Wind turbine blade tip comparison using CFD

E Ferrer and X Munduate
Wind Energy Department, CENER - National Renewable Energy Centre, Ciudad de la Innovacion 7, Sarriguren, Navarra, 31621, Spain
E-mail: eferrer@cener.com

Abstract. The effect of wind turbine blade tip geometry is numerically analysed using Computational Fluid Dynamics (CFD). Three different rotating blade tips are compared for attached flow conditions and the flow physics around the geometries are analysed. To this end, the pressure coefficient (Cp) is defined based on the stagnation pressure rather than on the inflow dynamic pressure. The tip geometry locally modifies the angles of attack (AOA) and the inflow dynamic pressure at each of the studied sections. However not all 3D effects could be reduced to a change of these two variables. An increase in loadings (particularly the normal force) towards the tip seem to be associated to a spanwise flow component present for the swept-back analysed tip. Integrated loads are ranked to assess wind turbine tip overall performance. It results from the comparison that a better tip shape that produced better torque to thrust ratios in both forces and moments is a geometry that has the end tip at the pitch axis. The work here presented shows that CFD may prove to be useful to complement 2D based methods on the design of new wind turbine blade tips.

1. Introduction
Wind turbine blade design mainly relies on Blade Element Momentum (BEM) based methods, using optimisation techniques to obtain optimal chord and twist distributions along the blade once appropriate aerofoils have been chosen. BEM methods are very fast and reliable in the design process, nevertheless these are limited due to their two dimensional nature. These codes require tabulated data for the lift, drag and moment distributions versus the angle of attack to calculate the blade aerodynamic loads. Furthermore, empirical corrections are necessary to account for rotational effects near the root and three dimensional flows around the tip region (e.g. Glauert [1]).

Over the last years, the wind turbine community has started to look at CFD methods to complement wind tunnel (e.g. IEA annex XX [2]) and in field tests (e.g. IEA Annex XIV [3]) on the understanding of the complex flow physics around rotating wind turbine blades. CFD codes are able to compute 3D and rotational effects and can be used to calculate all flow variables around a given geometry. These codes can thus prove useful on the calculation of aerodynamic coefficients required by engineering methods and on the explicit determination of loads since no corrections are necessary.

The NREL NASA-Ames (NREL phase VI) experiments have allowed comparisons between experimental data and CFD simulations with different codes (Sorensen et al. [4], [5], Duque et al. [6] or Le Pape et al. [7]). Within these works, good agreement between wind tunnel data and CFD prediction have been achieved; although limitations related to turbulence modelling
and transition modelling underlined. These previous works have been used as guidelines for the presented methodology and validation.

To the author’s knowledge limited research has been published on the use of CFD to design of new blade tips for wind turbines. Johansen and Sorensen (ref [8]) studied three different tips from the Tellus experiment at different wind speeds and extract some conclusions on how tapering and swept angle can influence blade loading. Conclusions include that the analysed swept tip had a earlier stall and that the tapered tip kept a higher loading. Recently the same authors have published interesting work on the influence of blade winglets on blade loads (ref [9]), but this topic is out of the scope of this work.

In this study, three wind turbine blade tips have been computed and the complex flow physics analysed using a full Navier-Stokes code. The work shows how CFD can complement BEM methods in the design process.

Firstly, the methodology used for the simulations is presented in section 2. Secondly, a brief summary of the validation previously undertaken for the NREL phase VI blade that used this methodology is shown. Thirdly, the tip geometries and the operating condition selected for the tip comparison are outlined in sections 4 and 5 respectively. Fourthly, section 6 shows the obtained results on pressure coefficients (Cp) and limiting streamlines. Followed by a discussion of the previously presented results. Finally, within section 7, design considerations are explained and integrated loads compared to summarise the overall blade topologies performance.

Comparison between BEM methods and CFD calculations were also performed and showed variability depending on the computed blade tip geometry. This study will not be presented herein as it is out of the scope of this study.

2. Methodology and numerics

Within the validation section and the tip comparison study, Fluent 6.2 was used to compute blade loads on the different blade geometries. To create the grids, Gambit (fluent’s mesher) was used.

Non-structured grids have been created using tetrahedral and prismatic elements containing approximately $4 \times 10^6$ elements (as shown in figure 1).

The operating condition used for all cases corresponded to those of attached flow for all configurations. Even thought, the resolution of the boundary layer (and viscous sublayer) is always recommended, reasonable results have been obtained when using wall functions in conjunction with the $k-\omega$ SST model, for attached flow conditions. Section 3 outlines some results from previous works (ref [10] and [11]) using the presented methodology.

Simulations have been performed using the Moving Reference Frame (MRF) capability of Fluent. To model flow periodicity, only one blade needs to be modelled and periodic boundary conditions can be used. A 180° sector was used for NREL phase VI validation case (two bladed rotor) and a 120° sector for the tip comparison (three bladed rotor). The presented results are fully-turbulent converged steady state runs using the $k-\omega$ SST model. Second order discretisation schemes were used for all variables and the SIMPLE algorithm selected to solve the pressure-velocity coupling. All simulations were run using fluent parallel capability in not less than 4 machines.
3. NREL phase VI validation for attached flow

The outlined methodology has been applied to predict the pressure coefficient distribution (Cp) and force distributions using the NREL NASA-Ames experiments for validation (see ref [2] for details). The simulated conditions correspond to the S sequence with a free stream velocity of 7 m/s and a rotational speed of 71.9 rpm. Figure 2 shows the Cp comparison between CFD and experimental data for the 63% radial station. The compared Cp values have been calculated using the sectional stagnation pressure. It can be seen that the computations are able to predict quite accurately the experimental Cp values at these locations. Figure 3 shows the comparison of the normal force coefficient along the blade. Discrepancies are seen for the 30% and 95% station and are thought to be due to the modelled geometry (e.g. tip not properly modelled, root-blade transition). Reasonable agreement is found for the intermediate stations.

![Figure 2. Pressure distribution for 63% radial location for the NREL phase VI blade](image1)

![Figure 3. Normal force coefficient vs r/R for the NREL phase VI blade](image2)

The results show that the \( k-\omega \) SST model, in combination with wall functions and Fluent’s MRF capability, gives reasonable results for rotating blades, as long as the flow remains attached.

4. Tip geometries

The tip study that follows consists on the comparison of three blade tip shapes and their aerodynamic behaviour. The blade was maintained identical up to \( r/R=90\% \), as well as the operating conditions. Only the outer part of the blade above \( r/R=90\% \) was varied for the three analysed tips. The twist distribution was maintained identical, for all geometries, up to \( r/R=100\% \). The three geometries are shown in figures 4, 5 and 6 along with their absolute pressure distributions on the suction side. The 92.5\%, 95\% and 97.5\% radial sections are plotted for further analysis. Within these figures the leading edge appears at the top of each figure, the trailing edge at the bottom and the tip end at the right side.

(i) **CASE 0** (figure 4): A square tip with no significant taper or sweep was retained as a baseline configuration. 3D effects are localised near the rear half of the tip end (above \( r/R=97.5\% \)), where the tip vortex impinges on the suction side, as seen on the pressure contours.

(ii) **CASE 1** (figure 5): A tip that has all sections located around the pitch axis (tip that ends at the pitch axis) was thought to be a representative shape of the current trend in the wind turbine community.

(iii) **CASE 2** (figure 6): A swept-back tip was shaped to have the trailing edges of all aerofoil sections aligned. This configuration has the aerofoils perpendicular to the pitch axis as the previous cases. This topology was selected to see if a redistribution of loading towards the tip could be obtained.
To elucidate the effect of aerofoil positioning, with respect to the blade pitch axis, the chord distribution has been maintained identical for cases 1 and 2. Therefore only 3D effects due to the aerofoil positioning are studied.

5. Operating condition
The operating condition corresponds to an averaged wind speed of 8.5 m/s and ensures that the flow is attached for all geometries. The following table 1 summarises the main considered aerodynamic parameters for the 95% and 97.5% radial sections. These are: the Reynolds number divided by the chord (Re/c) and the geometric angle of attack (geom-AOA). Both parameters are calculated taking into account the rotational velocity component and the free stream wind speed.

| r/R [-] | Re/c [1/m] | geom-AOA [deg] |
|---------|------------|----------------|
| 95%     | 4.19e6     | 5.9            |
| 97.5%   | 4.29e6     | 6.0            |

6. Pressure coefficients and spanwise flow component
Within the following section 6.1 the results from the three simulations are presented. An attempt to explain the different aerodynamic behaviours is made in the discussion section 6.2.

6.1. Results
This section presents, for the three cited cases and the defined operating condition, the computed results. Firstly, section 6.1.1 shows the pressure coefficients (Cp) for two sections, which are compared for the three geometries. Secondly, within section 6.1.2, streamlines are depicted to help on the understanding of the differences seen on the Cp distributions.
6.1.1. Pressure coefficients  

This section compares Cp distributions against the non-dimensional chord for the three analysed tips at two blade radial sections: $r/R=95\%$ and $r/R=97.5\%$. At this stage, it is important to remember that the aerofoils geometry and the twist distribution are identical for the three blades. Therefore only differences due to the tip geometry changes are visible. Absolute pressures have been divided by the stagnation pressures to obtain the retained Cp coefficient (following equation 1) for each blade and the considered radial section, instead of using the classical definition for Cp (equation 2).

$$C_p = \frac{(P - P_\infty)}{(P_{stagnation} - P_\infty)}$$  \hspace{1cm} (1)

$$C_p = \frac{(P - P_\infty)}{\frac{1}{2} \rho_\infty \left[ (\Omega \cdot r)^2 + V_\infty^2 \right]}$$  \hspace{1cm} (2)

The classical definition (equation 2) assumes that for the same section, all tips would see the same free stream dynamic pressure at the stagnation point and does not take into account the local induction factors that are modified by the local blade geometry and rotation. At a given station, the stagnation pressure varies depending on the tip shape. The retained definition shows the effective sectional loading, that would not be seen if the classical definition was used. With the retained definition, the Cp area (curve integral) is directly related to the sectional normal force coefficient.

Figures 7 and 8, which will be detailed later, show the pressure coefficient distribution along the non-dimensional chord for the three configurations. It is important, when looking at the Cp curves, that use the stagnation pressure (equation 1), to understand if the differences seen on the Cp distributions are due to a change in the stagnation pressure ($P_{stagnation} - P_\infty$) or to an actual change of the absolute pressures along the chord ($P - P_\infty$).

![Figure 7: Cp distributions vs x/c at 95% radial section.](image)

![Figure 8: Cp distributions vs x/c at 97.5% radial section.](image)

To account for the changes in the stagnation pressures, equivalent Reynolds ($Re/c$) number have been tabulated in table 2. The equivalent Re is calculated using the velocity resulting from the stagnation pressure.

2D Xfoil calculations were performed in order to obtain an order of magnitude of the local AOA that is experienced by the sections by comparing the Cp shapes. Although no exact equivalence was found between the 2D and 3D curves, a rough estimated AOA could be obtained. Figure 8 shows the Cp curve obtained with Xfoil and performed at a Reynolds number of 3.5
millions and an AOA of 3 degrees. Table 2 summarises these results for the two considered sections and the three cases.

| r/R [-] | Re/c [1/m] | estimated AOA [deg] |
|---------|------------|---------------------|
| Case 0  | 95%        | 3.64e6              | ~ 3 |
|         | 97.5%      | 4.02e6              | ~ 3 |
| Case 1  | 95%        | 3.45e6              | ~ 3 |
|         | 97.5%      | 3.43e6              | ~ 4 |
| Case 2  | 95%        | 3.47e6              | ~ 3 |
|         | 97.5%      | 3.37e6              | ~ 4.5 |

When looking at the tabulated results two approaches are possible. Firstly, one could be interested in the changes of these parameters for one blade along its span. Secondly, the interest could be centred in spotting the differences for a fixed spanwise location for the different tip geometries.

(i) Changes along the radius: It can be observed that the equivalent Reynolds number at the two considered sections, increases for case 0 when moving to an outboarder section, remains almost constant for case 1 and decreases for case 2. Table 1 in section 5 showed how the Re extracted from the freestream dynamic pressure increased with the radius as the component $\Omega r$ increased. However, the tip geometry seems to influence differently the stagnation pressure leading to differences on the calculated equivalent Re. When looking at the estimated angles, it appears that the change along the radius remains constant for case 0. Cases 1 and 2 show a different behaviour, the outboarder the section the higher the associated AOA. The tip that influences the most the local angle of attack, but the less the Re/c, when comparing to the undisturbed values (table 1) seems to be case 0. When comparing 7 and 8, it is seen that the Cp curves at 97.5% for case 0, has a smaller area (or normal force) than at 95%. This change is caused by a change in the stagnation pressure. On the other hand, case 1 and 2 have seen their areas increased for their outboarder stations.

(ii) Changes for a fixed radial station: At the 95% station it is shown that the Reynolds number is slightly higher (5.5%) for case 0 than for cases 1 and 2, that have almost the same Re/c. The estimated AOA is the same for the three tips at this station. Figure 7 shows that for station 95%, the Cp distributions are very similar for case 1 and case 2, and their corresponding equivalent Re and AOA are also very similar. The difference seen for case 0 is mainly due to the difference in the stagnation pressures (as seen for Re/c), the shapes of the curves being identical for the three distributions. At the 97.5% section, shown in Figure 8, different trends are seen. Case 0 has the higher Re/c value and case 2 the lowest. The AOA are also different, case 0 shows the lowest, and case 2 a slightly higher value than case 1. These changes in the local AOA are also revealed in figure 8, where the stagnation point is closer to the leading edge for case 0, and almost identical for cases 1 and 2. When comparing Cp for the 97.5% station (figure 8), a higher suction is observed on the case 2 tip in comparison with case 1 (this was also visible in the contour plot on figure 6 seen in section 4. Since the stagnation pressures (or Re/c) for these two blades are very similar (less than 1.7%) and their estimated AOA are very similar (~ 0.5deg), it can be
concluded that the evidenced changes are due to another effect that modifies their shape and particularly the suction side curve. This will be further analysed in the next section.

6.1.2. Streamlines and spanwise flow component  Figures 9, 10 and 11 show the limiting streamlines released on the suction side from the tip leading edge. The stations 92.5%, 95% and 97.5% and the pitch axis are also plotted.

![Figure 9. Case 0: Streamlines. Leading edge on top.](image)

![Figure 10. Case 1: Streamlines. Leading edge on top.](image)

![Figure 11. Case 2: Streamlines. Leading edge on top.](image)

The streamlines for case 0 have an outboard component. In case 1, the streamlines tend to follow the chord direction, although an inboard component is observed towards the trailing edge. The suction side streamlines for case 2 have a larger inboard component from the leading edge and do not follow the chord direction, which is the expected behaviour for a swept-back geometry.

Results are summarised in the next section, where conclusions on how the tip geometry influences Cp distributions are outlined.

6.2. Discussion
Within section 6.1.1 differences on the Cp plotted values became apparent. For the 97.5% section particularly various effects could be separated to explain the differences seen on the Cp curves for the three tips. Firstly, a given radial station sees different stagnation pressure values (Re/c) depending on the tip geometry. Secondly, the tip modifies the local AOA that the section sees, modifying the Cp shape and the stagnation point location. Indeed, case 0 showed less variation in AOA and Re/c when compared to the undisturbed values. The studied sections showed an almost 2D behaviour. Thirdly, another effect seems to be present when comparing case 1 and case 2 at $r/R = 97.5\%$ in figure 8, and particularly for the suction side. No significant change were evidenced on stagnation pressure values (or Re/c) and a minor change spotted in the AOA ($\sim 0.5\,\text{deg}$), however a higher suction was calculated for case 2. The streamlines plotted in figure 11 within section 6.1.2 and previously discussed, revealed that case 2 had an inboard radial velocity that was not seen for the two other cases. This component is thought to be responsible for the increased suction seen on the Cp curves. A possible explanation could be that the outboarder flow with higher momentum is being moved towards inboarder sections, which could lead to a local transfer of momentum that would give rise to a higher suction.

At this point, it can be mentioned that it would be of interest to decouple the influence of rotation from the geometric 3D effects.
To summarise, the presence of a spanwise flow component could be the explanation of the increase of local loading for the swept-back tip (case 2).

7. Design considerations
When projecting blade loads to the perpendicular rotor direction (Thrust) and on the plane of rotation (Torque), it is possible to establish the following design criteria. Generally, wind turbine blade design will concentrate in producing maximum power (maximum torque moment), while minimising hub loads (minimum thrust force and moment). The following section 7.1 exposes the radial distribution of forces along the tips. To finalise, section 7.2 analyses forces (F-Thrust and F-Torque) and moments (M-Thrust and M-Torque) to check the configuration that maximises torque forces and moments and minimises thrust forces and moments. It was thought convenient to use the torque to thrust ratios for forces and moments in order to assess the designed blades. The torsional moment (nose down) should be minimised to avoid torsional stresses that could produce structural failure.

7.1. Load distribution
Figure 12 and 13 show the distribution of forces perpendicular to the rotor plane and on the rotor plane respectively for the three tips. All sectional forces have been divided by the force at the 90% section for the case 0 tip (called Thrust$_{ref}$ and Torque$_{ref}$), so the maximum allowable value does not exceed 1. Case 0 (squared tip) has overall higher forces, and the interesting fact is that at r/R=100% the torque force becomes negative, which means that this section is actually acting against the rotational direction (braking) the blade. This is due to the strong tip vortex impinging on the suction side.

These figures show that the integrated forces are similar for cases 1 (tip at pitch axis) and case 2 (swept-back), but the moments are different. Case 2 has displaced the forces toward the tip, having more loading as r/R approaches 100%. Curves seem to cross at around the 95% station. These observations seem to agree with the presented Cp curves seen in previous sections.

![Figure 12. Radial distribution of Fluent thrust.](image1)

![Figure 13. Radial distribution of Fluent torque.](image2)

7.2. Integrated loads
In order to quantify the above discussed phenomena, integrated loads have been used to rank the three simulated tips.
Table 3. Integrated Forces and Moments for the three analysed configurations.

| Tip shape | F-Thrust [N] | F-Torque [Nm] | M-Thrust [Nm] | M-Torque [Nm] | M-Torque/F-Thrust | M-Torsional [Nm] |
|-----------|-------------|--------------|--------------|--------------|------------------|-----------------|
| CASE 0    | 9265        | 655          | 337259       | 23701        | 0.0707           | 1213            |
| CASE 1    | 6865        | 583          | 247854       | 21099        | 0.0849           | 747             |
| CASE 2    | 6974        | 579          | 251989       | 21093        | 0.0830           | 2086            |

Tables 3 summarise the integrated loads. Forces and moments have been integrated for r/R above 90%. Moments are calculated at hub centre. In order to calculate moments, each computational cell force has been multiplied by its corresponding distance and added together to obtain the integrated moments. Bold values correspond to maximum or minimum values chosen. As can be seen in table 3 the tip case 0 has the higher torque and thrust forces and moments that correspond to its wider surface (bigger chord distribution). If the local twist was changed to change the normal force into torque, and a similar study to the one performed by Hoerner in 1952 (ref [12]) to round the tip end was carried out, this configuration could become a suitable configuration. However, strong deflection and thus strong stresses are expected and should be taken into account by the structural engineers.

The tip shape that minimises thrust (force and moment) while maximising torque (force and moment) is case 1. This seems to be a more appropriate tip since it has the better ratios of torque over thrust for both forces and moments. Furthermore, this tip has the minimum torsional moment and will prove better from a structural point of view.

8. Conclusions
The presented results have helped the authors on the understanding of the flow physics around three rotating tip shapes. It has been proposed that the definition to be used for comparison of radial sections is the pressure coefficient using the stagnation pressure for the given section and tip shape, instead of the classical definition, since the first takes into account the influence of the local geometry. It has been shown that for attached flow conditions, the tip shape modifies the radial flow leading to 3D effects affecting the local loading that cannot be explained only by a change in local velocity (or equivalent Re/c) and AOA. A higher suction for the outboarder part of the swept-back tip was spotted. These increased loads and particularly the normal force component, may be explained by the presence of spanwise flow component towards inboarder sections for the swept-back tip. CFD calculations can therefore be used to create simple tip corrections to be implemented into 2D based methods that take into account a change in Re, AOA and local suction, depending on the tip shape.

Finally, integrated loads can help to rank new designs as long as appropriate objective functions are chosen. Within this study the ratios of torque over thrust of both forces and moments have been selected as objective functions. From this ranking, it resulted that the best tip shape is case 1, a geometry that has the tip end located at the pitch axis. This shape follows the current trend in the wind turbine industry, and resulted to be a good compromise between production and loading. Further studies may be necessary to conceive an optimal tip, however this study served as a preliminary comparison to detect the main important parameters that may be taken into account for future designs.
9. References

[1] Glauert H. *Airplane Propellers. In Aerodynamic Theory*, volume 4. Dover edition edition, 1963.

[2] Hand M, Simms D, Fingersh L, Jager D, Cotrell J, Schreck S, and Larwood S. Unsteady aerodynamics experiment phase vi: Wind tunnel test configurations and available data campaigns. Technical Report NREL/TP-500-29955, NREL, December 2001.

[3] Schepers J.G, Brand A.J, Bruining A, Graham J.M.R, Hand M.M, Infiled D.G, Madsen H.A, Paynter R.J.H, and Simms D.A. Final report of iea annex xiv: Field rotor aerodynamics. Technical Report ECN-C-97-027, ECN, May 1997.

[4] Sørensen N.N, Michelsen J.A, and Schreck S. Navier-stokes predictions of the nrel phase vi rotor in the nasa ames 80 ft x 120 ft wind tunnel. *Wind Energy*, 5(2-3):151–169, April-September 2002.

[5] Johansen J, Sørensen N.N, Michelsen J.A, and Schreck S. Detached-eddy simulation of flow around the nrel phase-vi rotor. In *EWEC*, Madrid, Spain, June 2003.

[6] Duque E, Burklund M.D, and Johnson W. Navier-stokes and comprehensive analysis performance predictions of the nrel phase vi experiment. In 22nd *ASME Wind Energy Symposium*, Reno, Nevada, January 2003. AIAA paper 2003-0355.

[7] Le Pape A and Lecanu J. 3d navier-stokes computations of a stall-regulated wind turbine. *Wind Energy*, 7(4):309–324, October-December 2004.

[8] Johansen J and Sørensen N.N. Numerical investigation of three wind turbine blade tips. Technical Report Ris-R-1353(EN), Riso National Laboratory, August 2002.

[9] Johansen J and Sørensen N.N. Aerodynamic investigation of winglets on wind turbine blades using cfd. Technical Report Ris-R-1543(EN), Riso National Laboratory, February 2006.

[10] Ferrer E and Munduate X. Cfd activities at cener. In *IEA Annex XX symposium*, Pamplona, Spain, May 2005.

[11] Ferrer E and Munduate X. Wind turbine blade tip design using cfd. In *Fluent user’s meeting*, Madrid, Spain, November 2006.

[12] Hoerner S. Aerodynamic shape of the wing tips. Technical Report AFTR-5752, United State Air Force, 1952.