Contents lists available at ScienceDirect
Aerospace Science and Technology
www.elsevier.com/locate/aescte

Computational characterization of the vortex generated by a vortex generator on a flat plate for different vane angles

A. Urkiola [a], U. Fernandez-Gamiz [a,*], I. Errasti [a], E. Zulueta [b]

[a] Nuclear Engineering and Fluid Mechanics Department, University of the Basque Country, Nieves Cano 12, 01006 Vitoria-Gasteiz, Araba, Spain
[b] Automatic and Simulation Department, University of the Basque Country, Nieves Cano 12, 01006 Vitoria-Gasteiz, Araba, Spain

ARTICLE INFO

Article history:
Received 18 March 2016
Received in revised form 9 January 2017
Accepted 12 February 2017
Available online xxxx

Keywords:
Vortex generators
Half-life radius
Flow control
Boundary layer
Computational fluid dynamics

ABSTRACT

Vortex generators (VGs) are usually employed to improve the aerodynamic performance for both spatial or energy issues; such as aircrafts and wind turbine blades. These structures present poor aerodynamic performance in the sections close to the hub enabling the lift to decay under critical conditions. One way to overcome this drawback is the use of VGs, avoiding or delaying the boundary layer separation. The main goal of this work is to characterize the size of the primary vortex generated by a single VG on a flat plate by Computational Fluid Dynamics simulations using OpenFOAM code. This is performed by assessing the half-life radius of the vortex and comparing it with experimental results. In addition, a prediction model based on two elementary parameters has been developed to describe in a simple way the evolution of the size of the primary vortex downstream of the vane for four different incident angles.

1. Introduction

Flow separation control has become a very important task due to its importance in many industrial applications related to Fluid Mechanics in the last decades of the past century. The most relevant reason of flow separation is the lack of momentum in the boundary layer which makes lift decrease and consequently turns the system into an unstable one. For instance, Taylor [21–23] and Wentz [28] investigated vortex generators (VGs) applied in aerodynamics in order to avoid this non-attached condition. An optimal design of these flow control passive devices, known as VG, could transfer high momentum from the outer side to the inner part of the layer, as sketched in Fig. 1, remaining the flow attached to the wall and ensuring stable conditions, according to Rao and Kariya [16] and Gibertini et al. [10].

VGs are widely used both on airplane wings and wind turbine blades because they enable the use of slender blades allowing less weight for the same load distribution. These flow control devices can be mounted on blades that do not perform as expected. VGs are commonly dimensioned in relation to the local boundary layer thickness δ in order to obtain an optimal interaction between the generated vortex and the local boundary layer (BL). Depending on the flow control application, the height h of these devices could be smaller than the boundary layer as demonstrated by Ashill, Fulker and Hackett [3] and Lin [13]. VGs are frequently triangular or rectangular vanes positioned at an incident angle with respect to the oncoming flow and placed in groups of two or more upstream the flow Anderson [1].

Bearing in mind how important VGs are to prevent or delay the flow separation, basic research has previously occupied several researchers: Fernandez-Gamiz et al. [7] studied the influence of some parameters of these particular devices such as the incident angle dependency, Ünal and Gören [24] the effect on a flow around a circular cylinder, Velte [27] and Fernandez-Gamiz et al. [9] self-similarity and helical symmetry, Fernandez-Gamiz, Zamorano and Zulueta [8] vortex path variation with the height and so on. All this research is based on different theoretical models as the one proposed by Smith [19] or Velte, Hansen and Okulov [26] whose main objective was to demonstrate the helical symmetry of the vortices generated by a passive rectangular vane-type vortex generator. Reader should notice that most of the models are based on the experimental BAY-model proposed by Bender [5] where body forces were used as source terms in the Navier–Stokes equations to simulate the presence of a vane.

The main purpose of this study is to characterize the size of the generated vortex by employing computational simulation. Computational Fluid Dynamics (CFD) simulation has been performed considering four different incident angles and validated not only analytically but also experimentally. The CFD simulations presented in this work have been carried out using the OpenFOAM [15] open source code. This non-commercial code can be optimized and customized to satisfy any kind of physical phenomenon according to

* Corresponding author.
E-mail address: unai.fernandez@ehu.es (U. Fernandez-Gamiz).

http://dx.doi.org/10.1016/j.ast.2017.02.008
1270-9638/

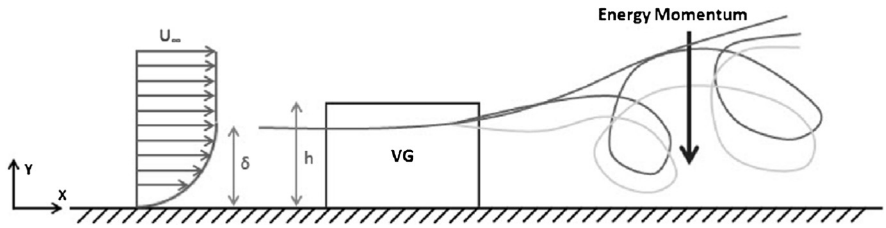

Fig. 1. Boundary layer alteration by a rectangular vortex generator on a flat plate.

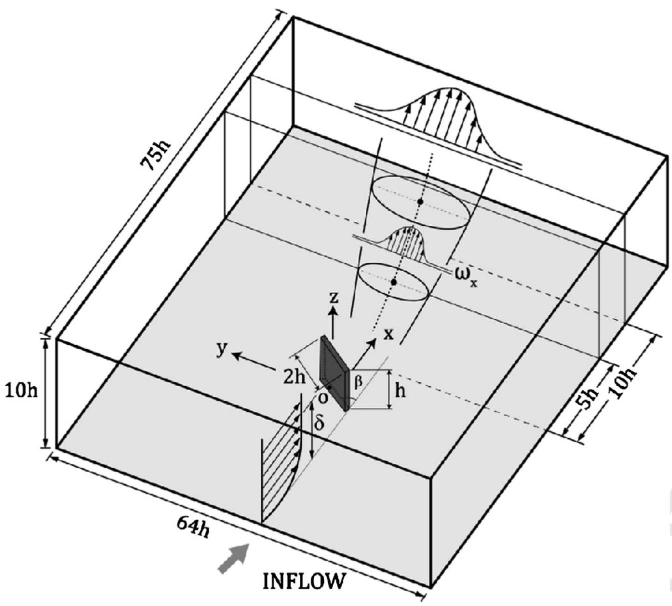

Fig. 2. Sketch of the computational domain where a rectangular vortex generator of height h is located on a flat plate at a specific incident angle of attack β with respect to the oncoming flow.

user needs. The present study compares the numerical results obtained by means of CFD simulations with the experimental ones achieved in wind tunnels by Bray [6]. Finally, the CFD simulations have also been validated with the analytical model of Velte [27]. In this case, the velocity profiles have been evaluated and compared. As mentioned, four incident angles of attack have been studied for the same Reynolds Number of 27000: $\beta = 10°$, 15°, 18° and 20°.

2. Numerical setup

2.1. Computational configuration

In order to obtain some of the main features of the vortex generated, CFD techniques have been employed. Nowadays, non-commercial and proprietary CFD codes are used to reproduce relatively well any physical problem. In this work, the open source code OpenFOAM [15] has been used for simulating the vortex. This open source CFD code is an object-oriented library written in C++ to solve computational continuum mechanics problems. One of its advantages is that the user can modify the code to create new solvers and applications as well as freely share the code developed. The current computational domain consists of a single rectangular vortex generator VG placed on a flat plate with a negligible pressure gradient at a specific incident angle β with respect to the oncoming flow as shown in Fig. 2.

The geometry dimensions of the rectangular vortex generator are defined with a length of two times its height $h = 0.25$ m which corresponds to the local BL thickness δ. The dimensions for the computational domain ($64h \times 10h \times 75h$) have been scaled to the VG height h. The computational domain's width and height are 64 times and 10 times the vortex generator height h respectively. The third dimension corresponds to the length which is 75 times the vortex generator height to capture the vortex generated.

There are several features that should be taken into account when focusing on the Fig. 2. Firstly, the computational domain is dimensioned according to previous published studies by Fernandez-Gamiz et al. [9] and Fernandez-Gamiz, Zamorano and Zulueta [8] where a similar computational setup was used Secondly, some details of the wake downstream of the vortex generator can be distinguished such as the axial vorticity ω_x of the generated vortex. Fig. 2 shows how the vortex increases its size as the distance to the trailing edge of the vortex generator gets bigger. In addition, the values of the axial vorticity ω_x of the vortex are ruled by a Gaussian distribution according to Lamb [12] and Squire [20]: the maximum value of the axial vorticity is obtained at the center of the vortex and its minimum value ($\omega_x = 0$) in the regions where there is no vortex influence. Finally, the location of the particular device has been determined according to Schlichting [18] who stated that the development of the boundary layer thickness δ is related to the axial Reynolds Number R_{e_x}:

$$\delta = \frac{0.37 \cdot x}{\sqrt[5]{R_{e_x}}} \quad [\mathrm{m}] \tag{1}$$

$$R_{e_x} = \frac{U_\infty \cdot x}{\nu} \quad [-] \tag{2}$$

where ν refers to the kinematic viscosity, x the axial position and finally, U_∞ the free stream velocity. Thus, the vortex generator was placed on a test section wall in such way that the local boundary layer thickness at that location was close to the VG height. The simulations have been carried out considering an oncoming flow speed of 20 m s^{-1} and a Reynolds Number $Re = 27000$ based on the VG height as defined in the expression:

$$R_{e_h} = \frac{\rho \cdot U_\infty \cdot h}{\mu} \quad [-] \tag{3}$$

where ρ is the density, μ the viscosity, h the vortex generator height and U_∞ the free stream velocity. The computational domain has been rotated to get the different incident angles of attack of 10°, 15°, 18° and 20°. The simpleFoam solver has been applied for steady-state, incompressible and turbulent flows using the RANS (Reynolds Average Navier–Stokes) equations. All along the calculations, this solver uses the k-omega SST (Shear Stress Transport) turbulence model according to Menter [14]. Allan, Yao and Lin [2] observed that this turbulence model resulted in a better prediction of the streamwise peak vorticity and trajectory. The domain analyzed in this study is discretized with a structured type mesh made of flat hexahedral faces of around 11.5 million cells. Part of the refined mesh can be seen in Fig. 3(a). Full second order linear-upwind scheme for the discretization has been used for all computations.

An optimized mesh plays a major role in the CFD simulations as the tool that will help the user to discretize the domain. It is important to identify the mesh regions where the results have to be

Please cite this article in press as: A. Urkiola et al., Computational characterization of the vortex generated by a vortex generator on a flat plate for different vane angles, Aerosp. Sci. Technol. (2017), http://dx.doi.org/10.1016/j.ast.2017.02.008