Comparison of four different models of vortex generators

Fernandez, U.; Réthoré, Pierre-Elouan; Sørensen, Niels N.; Velte, Clara Marika; Zahle, Frederik; Egusquiza, E.

Published in:
Proceedings of EWEA 2012 - European Wind Energy Conference & Exhibition

Publication date:
2012

Document Version
Publisher's PDF, also known as Version of record

Link back to DTU Orbit

Citation (APA):
Comparison of four different models of vortex generators

U. Fernández¹ (unai.fernandez@ehu.es), P.-E. Réthoré² (pire@dtu.dk), N. N. Sørensen² (nsqr@dtu.dk), Clara M. V.² (cmve@dtu.dk), F. Zahle² (frza@dtu.dk) and E. Egusquiza³ (egusquiza@mf.upc.edu).

¹University of the Basque Country, Nuclear Engineering and Fluid Mechanics Department, Nieves Cano 12, 01006 Vitoria-Gasteiz, Araba, Spain
²DTU Wind. AED, Risø, Building VEA 118, Frederiksborgvej 399, DK-4000 Roskilde, Denmark
³Technical University of Catalonia. Fluid Mechanics Department, Av. Diagonal, 647 08028 Barcelona Spain

Abstract

A detailed comparison between four different models of vortex generators is presented in this paper. To that end, a single Vortex Generator on a flat plate test case has been designed and solved by the following models. The first one is the traditional mesh-resolved VG and the second one, called Actuator Vortex Generator Model (AcVG), is based on the lifting force theory of Bender, Anderson and Yagle, the BAY Model, which provides an efficient method for computational fluid dynamic (CFD) simulations of flow with VGs, and the forces are applied into the computational domain using the actuator shape model. This AcVG Model enables to simulate the effects of the Vortex Generators without defining the geometry of the vortex generator in the mesh and makes it easier for researchers the investigations of different vortex generator lay outs. Both models have been archived by the in house EllipSys CFD code using Reynold-Average Navier-Stokes (RANS) methods. The third model is the experimental one, where measurements were carried out in a low speed closed-circuit wind tunnel utilizing Stereoscopic Particle Image Velocimetry (SPIV) with a single vortex generator positioned on a vertical wall in the center of the test section. The fourth model, used as a quantitative comparison, is the analytical model of the primary vortex based in the helical structure of longitudinal embedded vortex, which can reduce the complex flow to merely four parameters: circulation, convection velocity, vortex core radius and pitch.

The goal of this article is to validate the AcVG Model compared with a fully meshed VG, a wind tunnel experiment and an analytical VG model.

Keywords

Vortex Generators, actuator shape model, CFD, computational fluid dynamics, BAY Model

1. Introduction

A Vortex Generator (VG) is considered as a passive flow control device which modifies the boundary layer fluid motion bringing momentum from the outer region into the inner region. Through this transfer of energy, the velocity of the inner region is increased at the same time as the boundary layer thickness is decreased, which in turn causes the separation of the flow is delayed, Rao et al. [1]. Furthermore, Lin et al. [2] showed the Drag reducing and the Lift increasing effect of sub boundary layer VGs.

Vortex Generators are applied on wind turbine blades with the major aim to delay or prevent the separation of the flow and to decrease roughness sensitivity of the blade. They are usually mounted in a spanwise array on the suction side of the blade and have the advantage that they can be added as a post-production fix to blades that do not perform as expected. So, adding VGs is a
simple solution to improving the performance of a rotor, Schubauer et al. [3] and Bragg et al. [4].

In order to design a wind turbine blade, and to optimize the position of the VGs on the blade, Computational Fluid Dynamics (CFD) tools can be used. However, modelling the fully-meshed VGs on a full rotor computation becomes prohibitively expensive. Indeed, the Vortex Generators size is often similar to the boundary layer thickness and many small cells are needed in the VG geometry in order to have a reliable modelling of the flow. An alternative way of modelling VGs in CFD is to model the influence of the vortex generator on the boundary layer using body forces. Bender et al. [5] developed a source term model based on the Joukowski lift theorem and thin airfoil theory, called the BAY Model. This model was presented for simulating vane Vortex Generators in a finite volume the Navier-Stokes code that eliminates the requirement to define the geometry in the mesh. For the calibration of the model, a test case was created by [5] for comparison of the results with a modelled VG and the gridded VG. This test case consisted in a pipe with 24 VGs mounted circumferentially in a co-rotating configuration. The study showed to promising results.

Subsequently, a new improved version of the BAY model was developed by A. Jirásek [6], called jBAY Model. This new version was based on the lift force theory of [5] and provided a more capable method for simulating the flow with rows of VGs. Jirásek [6] used a simplified technique for defining the model control points, so in this way it was easier to implement the model and the results were more accurate. The model was tested with a single VG on a flat plate, in an S-Duct air intake in a high-lift wing configuration. The results showed very good agreement between experimental data and CFD computations. Afterwards, an empirical model of VGs was incorporated into the Wind-US Navier-Stokes CFD code by Dudek [7] and in 2011 a simplified implementation was developed by Dudek [8]. With the implementation of the BAY model in the CFD code, the effects of the VGs using fine mesh are simulated by adding lift forces in the region of cells at the VG position. With this simplification the reduction of mesh cells can reach 30% of the fine mesh needed with a resolved VG, Wallin [9].

2. Models Description

Four VG Models have been performed in this work for a detailed comparison, both qualitative and quantitatively. In order to carry out this comparison, a test case based on a single VG in a flat plate was designed. The proposal of this research has been divided as Figure 1 shows.
The computations of the first two models, mesh-resolved VG and AcVG model, were performed using the EllpSys CFD code Michelsen [10] and Sørensen [11], which is an in house structured finite-volume CFD package for the numerical simulations of flows using Reynolds-Averaged Navier-Stokes equations.

These two CFD methods are compared with a wind tunnel experiment. In this experiment, a parametric study was performed over a single vane placed on the test section wall in a low-speed wind tunnel. The flow was recorded using Stereoscopic Particle Image Velocimetry, in cross-planes at various positions downstream of the vane, providing instantaneous three-component realizations throughout the measurement plane. This enables an overview of the averaged downstream development of the wake, including both velocity field and streamwise vorticity, suitable for comparison with computations. The experimental conditions and setup are the same as described in Velte et al. [12].

Finally an analytical model of the primary vortex is considered in the context of the two CFD models and the wind tunnel experiment. The model described in this work, which can reduce the complex flow to merely four parameters (circulation, convection velocity, vortex core radius and pitch), enables quantitative comparison in addition to the qualitative one, Velte [13].

In every test case the measurements have been conducted in a spanwise plane, in a plane normal to the flat plate, positioned five VG heights downstream of the vortex generator.

2.1 Mesh-Resolved VG Model.

This model consists in a single VG on a flat plate and the computational domain has been defined with the following normalized dimensions with the VG height, as shown in the Figure 2. The flow domain width is 30 times the VG height and the height is 5 times. The flow domain length is 32 times the VG height in order to capture the generated vortex.

![Figure 2: Computational domain of mesh-resolved VG Model.](image)

The geometry dimensions of the rectangular VG are defined with a length of two times the VG high, see Figure 3. The thickness of the vane is constant and with no sharp edges. A boundary layer is developed over the flat plate, forced by the viscous interaction between the wall and the flow. The VG was positioned in the flat plate in such way that the boundary layer thickness at this location is equal to the VG high.

![Figure 3: VG Dimensions.](image)

The angle of attack defined is 20 degrees, see Figure 4, and the Shear Stress Transport SST turbulence model has been chosen due to its ability to solve swirling flows Liu et al. [18].

![Figure 4: Angle of attack.](image)

The Reynolds Number based on VG height is:

\[
Re = \frac{\rho u_c H}{\mu} = 833 \quad (2.1)
\]

The computational setup of the fully-meshed VG Model consists in a block structured mesh of 18 million cells with the large part of them used to capture the vortex generated downstream the VG, Figure 5. Five levels of mesh size have been defined for this model.
and a mesh dependency of less than 5% has been detected.

Figure 5: Mesh Section on the VG.

2.2 Actuator VG Model.

In the wind turbine, VGs are often used to improve the performance of the blades by minimizing the effects of the boundary-layer separation and the adverse pressure gradients. So, computational fluid dynamics (CFD) methods are used to simulate the flow and to predict the blade performance. Together with experiments in wind tunnel, it is a very useful tool for parametric studies of VG lay-out, however, these CFD methods are very time consuming in the computations and in generating a high level quality mesh.

In this work we introduce the Actuator Vortex Generator Model (AcVG) based on the Bay Model, developed by Bender et al. [5]. The main idea of the BAY Model is to replace the VG geometry by a subdomain at the original VG location and to apply the force distribution in this region, as shown in figure 6. The influence of the VG onto the flow is modelled through a source term normal to the local flow direction.

\[ \Delta (\rho u) = \sum F S_j + L_i \]  
\[ \Delta (\rho E) = \sum F E S_j + u L_i \]  

The BAY Model incorporates a source term in the momentum and/or energy equations where VGs are taken in account through the body forces exerted on a fluid.

\[ Li = c_{\text{tg}} S_{\text{tg}} \frac{V_i}{\sum V_i} \alpha \rho u^2 G \]  

The source term applies a force normal to the local flow direction, parallel to the surface which simulates the side force generated by a VG.

\[ \text{Total Force} \]

Figure 6: BAY Model source subdomain on a flat plate.

Figure 7: Calibration of the ct parameter.

A Mesh dependence study for Actuator VG Model has been performed with 3 different levels of grid. Results from the fine mesh are compared with the results with the coarse and medium mesh, Figure 8.
The procedure has been achieved by using the Richardson Extrapolation Method for a mesh dependency study, Stern et al [15]. Three parameters are calculated in the Richardson Extrapolation: p, R and RE, the order of accuracy, the error ratio and the extrapolated solution respectively. A fine, medium and coarse mesh are defined with the corresponding mesh sizes h1, h2 and h4. A mesh dependency around 2% has been detected.

The innovation of this research is that instead of applying forces in all cells of the subdomain, as the BAY Model, the force is applied in cells just in the outline of the VG geometry, see figure 2. The forces are applied into the computational domain using the actuator shape model presented in Réthoré et al. [16]. The body forces are applied in the domain using a modified Rhie-Chow algorithm presented in Réthoré and Sørensen [17], using the EllipSys CFD code, Michelsen [10] and Sørensen [11]. The Actuator VG Model has been designed to be user-friendly. Within the EllipSys CFD code, the user only specifies the following parameters for each VG to be modelled: the cells where the model will be applied and the angle of the incidence of the VG.

2.3 Experimental Data Model.

Consider the test section setup in Figure 11, the measurements were carried out in a closed-circuit wind tunnel with an 8:1 contraction ratio and a test section of cross-sectional area 300 × 600 mm with length 2 m. At the inlet of the test section, a turbulence-generating grid with mesh length 39 mm was situated.

The experiments were conducted at free stream velocity \( U_\infty = 1.0 \, \text{m s}^{-1} \). The wind tunnel speed was obtained by measuring the pressure drop across an orifice plate. The turbulence intensity at the inlet from laser doppler anemometry (LDA) measurements has been found to be 13 %. The boundary layer thickness at the position of the vortex generator has been estimated from LDA measurements to be approximately \( \delta VG = 25 \, \text{mm} \). The actuator, as seen in Figure 11, is a rectangular vane of the same height as the local boundary layer thickness, \( h = \delta VG \), with a length of 2h. The vortex generator was positioned on a vertical wall in the center of the test section with its trailing edge 750 mm downstream of the inlet grid when it is at zero angle to the mean flow.

The measurements were conducted in a spanwise plane, with plane normal parallel to the test section walls, positioned five device heights downstream of the vortex generator. The measurement plane has been indicated by a dashed line in Figure 11.
The SPIV equipment was mounted on a rigid stand and included a double cavity New Wave Solo 120XT Nd-YAG laser (wavelength 532 nm) capable of delivering light pulses of 120 mJ. The pulse width, i.e., the duration of each illumination pulse, was 10 ns. The light-sheet thickness at the measurement position was 2 mm and was created using a combination of a spherical convex and a cylindrical concave lens. The equipment also included two Dantec Dynamics HiSense MkII cameras (1344 × 1024 pixels) equipped with 60 mm lenses and filters designed to only pass light with wavelengths close to that of the laser light.

2.4 Analytical VG Model.

The existence of Lamb-Oseen reminiscent vortex structures embedded in wall bounded flow has been reported in various experiments and numerical simulations, Liu et al. [18]. For the Lamb-Oseen vortex, the vorticity is non-zero only for the axial component as

\[ w_r = 0; \quad w_\theta = 0; \quad w_z = \frac{\Gamma}{2\pi^2} \exp \left( -\frac{\rho^2}{\varepsilon^2} \right) \]  

A more general model is the Batchelor vortex [19], which includes the non uniform axisymmetrical axial velocity distribution \( u_z \) which approaches the Lamb-Oseen vortex in the extreme. This vortex model is commonly used in instability studies of swirling flows Heaton et al.[24]. To describe experimental swirl flows the Batchelor vortex model is usually referred to in the form:

\[ u_\theta = \frac{K}{r} (1 - \exp (-\alpha r^2)) \]  

\[ u_z = W_1 + W_2 \exp \left( -\alpha r^2 \right) \]

where \( \Gamma \) is the vortex strength (circulation), \( l \) is the pitch of the helical vortex lines, \( u_\theta \) is the advection velocity of the vortex and \( \varepsilon \) is the effective size of the vortex core with Gaussian axial vorticity distribution (see Figure (12)).

The profiles given in (2.6) can reproduce experimentally determined swirl flow with high accuracy. One possible approach is to test if the empirical model (2.6) can describe the longitudinal vortex in the present case. However, in accordance with Pierrehumbert [23] one needs to account for the possible disturbance of the mirror vortex, resulting from the presence of the wall. Another more suitable approach is therefore to extend the Batchelor vortex model to model the flow by helical symmetry of the vorticity, leaving no restrictions on the shape of the vortex core. Flows with helical vorticity can be described by correlation between the axial and circumferential vorticity vector components:

\[ \omega_r = 0 \]  

\[ \omega_\theta = \frac{r \omega_z}{l} \]  

\[ \omega_z = \frac{\Gamma}{\pi \varepsilon^2} \exp \left( -\frac{r^2}{\varepsilon^2} \right) \]
Based on the experimental observation the simple Batchelor vortex model is chosen as

\[
\begin{align*}
\Gamma &= \frac{\Gamma}{2\pi R} \left[ 1 - \exp \left( -\frac{R^2}{\varepsilon^2} \right) \right] \\
\Gamma &= \frac{\Gamma}{2\pi R} \left[ 1 - \exp \left( -\frac{R^2}{\varepsilon^2} \right) \right]
\end{align*}
\]

(2.7a, 2.7b)

The only requirements of this simple model are the size of the vortex core, the circulation, the helical pitch and the vortex advection velocity.


Results of a single gridded VG on a flat plate and the AcVG Model were performed using the EllipSys CFD code. Figures 13(a) and (b) show the evolution of the vortex downstream of the VG for the mesh-resolved VG Model and for the AcVG model respectively.

3.1 Qualitative Comparison

For a qualitative comparison between the mesh-resolved VG Model and the AcVG Model, four parameters have been chosen: pressure, axial velocity, vorticity and turbulent kinetic energy. All of these fields have been taken at the calibration distance of five VG heights downstream the trailing edge of the VG and plotted in the Figure 14.
Figure 14: Fields for qualitative comparison. Mesh resolved VG Model on the left and AcVG Model on the right. (a) Pressure Fields, (b) Axial Velocity Fields, (c) Vorticity Fields, (d) Turbulent Kinetic Energy.
3.2 Quantitative Comparison

As a quantitative comparison, an analytical model of the primary vortex is considered in the context of the two CFD models (mesh-resolved Vg and AcVG Models) and the wind tunnel experiment. This Analytical Model described in Velte [13], reduces the complex flow to four parameters (circulation, convection velocity, vortex core radius and helical pitch) and enables quantitative comparison in addition to the qualitative one.

Figure 15 shows the axial $u_z$ (upper) and azimuthal $u_\theta$ (lower) velocity profiles for a 20 degrees of the device angle and extracted along a line parallel to the wall through the center of the primary vortex.

In order to analyse the quantitative differences between the results of the four models, the Root Mean Square Error RMSE has been calculated

$$RMSE = \sqrt{\frac{\sum (a_i - a'_i)^2}{n}}$$

Figure 16 shows the differences between the models, having as a reference the Actuator VG Model. Green colour bars illustrate the axial velocity differences and the yellow ones the azimuthal differences. M1, M2 M3 and M4 represent the four models: mesh-resolved VG, AcVG, experimental data and the Analytical Model respectively. So, M2-Mi symbolizes the difference between the AcVG Model and the “i” Model.

Figure 16: Differences of the four models in the axial (green) and the azimuthal (yellow) velocities.

More plots have been calculated for a quantitative comparison based in the Analytical VG Model’s parameters defined in Velte [13]: vortex core radius circulation, helical pitch and advection velocity.

Figure 17: Comparison plots of the parameters: (a) Vortex radius, (b) Circulation, (c) helical pitch an (d) advection velocity. M1, M2 M3 and M4 represent the models: mesh-resolved VG, AcVG, experimental data and the Analytical Model respectively.
4. Results

Results of the calibrated Actuator VG Model have been compared with the mesh-resolved VG Model, the experimental data and the Analytical Model. Significant differences have been observed between the four models. The deviations between the values of the Actuator VG Model and the values predicted by the analytical model are quite important, also with the values of the experimental data.

In regard to the computational time, the mesh-resolved VG Model time has been estimated in 1080 hours using 36 CPUs; however, the Actuator VG Model only 3.2 hours, using 16 CPUs. So from the point of view of the computational effort, the efficiency of the AcVG Model is much higher than the mesh-resolved VG Model. A significant reduction in cells is achieved by replacing the detailed VG boundary layer mesh by the new modelling method. This mesh reduction decreases both the VG geometry meshing time and the computational time. These results show that AcVG and mesh-resolved VG Models are qualitatively similar. Once vortex produced by the mesh-resolved VG is fully developed at around 10 VG heights downstream the trailing edge of the VG, the AcVG Model matches the vortex generated by the mesh-resolved VG Model. Figure 13 shows a good agreement in the sense of the vortex development. However, some discrepancies are visible in the quantitative comparison, above all in the axial velocity. It might be because the vortex development at the calibration plane is not completed. As the Figure 16 shows, the most important divergences of the models in comparison with the Actuator VG Model are located in the axial velocity.

Also, some discrepancies are visible on the amount of turbulence kinetic energy generated between the fully meshed VG model and the AcVG Model, which produces less turbulence, as it can be expected from this type of source term model.

5. Discussion

The discussion focuses on the physical property of the new AcVG model, compared with the fully meshed model, the wind tunnel experiment and the analytical model. What are the compromises done in term of physics, with this new model?

6. Conclusions

In conclusion, a new model has been implemented in the EllipSys CFD code and demonstrated that it saves both meshing and computational time. This method could easily be applied for complementing full rotor computation and for doing parametric study of the VG layout. The potentially open applications of the Actuator VG Model are several.

Also, we can confirm that the analytical model developed by [14] can be used as a calibration tool for the AcVG Model.

• Acknowledgements

The author would sincerely like to thank the researchers of DTU Wind at the Technical University of Denmark contributed to this work and to the Centre for Industrial Diagnosis and Fluid Dynamics CDIF at the Technical University of Catalonia.
References


