Formation of tip-vortices on triangular prismatic-shaped cliffs.

Part 2: A Computational Fluid Dynamics study

A. Montlaur\textsuperscript{a}, S. Cochard\textsuperscript{b}, D. F. Fletcher\textsuperscript{c}

\textsuperscript{a}Escola d’Enginyeria de Telecomunicació i Aeroespacial de Castelldefels, Universitat Politècnica de Catalunya, Spain
\textsuperscript{b}School of Civil Engineering, The University of Sydney, Sydney, NSW, Australia
\textsuperscript{c}School of Chemical and Biomolecular Engineering, The University of Sydney, Sydney, NSW, Australia

Abstract

The flow over triangular-shaped cliffs is studied through Computational Fluid Dynamics simulations, using the RANS approach closed with the SST turbulence model. The formation of tip-vortices on the sides of the cliff is studied, taking into account the influence of the cliff geometry and the incoming wind. CFD results are compared with experimental data on 2D planes located above the cliff. Good agreement between the two is obtained once small experimental deviations from the nominal cliff geometry are accounted for in the simulations. In addition, 3D results, such as visualization of the vortices, allow a better understanding of the complex flow over such geometry to be achieved.

Keywords: wind turbines, steep topography, delta wing vortices, wind speed-up, Computational Fluid Dynamics, Shear Stress Transport model

1. Introduction

Lately, wind power has been growing fast as an energy source and the prospect of increasing its share in electricity production has given rise to significant interest in modeling wind turbines. Abe et al. (2005) and Porté-Agel et al. (2011), amongst others, have conducted Computational Fluid Dynamics (CFD) studies of wind turbines and wind farms. While the technical characteristics of a wind turbine are a very important factor for its efficiency, it is also fundamental to carefully choose its location. CFD studies have been
performed to investigate speed-up effects over escarpments by Mason et al. (2010) and Wood et al. (2001). Coastal and exposed hilltops are preferred development sites, taking advantage of speed-up effects. Nevertheless, some studies indicate that such sites can generate substantial turbulence, particularly lateral turbulence likely to damage wind turbines and to seriously impact wind energy forecasts (Ayotte, 2010).

A better understanding of the flow generated over complex topography as a first step in developing better site classifications is thus the motivation for the current study. Cochard et al. (2012) performed wind tunnel tests of flow over models made from generic 3D triangular elements arranged in sawtooth patterns. Flow changes with shape and wind direction were also studied. The present study, carried out with the commercial CFD solver ANSYS CFX13, aims to validate the experimental results of Cochard et al. (2012) and to present further and more complete results, such as 3D flow streamlines. Section 2 presents the main hypotheses considered for the CFD study. Section 3 first validates the CFD assumptions and compares CFD results with the experimental ones, and then studies the influence of the angle of the incoming wind on the flow over the cliff. Section 4 presents the main conclusions of this paper.

2. Test setup and methodology

The presented study aims to simulate the flow around a vertical triangular-shaped cliff, having a 90° included angle, see Figure 1. The flow is modeled as steady, with a Reynolds number of approximately 50,000 based on the height of the triangular cliff and on the inlet velocity at the height of the cliff. The fluid is air, considered incompressible and isothermal, at a temperature of 25°C. The computational domain is shown in Figure 1 and is $25h_{\text{Cliff}}$ long ($7.5h_{\text{Cliff}}$ in front of the cliff) and $10h_{\text{Cliff}}$ high, based on a cliff height, $h_{\text{Cliff}}$, of 0.1 m. Note that from now on, the following system of axes is used: the $x$ axis is orientated along the cliff, positive towards the back of the cliff, the $y$ axis is orientated laterally through the cliff, positive toward the right side of the cliff, and the $z$ axis is vertical, positive in the upwards direction. The origin of the system is taken at a 0 m height, in the middle of the cliff and at 100 mm from its leading edge. The zone of interest of this simulation is a rectangle plane located at $z = 0.1143$ m above the cliff and extending for $0.2$ m along the cliff, that is $-0.1$ m $< x < 0.1$ m and $-0.15$ m $< y < 0.15$ m. This zone is where PIV data from Cochard et al. (2012) are available.
The following boundary conditions were set for the numerical simulations, based on experimental data from Cochard et al. (2012). The boundary layer velocity profile can be approximated with the following logarithm law:

\[
\frac{u_{BL}}{u^*} = \frac{1}{0.4} \cdot \ln \left( \frac{z}{z_0} \right) \quad \text{with} \quad z_0 = 0.163 \text{ mm and } u^* = 0.377 \text{ m/s}
\]  

(1)

where \(z\) is the height above the wind tunnel floor in millimetres, \(u^*\), is the friction velocity, and, \(z_0\) is the roughness height. A turbulence intensity of 0.17, with an eddy viscosity ratio of 10, corresponding to the experimental data, were set as the inlet, with the previously defined horizontal velocity. Figure 2 shows the velocity profile (horizontal and vertical components) in a longitudinal plane in a stimulation without a cliff present. It can be seen that the velocity profile imposed at the inlet is reasonably well preserved. The top boundary was also set as an inlet with the same conditions as the previously defined inlet; that is, the horizontal velocity was specified at the top of the computational domain, setting it to a magnitude satisfying the \(u_{BL}\) relation. Note that several simulations with different computational domain sizes were run, to ensure that this location of top boundary did not affect the flow around the cliff. An average static pressure of 0 Pa was set at the outlet. The two lateral sides of the parallelepiped were set as periodic boundaries.
This hypothesis was validated by comparing the results with a triple cliff (side by side) configuration. No significant changes were observed as can be seen in Figure 12 in Section 3.2. The cliff surface was treated as a no-slip smooth wall, whereas the ground upstream the cliff was considered as a rough wall. The roughness height that must be specified, is the equivalent sand grain roughness height (ANSYS, 2010). Note that the sand-grain roughness is not equal to the geometric roughness height of the surface under consideration, since wall friction depends not only on roughness height but also on the type of roughness (Coleman et al., 1984). Here, the appropriate equivalent sand-grain roughness height was evaluated to be $0.005 \text{ m}$ corresponding to a physical roughness of around $0.163 \text{ mm}$, which corresponds to the wind tunnel setup values.

Note that as pointed out by Blocken et al. (2007), the accurate simulation of wall roughness is essential since unintended stream-wise gradients in the vertical mean wind speed profile may develop as the flow travels through the computational domain. Figure 2(b) shows that, in a simulation without a cliff present, the vertical velocity $w$ remains very close to $0 \text{ m.s}^{-1}$, the biggest amplitude detected being of the order of $0.03 \text{ m.s}^{-1}$, which is two orders of magnitude smaller that the vertical velocities observed in the region of interest of the flow around the cliff (see Figure 5(e)). This confirms that the flow over the cliff is not also affected by stream-wise vertical velocity development in the approaching boundary layer.

All numerical simulations were carried out with the commercial CFD solver, ANSYS CFX13 (ANSYS, 2010). CFX13 is based on a coupled finite
volume solver for the mass and momentum (and energy if required) equations. The numerical scheme uses a co-located pressure based method and a modified Rhie-Chow algorithm to avoid decoupling (ANSYS, 2010). The resulting algebraic equations are solved by an algebraic multi-grid method. All velocity calculations use a second order bounded differencing scheme while a first order upwind scheme is implemented for the convective terms in the turbulence equations, and a second order scheme is used for all diffusive terms.

The SST (Shear Stress Transport) model is used in this work. It combines the \( k-\varepsilon \) and the \( k-\omega \) models by way of a blending function (Menter, 1994). The standard \( k-\omega \) model and the transformed \( k-\varepsilon \) model are both multiplied by a blending function and both models are added together. The blending function is designed to be one in the near-wall region, which activates the standard \( k-\omega \) model, and zero away from the surface, which activates the transformed \( k-\varepsilon \) model. The SST model was designed to give highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients, by the inclusion of transport effects into the formulation of the eddy-viscosity (ANSYS, 2010). This results in a major improvement in terms of flow separation predictions. The superior performance of this model has been demonstrated in a large number of validation studies (Bardina et al., 1997). The curvature correction option was activated, given that vortices were expected to form at the cliff edges.

The computational mesh was composed of 1.3 million tetrahedral elements. An inflation mesh comprising 30 layers was used along the ground in front and on the cliff, with a first layer height of 0.1 mm. A body sizing of 4 mm was used in the region of interest, whereas a coarser body sizing of 20 mm was used around the cliff. Within the computational mesh, the coarsest element size was set to 70 mm. Values of \( y^+ \) of between 1 and 5 were obtained on top of the cliff. Mesh independence is presented in Section 3.1.

Two convergence criteria were set. First, the root mean square residuals were set to be below \( 10^{-4} \). Second, three monitor points were created at the height of interest \( z = 0.1143 \) m, one centered at the beginning of the cliff and two others further away along the cliff edge. The calculation was stopped when values of velocity at these points did not change anymore, using a tolerance of \( 5 \times 10^{-5} \) m.s\(^{-1} \).
3. Results

3.1. CFD validation and comparison with experimental data

The adequacy of the size of the computational domain was first validated. The length in front of the cliff was first extended by 50% from the original domain, then so was the height of the domain: in both cases no change in the results was observed. As previously commented, a simulation with three cliffs side by side was also run, in order to validate the assumption of periodic boundaries, for which some results are shown in Section 3.2. Mesh independence was checked as commented in Section 2. Figure 3 shows the profile of $u$, the $x$-component of the velocity, obtained along the cliff for three different meshes: the reference mesh described in Section 2, and a finer and a coarser mesh obtained respectively by multiplying all the mesh dimensions by 0.75 and 1.33. Although an important difference is noted between the reference and the coarser mesh, no relevant difference is observed between the reference and finer one, validating the chosen mesh. Figure 4 shows that the values of the dimensionless wall distance $y^+$ on top of the cliff are in an acceptable range.

Once the CFD settings had been validated, a comparison with the experimental results was carried out. Figure 5 shows the CFD and experimental velocity fields $u$, $v$, $w$ obtained at a height $h$ of 14.3 mm above the cliff. Both the experimental and CFD results for the three components of the velocity

![Figure 3: Profile of the $x$-component of the velocity obtained along the cliff (a) at a longitudinal cut ($y = 0$ m), and (b) at a lateral cut just after the leading edge of the cliff ($x = -0.095$ m), for three different meshes.](image)
show similar patterns, which are described in more details shortly. Nevertheless, while the CFD velocity fields are symmetric due to the properties of the flow and geometry, asymmetric velocity fields are clearly observed in the experimental data. Figure 6, which presents the experimental and CFD profiles of the $u$ component of the velocity in two different planes, emphasizes this asymmetry. Figure 6(a) demonstrates that both the experimental and CFD profiles of $u$ in the plane of symmetry of the cliff are in very good agreement. Figure 6(b), which presents $u$ at a lateral cut in the middle of the cliff ($x = 0$ m), confirms that in the centre of the cliff both the experimental and CFD results agree. It also shows that on the edges of the cliff some differences appear, along with a loss of symmetry in the experimental case. In particular a lower experimental $u$ velocity is observed above the right side of the cliff ($y > 0$), than on the left side. This asymmetry is also observed in Figure 5(b), where the $u$ velocity is further decreased on the right side of the cliff (away from the plane of symmetry) than on the left side.

This loss of symmetry can be explained by experimental uncertainties. First, it was verified that the velocity field is not very sensitive to small perturbations of yaw angle (around the $z$-axis) of the cliff. Then, CFD simulations demonstrated that the asymmetry can be explained by the fact that the experimental cliff is not perfectly horizontal but is inclined with a positive $1^\circ$ roll angle (around the $x$-axis) and a negative $1^\circ$ pitch angle (around the $y$-axis). Figure 7 shows the $u$ component of the velocity for a CFD simulation taking into account these two angle corrections. It can be seen in Figure 7(a) that, in the symmetry plane of the cliff there is no significant change with the ideal case of Figure 6(a), and in Figure 7(b) that, at a lateral cut in the middle of the cliff, the experimental and CFD results are now in much better agreement. The CFD simulation now captures the
Figure 5: Velocity fields with an incoming flow angle of $0^\circ$ at a height $h$ of 14.3 mm: (a) and (b) $u$ in the stream-wise direction, (c) and (d) $v$ in the cross-wise direction, (e) and (f) $w$ in the upward direction. CFD results are on the left, experimental results are on the right.

decrease in the stream-wise velocity $u$ on the right side of the cliff. After investigating the experimental setup, it was confirmed, that, once the cliff model is in place, the turntable, which is not sufficiently rigid, had a $1^\circ$ roll and pitch angle.
Figure 6: CFD and experimental profiles of \( u \) (x-component of the velocity) obtained along the cliff (a) at a longitudinal cut \( (y = 0 \text{ m}) \), and (b) at a lateral cut in the middle of the cliff \( (x = 0 \text{ m}) \).

Figure 7: CFD (taking into account pitch and roll angle corrections) and experimental profiles of \( u \) (x-component of the velocity) obtained along the cliff (a) at a longitudinal cut \( (y = 0 \text{ m}) \), and (b) at a lateral cut in the middle of the cliff \( (x = 0 \text{ m}) \).

Figure 8 shows the fractional speed-up ratio fields \( M_u, M_v, M_w \) and, on top of the speed-up contours, the vectors \( (M_u, M_v) \), for the CFD inclined cliff and for the experimental data. Following Cochard et al. (2012), the fractional
Figure 8: Fractional speed-up ratio fields $M_u$, $M_v$, $M_w$, and vectors $(M_u, M_v)$ with an incoming flow angle of 0° at a height $h$ of 14.3 mm: (a) and (b) $M_u$ in the stream-wise direction, (c) and (d) $M_v$ in the cross-wise direction, (e) and (f) $M_w$ in the upward direction. CFD results are on the left, experimental results are on the right.

The speed-up ratio $M(z = h)$, is computed as follows for a given height $h$

$$M(z = h) = (M_u, M_v, M_w)(z = h) = \frac{\nabla_{\text{Cliff}}(z = h + h_{\text{Cliff}}) - \nabla_{\text{BL}}(z = h)}{|\nabla_{\text{BL}}(z = h)|}$$

(2)
where $\nabla_{\text{Cliff}}$ is the mean velocity measured at a height $h$ above the cliff, $h_{\text{Cliff}}$ is the height of the cliff, and, $\nabla_{\text{BL}}$ is the mean velocity of the boundary layer measured at a height $h$ above the wind tunnel floor. Figure 8 confirms that the asymmetry of the experimental results is now well-captured by the CFD simulation. Figure 8(a) shows a fractional speed-up ratio $M_u$ between 0.5 and 1 along the symmetry axis of the triangle, demonstrating an increase of the stream-wise velocity of 50 to 100% and could, a priori, justify the installation of wind turbines on the top of the cliff. Away from the axis of symmetry of the triangle, and especially in the non-symmetric region on the right side of the cliff, the fractional speed-up ratio can have negative values down to $-0.5$, which means that the stream-wise velocity above the cliff is half the velocity of the free stream at the same relative height $h$. Clearly these are not good positions for a wind turbine as the power output is significantly reduced, and it is next seen that this is actually where vortices are created.

While the study of the fractional speed-up ratio $M_u$ shows where to take advantage of the speed-up effect, $M_v$ and $M_w$ give information about the vortices and thus indicate where a wind turbine should not be located. As commented by Cochard et al. (2012), the vectors and the $M_v$, $M_w$ fractional speed-up ratios, both in the CFD and experimental data, demonstrate the formation of two delta wing tip-vortices on the upper surface of the triangular cliff. Like the flow over a delta wing, as the flow approaches the cliff, it accelerates and moves up to pass the obstruction, as is clearly evident in Figure 8(a). $M_w$ shows this upward flow upstream of the cliff with a maximum value at the cliff edge (Figure 8(e) and 8(f)). Once the flow has passed the edge, two vortices start to form and to grow in a cone of around 30°. In Figure 8(c), $M_v$ displays the growth of the vortex cone both in the vertical and horizontal directions. Close to the tip of the triangle, the 14.3 mm high measurement plane only records an inward $M_v$ flow; the measurement plane is at the top edge of the vortices. Closer to the downstream end of the triangle, $M_v$ is of opposite sign, indicating an outward flow (from the centre of the triangle to the edge); the measurement plane is in the lower part of the vortices. The asymmetry is once again well captured, the vortex on the right side of the triangle being larger than the one on the left. Furthermore, the centre of rotation of the right vortex crosses the measurement plane about 30 mm upstream of the left vortex centre of rotation. Once the vortex centre of rotation crosses the measurement plane, the component $v$ of the velocity changes direction from inward to outward.

This section aimed to validate the CFD model used to simulate the flow.
past a triangular cliff. Once the reasons for the differences observed with the experimental results were identified, CFD and experimental data show very good agreement, and CFD can be used to complete the matrix of experimental results obtained by Cochard et al. (2012). From now on, only CFD results are presented and the *ideal* (non-inclined) cliff is considered.

3.2. Vortex evolution as a function of the incoming flow angle

The triangular cliff is now subjected to a flow with an increasing anti-clockwise yaw angle. Note that this yaw angle arises from changing the inlet velocity (not the orientation of the cliff). Figures 9 and 10 show the stream-wise and cross-wise fractional speed-up ratios $M_u$ and $M_v$ for various incoming yaw angles: from 5° to 45°. It can be seen that as the yaw angle increases, the windward vortex increases in size and strength, whereas the leeward one decreases. Until a yaw angle of 30°, a low vorticity middle zone region, with positive fractional speed-up ratio $M_u$, is still identified. Though this region would be a good candidate for a wind turbine, it gets narrower and closer to the windward vortex. As a consequence, turbines located in this area could cross the vortex edge and thus encounter strong velocity gradients and high vorticity. Above a wind angle of 30°, this region is reduced in size significantly.

Above 30°, most regions of the flow over the cliff show zero or negative fractional speed-up ratio $M_u$, making it extremely difficult to find a relevant location for a wind turbine. Note that, from 30°, on the 2D views of Figures 9 and 10, it is difficult to identify the two distinct vortices. The windward vortex is evident, especially in Figures 9(d), 9(e) and 9(f), but it is not clear if the leeward vortex is still formed. In order to obtain a better visualization of the vortices, Figure 11 shows the flow streamlines and the vortex cores, that is, a special type of isosurface that displays a vortex (ANSYS, 2010). Here the swirling strength method, which represents the strength of swirling motion around a local centre, with a value of 0.2 s$^{-1}$, has been chosen in all cases. Note that the swirling strength is the imaginary part of the complex eigenvalues of the velocity gradient tensor. It confirms that the windward vortex increases in size with the yaw angle. The streamline visualization also shows that the area between the two vortices, the area where the flow presents low vorticity, initially centered, is displaced leeward when the yaw angle increases. As previously commented, this low vorticity area is thus more and more reduced, but does not disappear completely. At high yaw
angle (from 30°), a smaller horseshoe vortex is created in front of the cliff on the windward side (Baker, 1980).

Figure 12 shows the velocity streamlines when a triple cliff configuration is considered. Only minor differences in the shape of a few streamlines can be observed between the three cliffs. Together with Figure 11, it also shows
that the vortices appear downstream until around one cliff length after the end of the cliff.

Note that 3D CFD results allow a better understanding of the tip-vortex formation at high yaw angle than the experimental data. The stereo-PIV data and Figure 10 could suggest that from 30° the windward and leeward
vortices merge. Nevertheless, observation of CFD streamlines and vortex cores shows that a low vorticity area remains between the two vortices, even at high yaw angle, though it is reduced in size dramatically, especially from 40°.
3.3. Pressure coefficient

The experimental and CFD obtained mean pressure coefficients $C_p$ on the top surface of the cliff for $0^\circ$ and $10^\circ$ yaw angles are shown in Figures 13. Note that the CFD mesh size used on top of the cliff (4 mm) is much finer than that used experimentally (pressure taps were positioned on a grid with a mesh of 25 mm), which is why the resolution of the experimental results is coarser. Nevertheless, both CFD and experimental results are in agreement, and show that the surface of the cliff divides into three zones: two vortices close to each edge of the cliff and a third region in between. For the case of an incoming flow at $0^\circ$ the coefficient of pressure is symmetric. For an incoming flow of $10^\circ$ the windward zone becomes larger and the leeward zone slightly narrower as anticipated from the flow field. At $10^\circ$, see Figure 13(c), the pressure coefficient shows a small kink, also observed in the shape of the vortex, see Figure 11(b). The reason for this is unclear.

4. Discussion

The 3D CFD simulations presented here provide a unique insight into the understanding of the complexity of the flow past triangular cliffs. The comparison of the CFD results with those from an equivalent experimental setup allowed us to:
detect the asymmetry in the experimental setup, due to the lack of rigidity of the turntable, which was estimated as an uncertainty of 1° in the pitch and roll angles in the experimental cliff,

• characterize the influence of the previously commented pitch and roll angles on the formation of the tip-vortices. In particular, it has been found that the lifted side of the cliff presents a bigger tip-vortex than the lower one.

Figure 13: Pressure coefficient \( C_p \) on the cliff surface for an incoming flow (a) and (b) at 0°, and, (c) and (d) at 10°. CFD results are on the left, experimental results are on the right.

With a pitch and roll angle of 1°, the CFD simulation results are similar to the PIV ones, and after investigation, it was confirmed that the experimental turntable presented 1° roll and pitch angles.

The 3D visualization tools of the CFD software also allowed a better interpretation of the results and complemented the experimental ones. The vortices produced at each side of the cliff were studied as a function of the incoming flow yaw angle, and it was observed that:
• the windward vortex increases in size and strength with the yaw angle, whereas the leeward one decreases,
• the area between the two lateral vortices, that is the area where the flow presents low vorticity, initially centered, is displaced leeward when the yaw angle increases,
• the low vorticity area is more and more reduced, but does not disappear completely, as seen in the streamline and vortex core visualization, as the yaw angle is increased.

5. Conclusions

This work presents CFD results of the simulation of the flow past triangular prismatic-shaped cliffs and its comparison with the equivalent experiment. Though, as expected, both CFD simulations and wind tunnel tests give similar results, CFD allowed us to detect and to quantify some uncertainties in the experimental setup. The CFD simulation especially highlights the sensitivity of the vortex formation to the experimental cliff roll and pitch angles.

This CFD simulation confirms that when an atmospheric boundary layer flow passes over triangular-shaped cliffs, two delta wing vortices are created on each side of the cliffs. These vortices are strong and quite sensitive to yaw angle. Small changes in wind direction not only lead to significantly different fractional speed-up ratios, but also influence the location of the vortices. The low vorticity region, where positive speed-up ratios are observed, is consequently displaced, and reduced in size when high angles of the incoming flow are considered. This study shows that, for wind energy considerations, it is not an easy task to determine a region that would allow a wind turbine to take full advantage of potential speed-up effects, independently of the incident flow.

References


