovery vane design for use with an advanced single-rotation propfan*. Joint Propulsion Conferences. American Institute of Aeronautics and Astronautics, 1988. doi:10.2514/6.1988-3152.
- [13] F. Çelik and M. Güner. Energy saving device of stator for marine propellers. *Ocean Engineering*, 34: 850–855, 2007.
- [14] J. Farnsworth, M. Amitay, D. Beal, and S. A. Huyer. Interactions of a propeller with a stator-induced circumferentially varying flow. *Experiments in fluids*, 52(2):495–510, 2012.
- [15] G. J. J. Ruijgrok. *Elements of Airplane Performance*. Delft University Press, 1990. ISBN 9789062756087.
- [16] L. L. M. Veldhuis. *Propeller wing aerodynamic interference*. Delft University of Technology, 2005.
- [17] A. W. Stuermer. Unsteady CFD Simulations of Propeller Installation Effects. 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, Sacramento, California, 9–12 July, American Institute of Aeronautics and Astronautics, 2006. doi:10.2514/6.2006-4969.
- [18] I. Samuelsson. Experimental investigation of low speed model propeller slipstream aerodynamic characteristics including flow field surveys and nacelle/wing static pressure measurements. In *ICAS, Congress, 17th, Stockholm, Sweden*, pages 71–84, 1990.

- [19] P. Beaumier. Numerical Tools Developed at ONERA for the Aerodynamic Assessment of Propellers and Counter-Rotating Open Rotors. In *28th Congress of the International Council of the Aeronautical Sciences, Brisbane*, 2012.
- [20] I. Kroo. Propeller-wing integration for minimum induced loss. *Journal of Aircraft*, 23(7):561–565, 1986.
- [21] J. J. A. van Kuijk. Analysis of Swirl Recovery Vanes: Propulsion system performance and slipstream-wing interaction. Master's thesis, Aerospace Engineering, Delft University of Technology, 2015.
- [22] L. G. M. Custers and A. Elsenaar. Test report of the APIAN wind tunnel test in the DNW-HST. Technical Report NLR-CR-98571, Nationaal Lucht- en Ruimtevaartlaboratorium, 1999.
- [23] B. Ortun, R. Boisard, and I. Gonzalez-Martino. In-plane airloads of a propeller with inflow angle: prediction vs. experiment. 30th AIAA Applied Aerodynamics Conference, New Orleans, Louisiana, 25–28 June, American Institute of Aeronautics and Astronautics, 2012.
- [24] ANSYS® Academic Research Release 15.0. help system, Meshing User's Guide, ANSYS, Inc., 2013.
- [25] E. W. M. Roosenboom, A. Stürmer, and A. Schröder. Advanced Experimental and Numerical Validation and Analysis of Propeller Slipstream Flows. *Journal of Aircraft*, 47(1):284–291, 2010.
- [26] B. Francois, M. Costes, and G. Dufour. Comparison of chimera and sliding mesh techniques for unsteady simulations of counter rotating open-rotors. In *20th ISABE Conference*, 2011.
- [27] ANSYS® Academic Research Release 15.0. help system, CFX, ANSYS, Inc., 2013.
- [28] L. Davidson. Fluid mechanics, turbulent flow and turbulence modeling. Lecture notes, Chalmers University of Technology, 2014.
- [29] K. Fujii. Progress and future prospects of CFD in aerospace-Wind tunnel and beyond. *Progress in Aerospace Sciences*, 41(6):455–470, 2005.
- [30] H. I. Andersson and M. Lygren. LES of open rotor-stator flow. *International Journal of Heat and Fluid Flow*, 27(4):551–557, 2006.
- [31] P. W. Bearman and T. Morel. Effect of free stream turbulence on the flow around bluff bodies. *Progress in aerospace sciences*, 20(2):97–123, 1983.
- [32] A. Saxena. Guidelines for the Specification of Turbulence at Inflow Boundaries. http://www.esi-cfd.com/esi-users/turb_parameters/ Accessed June, 2015.
- [33] L. L. M. Veldhuis and G. W. Luursemma. *Comparison of an actuator disk and a blade modeling approach in Navier-Stokes calculations on the SR-3 propfan*. Fluid Dynamics and Co-located Conferences. American Institute of Aeronautics and Astronautics, 2000. doi:10.2514/6.2000-4528.
- [34] E. Schüle, H. Rosemann, and S. Schaber. Transition Detection and Skin Friction Measurements on Rotating Propeller Blades. 28th AIAA aerodynamic measurement technology, ground testing and flight testing conference, New Orleans, Louisiana, 25–28 June, 2012.
- [35] W. S. Westmoreland, R. W. Tramel, and J. Barber. *Modeling Propeller Flow-Fields Using CFD*. Aerospace Sciences Meetings. American Institute of Aeronautics and Astronautics, 2008. doi:10.2514/6.2008-402.
- [36] M. Drela and H. Youngren. XROTOR: an interactive program for the design and analysis of ducted and free-tip propellers and windmills. <http://web.mit.edu/drela/Public/web/xrotor/> Accessed April, 2015.
- [37] R. Blumenthal, B. Hutchinson, and L. Zori. Investigation of transient CFD methods applied to a transonic compressor stage. In *ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition*, pages 1423–1430. American Society of Mechanical Engineers, 2011.
- [38] F. Montomoli, H. P. Hodson, and L. Lapworth. RANS-URANS in axial compressor, a design methodology. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 225(3):363–374, 2011. doi: 10.1177/2041296710394267.

- [39] I. H. Abbott and A. E. Von Doenhoff. *Theory of wing sections, including a summary of airfoil data*. Courier Corporation, 1959.
- [40] L. Prandtl. Applications of modern hydrodynamics to aeronautics. *NACA 116*, 1921.
- [41] M. Drela and H. Youngren. XFOIL: an interactive program for the design and analysis of subsonic isolated airfoils. <http://web.mit.edu/drela/Public/web/xfoil/> Accessed April, 2015.
- [42] M. Drela. XFOIL: An analysis and design system for low Reynolds number airfoils. In *Low Reynolds number aerodynamics*, pages 1–12. Springer Berlin Heidelberg, 1989.
- [43] J. Van Ingen. The eN method for transition prediction. Historical review of work at TU Delft. In *38th Fluid Dynamics Conference and Exhibit*, pages 23–26, 2008.
- [44] B. M. Kulfan. Universal Parametric Geometry Representation Method. *Journal of Aircraft*, 45(1):142–158, 2008.
- [45] W. F. Phillips and D. O. Snyder. Modern Adaptation of Prandtl’s Classic Lifting-Line Theory. *Journal of Aircraft*, 37(4):662–670, 2000.
- [46] W. F. Durand. *Aerodynamic Theory Vol. IV*. Julius Springer, 1935.
- [47] Mathworks. R2015a documentation: fmincon. <http://nl.mathworks.com/help/optim/ug/fmincon.html>, 2015.
- [48] S. A. Ning and I. Kroo. Tip extensions, winglets, and c-wings: conceptual design and optimization. 26th AIAA Applied Aerodynamics Conference, Honolulu, Hawaii, 18–21 August 2008, American Institute of Aeronautics and Astronautics, 2008.
- [49] C. L. Rumsey and S. X. Ying. Prediction of high lift: review of present cfd capability. *Progress in Aerospace Sciences*, 38(2):145–180, 2002.
- [50] L. Saxena, A. Fejer, and M. Morkovin. Features of unsteady flows over airfoils. In *AGARD-CP 227, Proceedings of AGARD Conference on Unsteady Aerodynamics, pp 22.1 - 22.11*, 1977.
- [51] R. T. Jones. Correction of the lifting line theory for the effect of the chord. Technical report, DTIC Document, 1941.
- [52] W. J. McCroskey. Unsteady airfoils. *Annual review of fluid mechanics*, 14(1):285–311, 1982.
- [53] K. Gharali and D. A. Johnson. Dynamic stall simulation of a pitching airfoil under unsteady freestream velocity. *Journal of Fluids and Structures*, 42:228–244, 2013.
- [54] H. Snel, R. Houwink, J. Bosschers, et al. *Sectional prediction of lift coefficients on rotating wind turbine blades in stall*. Netherlands Energy Research Foundation Petten, Netherlands, 1994.
- [55] M. Drela and H. Youngren. AVL: a program for the aerodynamic and flight-dynamic analysis of rigid aircraft of arbitrary configuration. <http://web.mit.edu/drela/Public/web/avl/> Accessed July, 2015.
- [56] L. L. M. Veldhuis, M. Kotsonis, and E. van Berkel. Non-Uniform Inflow Effects on Propeller Performance. 31st AIAA Applied Aerodynamics Conference, San Diego, California, 24 – 27 June 2013, American Institute of Aeronautics and Astronautics, 2013. doi:10.2514/6.2013-2801.
- [57] A. Stuermer and M. Rakowitz. Unsteady Simulation of a Transport Aircraft Propeller Using MEGAFLOW. Technical report, DTIC Document, 2005.
- [58] E. Obert. *Aerodynamic design of transport aircraft*. IOS Press, 2009.