Determination of the Maximum Aerodynamic Efficiency of Wind Turbine Rotors with Winglets

Mac Gaunaa & Jeppe Johansen
Senior Scientists, Risø National Laboratory, Roskilde, DK-4000, Denmark

Abstract. The present work contains theoretical considerations and computational results on the nature of using winglets on wind turbines. The theoretical results presented show that the power augmentation obtainable with winglets is due to a reduction of tip-effects, and is not, as believed up to now, caused by the downwind vorticity shift due to downwind winglets. The numerical work includes optimization of the power coefficient for a given tip speed ratio and geometry of the span using a newly developed free wake lifting line code, which takes into account also viscous effects and self induced forces. Validation of the new code with CFD results for a rotor without winglets showed very good agreement. Results from the new code with winglets indicate that downwind winglets are superior to upwind ones with respect to optimization of Cp, and that the increase in power production is less than what may be obtained by a simple extension of the wing in the radial direction. The computations also show that shorter downwind winglets (>2%) come close to the increase in Cp obtained by a radial extension of the wing. Lastly, the results from the code are used to design a rotor with a 2% downwind winglet, which is computed using the Navier-Stokes solver EllipSys3D. These computations show that further work is needed to validate the FWLL code for cases where the rotor is equipped with winglets.

Keywords: Winglet, Maximum Aerodynamic Efficiency

1. Introduction

Many wind turbine sites have restrictions on rotor diameter in one form or another. In those cases, the only way the power production can be optimized at any specific wind velocity is through maximizing the power coefficient, Cp, of the wind turbine. To that end, winglets can be used, since these can improve the power production without increasing the projected rotor area.

Winglets have been studied extensively for non-rotating applications such as airplanes and sailplanes. The total drag on a three-dimensional wing basically consists of two parts: a viscous (2D) part, and an induced (3D) part. The induced drag stems from the lift because the trailing vortex system behind the wing induces velocities perpendicular to the free stream direction. The local inviscid lift forces act perpendicular to the relative velocity, so the induced velocities results in forces in the direction of the
free stream. This is what is called induced drag. Since the downwash is proportional to the lift, the induced drag scales with the square of the lift (see for instance the excellent review paper by Kroo[1]). Essential works within the field of winglets on non-rotating wings include Munk’s [2] pioneering lifting line work, which showed that the optimal normal-wash distribution for a non-planar wing is proportional to cosine of the dihedral angle. The same work showed that the condition for a minimum drag does not depend on the longitudinal coordinates, which includes the effects of sweep. It can be shown that a vertical surface (with no viscous drag) located at the tip of a wing is worth approximately 45% of its height as additional span[1]. Other works within non-rotating winglet literature include references by Witcomb[3] Jones[4], Eppler[5] and Maughmer[6]. These works all include considerations on viscous effects. Whitcomb[3] outlined a design approach for the design of winglets on airplanes. Jones[4] used the variational approach to find the optimum load distribution for wing-winglet combinations with restrictions on bending moments, which is related to a considerable portion of the weight of the wing. Eppler’s work[5] uses an ingenious method to investigate the effect of the assumption of a rigid wake, as it was in the early works dealing with winglets. This investigation results in a determination of what Eppler dubs induced lift forces. Maughmer’s work[6] deal with how to use winglets to improve the performance of sail planes, which work under a wide range of operating conditions and restrictions (multi point optimization). As explained very nicely in Maughmer’s works, the main purpose of adding a winglet to an airplane wing is to decrease the total drag for a given lift. Reduction of total drag is obtained if the additional drag from the winglet is less than the reduction in induced drag on the remaining blade. The art is then to design a winglet which optimizes drag reduction for a given lift. The working principle behind the winglet in the non-rotating case is that the winglet diffuses and moves the tip vortex away from the main wing/wake plane, which reduces the downwash and thereby the induced drag on the wing.

Compared to the number of references on winglets on non rotating wings, the work done so far for application of winglets on rotors is fairly limited.

A tip attachment that has been investigated by Delft university (Van Holten[7]) and Mie university (Shimizu et al.[8]), is the Tip-vane, or Mie-vane, which is basically a t-wing at the tip of the main wing. At first sight, this device looks somewhat like a winglet for wind turbines. The main role of this device, however, is to act as a concentrator system, increasing mass flow through the main rotor disc due to the formation of a ring-vortex-like vortical structure emanating from the upwind tips of the Mie-vanes. Since the topic to be investigated in the present paper is the use of standard one-sided winglets, which does not act as a concentrator system, the similarity between Mie vanes and winglets for wind turbines is not very great. According to the wind tunnel measurements [8], the maximum power coefficient, Cp, obtained with a Mie-vane was Cp=0.45 at a tip speed ratio of 4. This corresponded to a 27% increase compared to the case without tip attachments. Furthermore, according to the measurements, the Mie-vane has not proven to increase Cp for a turbine for tip speed ratios above 8.

For the application of winglets on horizontal-axis wind turbines, previous works has been done by Van Bussel [9], Imamura et al.[10] and Johansen[11-12]. Van Bussel[9] developed a momentum theory for winglets on HAWTs, explaining the positive effect of a winglet on a rotor by the downwind shift of the wake vorticity, which the authors of the present work believe is erroneous, as shall be shown in the section on simple modeling later on. The works by Imamura et al. and Johansen are numerical in nature, as the former work is based on a free-wake vortex-lattice method. Johansen’s work is based on Navier-Stokes simulations with the CFD code EllipSys3D, and predicts an increase in Cp of 1.74% on a modern MW sized turbine.

Along the same lines as for the non-rotating case, the main purpose of adding a winglet to a wind turbine rotor is to decrease the total drag from the blade and thereby increase the aerodynamic
efficiency of the turbine. Reduction of total drag is obtained if the additional drag from the winglet is less than the reduction in induced drag on the remaining blade. The art is then to design a winglet which maximizes power production for the whole rotor.

The use of winglets on rotors other than wind energy applications was performed by Müller & Staufenbiel[13] and Müller[14], and deals with using winglets on helicopters.

The current work aims at describing the mechanisms of winglets on turbine rotors, and furthermore to determine the maximum achievable aerodynamic efficiency, defined as the mechanical power coefficient, Cp, for wind turbine rotors with winglets, using both a newly developed Free Wake Lifting Line (FWLL) method and the CFD code EllipSys3D. This is of interest to both the wind turbine industry as well as the research community.

2. The Problem in a Nut-Shell
By considering a lifting-line type model for a wind turbine rotor with winglets, an interesting insight into what winglet do may be obtained. Consider the model outlined in Figure 1.

![Figure 1: Coordinate systems used in lifting line analysis.](image)

The force per unit length vortex element was given by Joukowski as $\vec{F} = \rho \vec{V}_{rel} \times \vec{\Gamma}$, where $\vec{\Gamma}$ is the circulation vector, $\vec{V}_{rel}$ is the velocity vector relative to the vortex element, and $\rho$ is the density of the fluid. Using the notation from Figure 1, we get for the forces on the main wing that lies on the X-axis:

$$\vec{F}_{main} = \rho \vec{V}_{rel} \times \vec{\Gamma} = \rho \begin{bmatrix} 0 \\ V + V_{iZ} \\ \Omega X - V_{iy} \end{bmatrix} \leftarrow \text{Radial forces}$$

$$\vec{F}_{main} = \rho \vec{V}_{rel} \times \vec{\Gamma} = \rho \begin{bmatrix} -\Omega X + V_{iy} \\ -V_{ix} \\ 0 \end{bmatrix} \leftarrow \text{Tangential forces ("Power"), Axial forces ("Thrust")}

And for the forces on the winglet:

$$\vec{F}_{winglet} = \rho \vec{V}_{rel} \times \vec{\Gamma} = \rho \begin{bmatrix} -\Omega X + V_{iy} \\ -V_{ix} \\ 0 \end{bmatrix} \leftarrow \text{Radial forces, "Power" contribution from winglet}

\text{No thrust generation!}

Note, that subscript $i$ refer to all induced velocities on a specific point on the rotor, which includes both the classical induced velocities, as known from blade element theory, plus tip effects and self-
induced velocities. The expression for the force on the winglet is derived for a downwind winglet, corresponding to what is shown in Figure 1. If an upwind winglet is used, the forces change sign. As seen from the result, the component of the forces that contribute to the power production from the main wing is a product of circulation and the effective axial velocity. Due to the “vortex laws” any change in circulation along the wing is trailed into the wake, a higher circulation generally results in lower effective axial velocities. The problem of finding the optimal circulation distribution for a non-wingletted rotor was treated by Goldstein[15]. However, when a winglet is present, the loading can be extended onto this, which makes possible for the outer radial positions to run with higher bound circulation, and thus, increased production. Furthermore, the trailed vorticity emanating from the winglet is brought downstream, and out of the main rotor-plane. Both these has a positive effect on power production on the main wing; the latter being analog to the benefit from winglets in the non-rotating case. The power contributing forces from the winglet itself includes a product between bound circulation and negative axial velocity. Due to the decrease of the axial velocity in the axial direction, it is evident from mass conservation that the radial flow is positive, and therefore, that, for the downwind winglet, the power production is negative. The benefits that the main part of the wing enjoys come at this cost. We will go into this in more detail later. Considering this argument, an upwind winglet might seem like a better choice, since in this case the power production on the winglet itself would be positive. In the upwind case, however, the vorticity from the winglets are trailed at an upstream position (axial direction), which reduces the axial velocities, and thereby, the power production, on the main rotor. As seen from the equations, the thrust is mainly affected due to the increased loading near the tip of the main rotor; the winglet itself produces no thrust.

3. Simple Modelling: Actuator Cap/Vortex Tube Theory

In order to shed some light on the physics behind using winglets on wind turbine blades, we consider a simple model analogous to a non-wingletted model by Øye[16]. In this model we consider a wind turbine with winglets, as shown in Figure 2, with infinitely many uniformly loaded blades.

Figure 2: Geometry used in the actuator cap/vortex tube model. Black: actuator cap. Blue: distinct root vortex and tip vortex tube.

Due to the uniformly loaded blades, vorticity is shed at the root and tip only. The root vortices combine to one single vortex along the rotor axis, and the vorticity shed from the tips form a vortex tube, as shown in Figure 2. The results from this model do not include tip effects, since the number of blades is infinite, so the results can be compared to basic actuator disc computations. A more fitting term for the present geometry is actuator cap, which is adopted in the following.

For a model such like the present, it can be shown [16] that all axial and radial induction in the actuator cap case comes from the outer vortex wake tube only. This means that the axial component of the flow passing through the plane through the winglet tips, where the tip vorticity is shed, is independent of the axial position of the main rotor. From application of mass conservation through “the cap”, it follows that

\[
\int_{0}^{R} (V + V_{az}(x,0)) 2\pi x \, dx = \int_{0}^{L_{wl}} 2\pi R V_{ax}(R, z) \, dz + \int_{0}^{R} (V + V_{az}(x, L_{wl})) 2\pi x \, dx.
\]
Note that $L_{wl}$ is the length of the winglet. From Equations (1) and (2) we can integrate the power production for the actuator cap

$$P_{AC} = \rho \Gamma \Omega \left( \int_{0}^{R} (V + V_{dz}(x,0))dx - \int_{0}^{L_{wl}} V_{dz}(R,z)Rdz \right),$$

which after insertion of the result from Equation (3) yields

$$P_{AC} = \rho \Gamma \Omega \left( \int_{0}^{R} (V + V_{dz}(x, L_{wl}))dx \right).$$

However, since axial and radial induction stems from the outer wake tube only, this is seen to be identical to the power production for the ordinary actuator disc without winglets

$$P_{AC} = P_{AD}. \quad (6)$$

The arguments leading to the above result is equally valid for downstream as well as upstream winglets. The increase or decrease of power production on the main wing is exactly balanced out by the power consumption or production on the winglet. This means, that in the absence of tip effects, adding winglets does in fact not increase power production. This tells us that what winglets can help us do on wind turbine wings is to reduce tip effects, known in the Blade Element Momentum frame of reference as tip loss. So in real life, the positive effect of a winglet has to come from a higher reduction of tip-effects than the increase in viscous drag from the winglet. So, according to the present work, the (global) Betz limit should still be the absolute upper limit for $C_p$.

In fact, taking a Trefftz-plane approach with the mass conservation and energy equations, the result is that the power production of anything inside the cylindrical control volume extending from far upstream to far downstream is given by the velocity profile of what flows out of the control volume. Since the only axial and radial induction comes from the wake itself, any uniformly loaded actuator-cap shape ending at the same radial position will lead to the same velocity profile in the far wake, and will thus have identical productions.

So how do this result compare with classic non-rotating theory? In the actuator cap analysis, we have an infinite number of blades, which cancels out tip effects. An infinite number of blades in the rotating case correspond to an infinite aspect ratio, for which induced drag is zero. Induced drag in the non-rotating case is the equivalent to tip-loss in the rotating case, so it would seem that there are more similarities between the rotating and non-rotating cases than was apparent at first sight.

4. An annoying mismatch with previous momentum theory results

As mentioned in the introduction, the result that in absence of tip effects, there is no effect on power production by use of winglets, is in direct contradiction to the findings of Van Bussel [9], which to the authors’ knowledge is the only previous work to be offering an insight into the physical understanding of winglets on wind turbine rotors. Van Bussel, based on his momentum theory analysis of the problem, concluded that the power augmentation by use of downwind winglets is due to the downwind shift of the wake vorticity. This had the implications, that the maximum obtainable power coefficient for a complete transfer of the load on the main wing to the winglet was found to be an increasing function of the downwind extension of the winglet, $C_{p_{\text{max}}} = 16/27 * f(L_{wl}/R)$, where $f(-\infty) = 0.5$, $f(0) = 1.0$, $f(0.1) = 1.1$, $f(1) = 3.4$, and $f(\infty) = \infty$, the upper (un)limiting value clearly being unphysical, and the result, that upwind winglets decrease production being in disagreement with the latter CFD investigations by Johansen[11-12].

In order to investigate where the mismatch between the results of the present work and the work of Van Bussel comes from, a different approach was taken. Consider the power production part from only the main rotor of the actuator cap in equation (4)

$$P_{AC,\text{Main}} = \rho \Gamma \Omega \int_{0}^{R} (V + V_{dz}(x,0))dx \quad (7)$$
If the axial velocity at the main rotor is approximated by the center-line value from the half-infinite tangential vorticity cylinder from Wilson [17] used in Van Bussel’s work, we arrive at the following result

$$P_{AC,Main} = 0.5 \rho \Omega R^2 V \left( 1 - \left( 1 - \frac{Lwl / R}{\sqrt{1 + (Lwl / R)^2}} \right)^2 a \right),$$  \hspace{1cm} (8)

where $a$ signifies the axial induction factor at the plane through the winglet tips. If the relation between the thrust coefficient and the circulation derived in Øye’s work (neglecting tangential induction)

$$\Gamma \Omega \cong C_T \pi V^2, \hspace{1cm} (9)$$

is used with the expression for the thrust coefficient in Van Bussel’s work (referred to as the axial drag coefficient of the rotor in that work)

$$C_T = 4 \left( 1 - \left( 1 - \frac{Lwl / R}{\sqrt{1 + (Lwl / R)^2}} \right)^2 a \right),$$  \hspace{1cm} (10)

in Equation 8, and the definition of the Power coefficient is employed, we arrive at the power coefficient from the main rotor

$$C_{P,AC,Main} = 4a \left( 1 - \left( 1 - \frac{Lwl / R}{\sqrt{1 + (Lwl / R)^2}} \right)^2 a \right)^2,$$  \hspace{1cm} (11)

which is exactly the result predicted by Van Bussel’s work, indicating that the power production part of the winglets themselves were not included in that work.

Based on these considerations, we find the results of that model to be erroneous, as likewise the explanation of the power augmentation by use of downwind winglets is due to the downwind shift of the wake vorticity.

### 5. Free-Wake Lifting Line (FWLL) algorithm

Now that the analysis using the simpler models have indicated that the effect of winglets is to reduce tip effects, we need to employ a model capable of modelling such effects if we are to estimate the maximum power production obtainable with winglets.

For this task a Free-Wake Lifting Line (FWLL) method was developed. The reason for this is that such a method is better suited to find the optimal loading for a given rotor geometry (winglet) using numerical optimization than a vortex lattice panel code or CFD, since for these codes, the load can be changed both by changing chord distribution and changing the twist distribution.

In the new Free-Wake Lifting Line method, the wings are represented by concentrated line vortices, from which shed vorticity emanates into the wake. The method does not take into consideration the actual local geometry of the wing cross-section (airfoil shape), but models only the effect of the circulations that the airfoils generate. This makes this method well suited for investigation of mechanisms and optimization of aerodynamic loads. The inviscid lift forces from the fluid on the wings are evaluated from the Kutta-Joukowski Theorem, $L = (\Delta x \cdot \rho) \cdot V_{rel} \times \Gamma$, using the relative velocity of the flow with respect to the wings, including also the contributions from the free wakes of the wings. Since viscous effects are not naturally a part of a lifting line algorithm, they are taken into account separately. The local drag forces act in the direction of the relative flow direction, and the magnitudes are obtained from the lift forces using 2D lift-to-drag ratios of the airfoils.

A new algorithm for evaluating the “induced forces” from the velocities that the non-planar bound vorticity induce on themselves was formulated. In order to avoid the lifting line singularity on itself, this “extra” induced velocity is computed using a vortex-lattice type grid. The self induced velocities are obtained using an iterative scheme using a weighted difference between 3D and 2D self-induced
velocities. From the converged self induced velocity, the induced forces are computed using the lifting line. The self-induced forces in the case of winglets straight up- or down-wind produces negligible changes in power production. A non-negligible effect on power production from this term is only seen in cases where the winglets have sweep, i.e. where the winglets are tilted ‘forward’ or ‘backward’ (a Y component of the winglet on the vertical wing in Figure 1). Since the winglets considered in the present work have no sweep, the self-induced effect will not be discussed further.

In order to determine the shape of the free wake, a steady-state free wake method was adopted. Due to the inherent unstable nature of free wake methods for wind turbine applications, some care must be taken to obtain converging solutions. Since the free-stream velocity is constant, and the turbine is assumed not to be operating in yawed conditions, only the vortices from one wing need be updated; the other ones are obtained from symmetry conditions. In order to ensure adequate resolution of the wake, the position of the wake is determined in specific planes parallel to the rotor plane, with narrow spacing near the rotor plane and increased spacing further down the wake (\(\Delta Z=0.005R\) at the rotor disc and \(\Delta Z=0.018R\) at \(Z=3R\) using 224 cross-sections). In the first part of the wake (up to 3R), the wake is updated freely, and the wake velocities are evaluated at the intersection between the linear vortex elements. In the second part of the wake, the vortex strings keep constant radial distance to the rotational axis, and the azimuthal positions are obtained from extrapolation of the values at the end of the free wake zone. The last zone is a half-infinite vortex tube to model the effect of the far wake. The positions of the free wake are updated for one cross-section at the time, and the differences in radial and azimuthal positions are convected to all downstream coordinates of the wake after updating all positions at that specific axial position. In order to avoid stability problems with the free wake method, the cross-sections are not updated in the typically parabolic marching fashion, but according to a scheme that ensures that the update cross-section position varies as much as possible in space while still covering all cross-sections in the free wake domain during a single global iteration. This scheme is an adoption of the integer sequence A049773 in the on-line encyclopedia of integer sequences [18]. Furthermore, relaxation is employed to facilitate a more stable numerical behavior. The relaxation coefficient used for the present computations was set to 0.6. The wake vorticity is modeled by rectilinear vortices with a viscous Rankine vortex core: 0.01R at rotor disc going toward 0.05R exponentially with a half-distance of 2R. The results shown in this paper were all obtained with the bound vorticity along the wings discretized in 40 elements, with finer resolution towards root and tip where gradients are steeper. Investigations of the discretization have shown that the present setup produces results that changes only marginally by further increasing resolution.

The integral forces and dimensionless numbers are obtained from integration of the total distributed forces. In order to maximize \(C_p\) values, Nelder-Mead simplex optimization was used in combination with a 3rd order cubic spline representation of the bound circulation to reduce the number of optimization variables.

A validation of the main part of the code is found in Johansen [19], where comparison of a priori results obtained with the current code and an actuator disc code is made with results from the full 3D CFD code EllipSys3D[20-22] on an aerodynamically maximum efficient rotor (\(R=63m\), \(\text{TSR}=8\), \(\text{Cl/Cd}=110\) (corresponding to the Risø B1-15 airfoil at \(\alpha=8^\circ\), which operates at \(\text{Cl}=1.4\))). As is evident from Table 1 and Figure 3, the agreement between the 3D CFD results (black lines) and the present method (red lines) is excellent in the non-winglet case.

Table 1: Comparison of integral FWLL and 3D CFD results on the aerodynamically maximum efficient wind turbine without winglets. \(\text{TSR}=8\), \(\text{Cl/Cd}=110\)

|            | \(C_p\)     | \(C_t\)     |
|------------|-------------|-------------|
| EllipSys3D  | 0.515       | 0.872       |
| FWLL       | 0.514 (-0.2%) | 0.868 (-0.5%) |
6. Results from the FWLL algorithm

The results from the FWLL algorithm shown in this paper are all based on turbulent 2D data from a Risø-B1-15 airfoil, for which the design point is at $\alpha=8^\circ$, resulting in $Cl/Cd=110$ and $Cl=1.4$. Furthermore a tip-speed ratio of TSR=8 is used throughout. The initial guess for the bound circulation is the circulation of the baseline non-wingletted optimal rotor investigated in [19]. It was found that the final converged optimal solution for the optimal circulation was rather close to this value for the rather small winglets investigated in this work.

The computational results with winglets show that the obtainable increase in power by use of winglets is somewhat higher for a downwind winglet than for the corresponding upwind winglet length, Figure 4. The reason for this is not clear to the authors, but the trend is clear.

Furthermore, the computations show that increasing the length of the winglets decreases the fraction of power available with the winglet compared to the same increase in tip radius. An interesting result from the computations is that for relatively small winglets compared to the tip radius (~0.01), the increase in power corresponds to what could be obtained by increasing the length of the
plane wing with the length of the winglet. Figure 5 shows the corresponding increases in thrust coefficient for the simulated cases.

![Figure 5: Left: Ct as function of winglet length for the cases shown in Figure 4. Right: fractional increase in Ct compared to the non-wingletted case. Downwind: Circles, Upwind: Stars, Gray curve: Radius extension, Dashed green line: 8/9. TSR=8, Cl/Cd=110.](image)

As seen, the increase in Ct is of the same order as the increase in Cp. The difference between upwind and downwind configurations is greater for Cp than for Ct, so based on these results, a downwind winglet is preferable over the upwind one.

Computations with changes in the geometry of the junction between the main wing and the winglet were performed. Computations with 0% (a kink), a 25% and a 50% bend radius/Lw1 were performed for the 2% downwind winglet case. No significant difference between the 0% and the 25% cases were found, and only a slightly lower Cp was found for the 50% bend case.

7. Comparison with CFD results
In order to finally assess the performance of the FWLL code results, a rotor was designed from the found optimal rotor. This can be done because both the bound circulation and the velocity field are known from the computations. The wing was laid out using 2D data from the Risø-B1-15 airfoil, as described previously.

The Navier-Stokes (NS) solver, EllipSys3D is used for the full 3D CFD computations. The code is developed by Michelsen[20-21], and Sørensen[22] and is a multiblock finite volume discretization of the incompressible Reynolds Averaged Navier-Stokes equations in general curvilinear coordinates. The code uses a collocated variable arrangement, and Rhie/Chow interpolation is used to avoid odd/even pressure decoupling. As the code solves the incompressible flow equations, no equation of state exists for the pressure and the SIMPLE algorithm is used to enforce the pressure/velocity coupling. The EllipSys3D code is parallelized with MPI for executions on distributed memory machines, using a non-overlapping domain decomposition technique. Solution of the momentum equations is obtained using a third order quadratic upwind interpolation scheme (QUICK) for the convective terms. All computations are performed assuming steady state conditions with a moving mesh technique based on analytical prescribed rotation[23]. The turbulent eddy viscosity is modeled using the k-ω SST[24]. The surface mesh is generated using Gridgen; a commercial mesh generator developed by Pointwise, Inc. and consists of 108 blocks of 32^2 cells for the entire rotor. The volume mesh is generated using Risøs in-house grid generator HypGrid[25] and away from the surface 128 cells are used resulting in 14.2·10^6 cells in total. The grid density used is based on more than 10 years of experience in rotor computations. The outer boundary of the computational domain is placed approximately six rotor diameters away. On the entire surface of the rotor geometry no-slip boundary conditions are used. On the outer boundary of the computational domain inflow velocity is assumed...
constant with zero shear and a low turbulence intensity, while zero axial gradient is enforced at the outlet.

Figure 6 show streamlines, surface pressures and iso-vorticity contours of the CFD computations.

Figure 6: Left: Streamlines and surface pressure plot on the outer part of the wing. Right: iso-vorticity contours.

From Figure 6 it is seen that there is no separation of spanwise flow patterns, indicating 2D flow over the wing at all positions. The coloured surface in the left hand side figure signifies surface pressure. The blue color is low pressure, and red is high. The wing was designed to operate at “2D” angles of attack at $\alpha=8^\circ$ on the whole rotor because that is the design point of the airfoil used in the computations. The surface pressure plot reveals that there is a lower pressure at the suction peak of the outer part of the main wing than on the winglet. This is surprising, since the relative velocity in the region near the tip is fairly constant, and should therefore translate itself into constant suction peak values because the values of the pressure coefficient should only depend on the angle of attack for a given airfoil.

This indicates that the tip is too lightly loaded. This is also seen on the iso-vorticity contour, where it is evident that more vorticity is shed from the intersection between main wing and winglet than any other place on the outer part of the wing. This does not correspond with the bound circulation distribution used to design the wing with. It is not clear to the authors whether this is due to effects of the chord near the tip, or if the radial flow is predicted differently in the FWLL and CFD codes. That investigation is an ongoing work. It is noted, however, that apart from the apparent change in pressure contours at the bend, the pressure contours and the length of the iso-vorticity surface look nice and continuous. Computations will be run with different offset in “winglet pitch” to try to get a smooth loading along the entire wing.

Table 2: Comparison of integral FWLL and 3D CFD results on the aerodynamically maximum efficient wind turbine with 2%t winglets. TSR=8, $C_l/C_d=110$

|            | $\Delta C_p/C_{pref}$ | $\Delta C_t/C_{tref}$ |
|------------|------------------------|------------------------|
| FWLL       | 2.47%                  | 2.61%                  |
| EllipSys3D  | 1.74%                  | 2.80%                  |

Table 2 show the increase in $C_p$ and $C_t$ predicted with the CFD and FWLL codes. Note however, that due to the unresolved issues brought up previously, the bound circulation distributions are not identical in the two cases. It is suspected that getting rid of the surprises at the bend will increase the $\Delta C_p/C_{pref}$ of the EllipSys3D computation closer to what is predicted by the FWLL code.
8. Conclusions and outlook
A summary of the main findings of this work is

- Yes, winglets can be used successfully to increase $C_p$
- The positive effect of winglets on power production is due to a reduction of tip losses, and is
  not connected with a downwind shift of wake vorticity.
- A new free wake lifting line code was developed, and comparisons with CFD computations on
  an optimal non-wingletted wing showed remarkable agreement between results.
- Results from the new code indicate, that:
  - Downwind winglets are more effective than upwind ones of the same length
  - The increase in power using winglets is smaller than what may be obtained by
    extending the wing radially with the same length
  - However, for small winglets (>2%), we get close to the same increase as for a
    corresponding increase in radius using downwind winglets
  - The results are not very dependent on the radius of the bend between the main wing
    and the winglet, but increases slightly for bend radii above 25% of the winglet height
    for the 2% downwind winglet case investigated
- A wing with 2% downwind winglet was constructed to get the same bound circulation as in
  the FWLL case. The wing was analyzed using the CFD code EllipSys3D:
  - The agreement between FWLL results and CFD results in the case with winglets is
    not as good as in the case without winglets
  - Due to this, the predicted increase in $C_p$ is lower for the CFD results
- Future work is needed to:
  - Explain what causes the difference in the CFD and FWLL predictions
  - Get a better estimate of the maximum power increase by use of winglets using CFD
    for a wing geometry that produces the predicted bound circulation distribution
  - Be able to run the FWLL code using 2D profile input data

References
[1] Kroo I. "Drag due to lift: Concepts for prediction and reduction", Annu. Rev. Fluid Mech. 2001. 33:587–617
[2] Munk M. “The Minimum Induced Drag of Aerofoils” NACA Report 121, 1923.
[3] Whitcomb RT. 1976. “A design approach and selected wind tunnel results at high subsonic
speeds for wing-tip mounted winglets”. NASA TN D-8260. Natl. Aeronaut. Space Admin.,
Washington, DC
[4] Jones RT, Jones RT, Lasinski TA. 1980. “Effect of winglets on the induced drag of ideal wing
shapes”, NASA TM 81230, Natl. Aeronaut. Space Admin. Ames Res. Cent., Moffett Field, Calif.
[5] Eppler R. “Induced Drag and Winglets” Aerospace Science and Technology, 1, p.3-15, 1997
[6] Maughmer M.D. “The Design of Winglets for High-Performance Sailplanes” AIAA paper
2001-2406
[7] Van Holten Th. “Concentrator systems for Wind Energy, with emphasis on Tip-vanes”, Wind
Engineering Vol.5, No.1, 1981.
[8] Shimizu, Y et al. Studies on horizontal axis wind turbines with tip attachments. EWEC ’90
Conference proceedings, Madrid, Spain, September 10-14 1990, pp. 279-283, Ass, Bedford,
U.K., 1990.
[9] Van Bussel, G.J.W.A momentum theory for winglets on horizontal axis wind turbine rotors and
some comparison with experiments Fourth IEA Symposium on the Aerodynamics of Wind
Turbines, November 1990, Rome, Italy. Proc. edit. K.McAnulty, ETSU-N-118, Harwell,
Didcot, U.K., January 1991
[10] Imamura, H. et al Numerical Analysis of the Horizontal Axis Wind Turbine with Winglets, JSME International Journal, Series B vol. 41 nr. 1, Feb 1998 JSME Tokyo Japan pp 170-176

[11] Johansen, J.; Sørensen, N.N.; Aagaard Madsen, H.; Wen Zhong Shen; Okulov, V. “Advanced rotor aerodynamics - including tip and root aerodynamics”. In: Bak, C. (ed.), Risø National Laboratory (DK). Wind Energy Department. Research in aeroelasticity EFP-2005. Risø-R-1559(EN) p. 11-17

[12] Johansen, J. and Sørensen, N.N. “Numerical Analysis of Winglets on Wind Turbine Blades using CFD”, EWEC 2007 Conference proceedings, Madrid, Spain.

[13] Müller R.H.G & Staufenbiel R "The Influence of Winglets on Rotor Aerodynamics” Vertica Vol. 11, No. 4, p.601-618. 1987

[14] Müller R.H.G “Winglets on Rotor Blades in Forward Flight – a Theoretical and Experimental Investigation” Vertica Vol. 14, No. 1, p.31-46, 1990

[15] Goldstein, S., "On the Vortex Theory of Screw Propellers", Proc. of the Royal Society (A) 123, 440, 1929.

[16] Øye S.,”A simple Vortex Model – using vortex rings to calculate the relation between thrust and induced velocity at the rotor disc of a wind turbine”, In Proc. of the third IEA Symposium on the Aerodynamics of Wind Turbines, ETSU, Harwell, 1990, p.4.1-5.15.

[17] Wilson R.E., “Wind turbine flow field model”; Trans. ASME, Vol. 108, Nov. 1986, pp. 344-345.

[18] The On-Line Encyclopedia of Integer Sequences. http://www.research.att.com/~njas/sequences/A049773

[19] Johansen, Madsen, Gaunaa, Bak & Sørensen. "3D Navier-Stokes Simulations of a rotor designed for Maximum Aerodynamic Efficiency”. AIAA-2007-217, presented Jan 2007 at Reno.

[20] Michelsen J.A. “Basis3D - a Platform for Development of Multiblock PDE Solvers.” Technical Report AFM 92-05, Technical University of Denmark, 1992

[21] Michelsen J.A. “Block structured Multigrid solution of 2D and 3D elliptic PDE’s.” Technical Report AFM 94-06, Technical University of Denmark, 1994

[22] Sørensen N.N. “General Purpose Flow Solver Applied to Flow over Hills.” Risø-R-827-(EN), Risø National Laboratory, Roskilde, Denmark, June 1995

[23] Sørensen, N.N., “Rotor computations using a ‘Steady State’ moving mesh”, IEA Joint Action Committee on aerodynamics, Annex XI and 20. Aero experts meeting, Pamplona (ES), 25-26 May 2005.

[24] Menter F.R. “Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows”. AIAA-93-2906, 1993.

[25] Sørensen N.N. “HypGrid2D a 2-D mesh generator”, Risø-R-1035(EN), Risø National Laboratory, Roskilde, Denmark, 1998.