Numerical study of minute vortex generator jets in a turbulent boundary layer

M J PourRazzaghi¹ and C Xu².

¹ Ph.D. student, Mechanical school of Mechanical Engineering, Nanjing University of science and technology, Nanjing 210094, China;
² Full professor, Mechanical school of Mechanical Engineering, Nanjing University of science and technology, Nanjing 210094, China.

Email: mj.pourrazzaghi@njjust.edu.cn.

Abstract. The air-jet vortex generator has been proposed as a credible alternative to the more conventional vane vortex generator. Some numerical models of the flow in the neighborhood of these devices have been constricted to evaluate their potential benefits. The objective of this paper is to consider the effects of a single minute AJVG in a turbulent boundary layer on a flat plane and study the longitudinal vortex caused by merging two-minute co-rotating vortex in turbulent boundary layer. The flows were assumed fully turbulent and were solved using the finite volume, OpenFoam 5.0 on a structures grid, using SST k-ω turbulence model and standard wall functions. Predicted results were also validated with experimental data. A reasonably good match was served between the predicted and experimental Reynolds stress data. However, in compare of experimental data the longitudinal vortices were not remain along the stream and were disappeared early.

1. Introduction

Jets issuing through small holes in a wall boundary of a flow have proven effective in control of turbulent boundary layer separation resulting from adverse pressure gradient [1-4]. The beneficial effects are obtained only of the jets are pitched and shewed with respect to the main flow direction. This interaction with the crossflow, creates a single longitudinal vortex which persists well downstream and enhances cross-stream mixing. This technique is known as the vortex generator jet (VGJ) method of separation control which is resembles a well-known method of using a small vane vortex generator [5]. VGJs have some obvious advantages over vane vortex generators, for example, they can be used in active control and do not suffer from drag penalties. However, the flow field over the VGJs is very complicated. Modeling the flow with VGJs and designing VGJs application systems are more difficult than that for vane vortex generators. Many problems about VGJs are not understood completely. One of most possible and common engineering applications of VGJs is embedding and one or away of co-rotating VGJs in the adverse pressure gradient boundary layer and using them to delay or eliminate flow separation. The effect of an array co-rotating VGJs, specially, the minimized co-rotating VGJs system on adverse pressure gradient turbulent boundary layer is a basic problem required to study that associated with this kind of application.

Wallis [1,2] appears to have been the first to employ air-jet vortex generators (AJVGs) to delay separation of a turbulent boundary layer. A few years later, Pearcey [5] investigated the ability of
circular AJVGs to attenuate shock-induced flow separation on aerofoils and wings. AJVGs are generated by forcing air through small orifices. These orifices are drilled through the surface at a pitch and skew angle to the on-coming flow. When the jet issues into the main flow it is bent over and a single vortex is formed. The longitudinal vortex aligns with the on-coming flow and promotes redistribution of momentum within the boundary layer. More recently, several research groups have studied in more detail the impact of AJVGs on a turbulent boundary layer over a flat plate. Johnston and Nishi [3] and Compton and Johnston [6] conducted experiments whereas Zhang [7], Henry and Pearcey [8], Akanni and Henry [9] and Kupper and Henry [10] performed computational studies. All concluded that these devices have distinct advantages over the more conventional vane vortex generators. Kupper and Henry [10] conducted a detailed comparison of the flow fields in the vicinity of both an AJVG and a vane vortex generator array.

The purpose of this paper is to present an initial account of a numerical study of a minute VGJ and the subsequent interaction between vortex and boundary layer. Specifically, the problem considered is that of an incompressible turbulent boundary layer over a flat plate into which are issuing a pitched and skewed air jet. The aims of the study were: to show that it was possible to model the three-dimensional flow numerically with reasonable accuracy using comparisons with published experimental results where possible, to investigate the influence of some of the various parameters of the problem, and to quantify the manner in which the vortex sustain positive skin friction in an adverse pressure gradient that would otherwise cause the boundary layer to separate. In the following, a brief account is given of the merging the two-minute vortices and investigate the efficiency of new vortex were discussed. The predictions are compared to the experimental data of Cheng Xu. [11].

2. Modeling

The numerical model reflects the geometry and flow conditions of the experimental set-up of Cheng Xu. [11]. the experiment was conducted in a low speed, closed-circuit boundary layer wind tunnel. The test section, is 60cm wide between parallel side wall and 13.0 cm high at the constant height section near nozzle exit. The inlet air speed was set at 15.0m/s in these experiments. The tunnel coordinate system, such that x, y, z is the free-stream, wall-normal and spanwise directions respectively, is used.

As it can be seen in the figure 1 the origin of the coordinate system (x=0) is defined as the location 83.0cm downstream from the inlet, same as from leading edge of the test wall in experimental model, which is the same station the VGJs mounted.

![Figure 1. Computational domain based on Wind tunnel geometry of experimental data.](image)

The plate and jet geometry, and solution domain of the numerical models are shown in figure 2. In this case, the jet inlet is situated at the center of the lower surface of the solution domain. Modelling schematic of one AVGJ and also two AVGJs which are pitched and skewed relative to the crossflow can be seen in figure 2. The local boundary layer height was calculated using the correlation [12].

$$\delta = 0.37x\left(\frac{U_jx}{v}\right)^{-0.2}$$ (1)
The finite volume code by OpenFoam 5.0 was used to solve the equations for the incompressible, turbulent and steady flow on a block-structured. The SST k-ω turbulence model with standard wall functions were used.

![Figure 2](image)

**Figure 2.** Geometric details of numerical model of jet hole and outflow direction.

Where the kinematic viscosity: \( \nu = \frac{\mu}{\rho} \), and \( \rho = 1.12 \, \text{kg/m}^3 \) and \( \mu = 1.81 \times 10^{-5} \, \text{kg/s m} \).

The SST k-\( \omega \) turbulence model [13] it is assumed.

The transport equations for the turbulent kinetic energy \( k \) is given by:

\[
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ (\nu + \sigma_k v_f) \frac{\partial k}{\partial x_j} \right]
\]  

(2)

The governing equation for the Specific Dissipation Rate, \( \omega \), is given by:

\[
\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ (\nu + \sigma_\omega v_f) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \sigma_\omega \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
\]  

(3)

### 2.1. Boundary conditions

#### 2.1.1. Inlet boundary.

The streamwise velocity at the solution domain inlet \( U_s \), was assumed a uniform velocity, \( U_s = 15 \, \text{m/s} \). The cross stream and vertical velocity components at the domain inlet were set to zero. The inlet values of \( k \) and \( \omega \) are usually unknown, and one needs to take guidance from experimental data for similar flows. The simplest practice is to assume uniform values of \( k \) and \( \omega \) computed from [14]:

\[
k = (I \ast U)^2
\]  

(4)

\[
\omega = \varepsilon / C_D \ast k
\]  

(5)

\[
\varepsilon = C_D \frac{3}{2} \ast k \sqrt{\frac{l_m}{l}}
\]  

(6)

Where \( U \) is the bulk inlet velocity, \( I \) is the turbulent intensity (typically in the range \( 0.01 < I < 0.05 \)) and the mixing length \( l_m \sim 0.1 \), where \( H \) is a characteristic inlet dimension, \( C_D = 0.09 \), say the hydraulic radius of the inlet channel.

#### 2.1.2. Jet inlets.

In the model the plug uniform velocity profile was defined on the surface of the AJVG orifice (i.e. on the surface of the plate). The magnitude of the jet inlet velocity, \( U_j \), was defined
to be a multiple of the free stream velocity \( U_0 \). In all predictions discussed in this paper \( U_0 \) varies from \( U_0 \) to \( 4U_0 \).

The upper and outlet boundaries were set to constant pressure. The side boundaries were set to be symmetric. However, the width of the solution domain was very large compared to the vortex width and thus the influence of the side boundaries was negligible.

2.2. Grid and numerical solution
A non-orthogonal, block-structured grid with 32 blocks was generated via Ansys-ICEM. A plan and a side view of the mesh can be seen in figure 3a. A detailed view of the inner block configuration the jet inlet is shown in figure 3b, also generated grid around the jet hole can be seen in figure 3c.

The Reynolds stress distribution over a X-plane \((x=2\text{m})\) cross plane was chosen as an indicator of grid independence. Reynolds stress distribution was found to be insensitive to further grid refinement for a grid of about 2,000,000 cells. As A variety of advections schemes were employed for the initial calculations, but as will be shown in the next section, the second-order Higher Upwind (HUW) advection scheme was found to give the best overall performance. Typically, convergence was achieved in 1500 iterations, with a mass source error reduction of the order of \( 10^{-7} \).

![Figure 3](image)

Figure 3. (a) Side view numerical grid and (b)&(c) detailed view of the inner block configuration and generated grid around the jet hole.

The Reynolds stresses, \( \overline{u'\sqrt{}} \) and \( \overline{u'\sqrt{}} \) are both evaluated with different number of meshes. The comparison between the simulation result and experimental data in figure 4 shows that the model represents reliable predictions. Base on this result, sensitivity of predicted data become ignorable when the number of grid exceed more 2 million cells, so rest of study has been done by this grid number.
Figure 4. Sensitivity of mesh number on results and validation with experimental data.

3. Result

3.1. A single air jet vortex generator with different velocities
In this part of this study, the effects of a single air jet vortex generator on turbulent boundary layer parameters will be discussed. Different air jet velocities are shown by velocity ratio \( V_r \) which is:

\[
\frac{V_{jet}}{U_0} = V_r
\]

(7)

Figure 5. Stream wise velocity contour different sections along the solution domain while vortex formed in the boundary layer.

As can be seen in figure 5 the streamwise velocity contours along the solution domain which represents the vortex formation in different position alongside the stream. Since the vortex due to air jet injection formed in the boundary layer, the effect of circular motion is remained.

Here the effect of minute single jet with different velocity ratios \( V_r \) on pressure coefficient \( (C_p) \) and pressure gradient \( (\partial p/\partial x) \), are discussed. In figure 6 \( (C_p) \) and \( (\partial p/\partial x) \) are shown on center line of a solution domain. It is clear in different velocity ratios, pressure coefficient along the stream changed,
specifically after jet inlet while the vortex forms and continues in stream direction. Pressure is decreased and that it has effect on velocity inside the underlayer. Streamwise direction pressure gradient of turbulent boundary layer is affected by a single air jet. figure 6 shows that by higher velocity ratio of air jet longer distance of pressure gradient can be affected by the vortex that causes a delay in separation flow.

![Figure 6. Centerline Pressure coefficient and Pressure gradient with different velocity ratios.](image)

Predicted streamwise velocity profiles are given in figure 7 in the plane ($x_1 = 100 \, mm$) and plane ($x_2 = 1000 \, mm$) from VGJ inlet at lateral position of vortex core. This prediction shows that streamwise velocity profile doesn’t change significantly at $V_r = 1 \ and \ 2$, but $V_r = 4$, the profile obviously changes and the streamwise velocity of inside the boundary layer increases, which it causes the boundary layer energy. As it can be seen in the figure 8 this effect is more obvious at plane $x_2$.

![Figure 7. Streamwise Profile at two position from VGJ inlet, ($x_1 = 100 \, mm$) and ($x_2 = 1000 \, mm$).](image)

Cross-stream velocity components, $v, w$ profiles at the same locations are given in figure 8. All the predictions are on the cross-stream line which passes the vortex core. As it can be seen in the graphs there are maximum and minimum values for streamwise, $u$ and two cross-stream velocity components $v, w$. This max and min values represent a single vortex. In both plane positions ($x_1, x_2$), relatively as the velocity ratio of VGJ increases the max and min values increases and it shows the vortex strength increases too.
Figure 8. Streamwise and Cross-stream velocity profiles at cortex core in positions of $(x_1 = 100 \text{ mm})$ and $(x_2 = 1000 \text{ mm})$ from jet inlet.

Reynolds stress profiles in plane positions $(x_1 = 50 \text{ mm})$ and $(x_2 = 1000 \text{ mm})$ at the vortex core are illustrated in figure 9. Predicted results of Reynolds stress values are normalized with square mean velocity at edge of boundary layer. As the velocity ratio increases the Reynolds stress components $u'v'$ maximum and minimum, increase and decreased, respectively.
Figure 9. Reynolds Stress profiles $\overline{uv}$ at cortex core in positions of ($x_1 = 50 mm$) and ($x_2 = 1000 mm$).

3.2. Two single air jet vortex generators at $V_r = 4.0$
In this section, effect of two co-rotating minute vortex generator is studied. In pervious section a single minute VGJ with velocity ratio of $V_r = 4.0$ was obviously effective, so here all the results are presented with $V_r = 4$.

In figure 10a the streamwise velocity contours of several sessions along the stream are shown. Streamlines and the merging effect of two-minute co-rotating VGJ is presented in figure 10b. As it shows at first two single minute vortices formed but as long as they continue, two-minute vortices merge and formed one bigger vortex with higher strength.

Figure 10. (a) streamwise velocity contours with two minute VGJs figure 12 (b) streamline of two VGJ.
The comparison of streamwise velocity component $u$, profiles in two planes at ($x_1 = 50 \text{ mm}$ ) and ($x_2 = 1000 \text{ mm}$), for a single jet and two co-rotating minute jets are shown in figure 11. At the first position, before two jet merging, effect of a single vortex in boundary layer is more obvious, but later when two-minute vortex merge together at $x_2 = 1000 \text{ mm}$ position, higher streamwise velocity caused by merged vortices is significant. it is clear that boundary layer is energized by a vortex with higher strength.

![Figure 11. Streamwise velocity profiles comparison between a single VGJ and two co-rotating VGJs.](image)

4. Conclusion

Two models of a single and a pair of AJVG were studied numerically. A single longitudinal vortex was predicted and streamwise and cross-stream velocity profiles in different jet velocities were compared to each other. The numerical results were valid with experimental data. Higher jet velocity ratio enhance mixing in boundary layer. Although Numerical prediction shows cross-stream velocities were increased with the same amount which kept the circular shape of vortex and the vortex was not remain along solution domain and was disappeared at middle of solution domain. Despite this fact the general trend of predicted results agrees experimental data. Later merging two minute co-rotating AJVG was studied. Based on prediction results, two minute vortices merge together and formed bigger longitudinal vortex which enhanced the boundary layer energy. This effects on adverse pressure gradient and due to this, delay septation flow near wall boundary.

References

[1] Wallis R A 1960 A Preliminary Note on a Modified Type of Air Jet for Boundary Layer Control Aeronautical Research Council 513 London
[2] Ball W H 1984 Tests of Wall Blowing Concepts for Diffuser Boundary Layer Control AIAA Journals June 84-1276
[3] Johnston J P and Nishi M 1990 Vortex Generator Jets-A Means for Flow Separation Control AIAA Journal 28(6) 989-94
[4] Lin J C, Howard F G, Bushnell D M and Selby G V 1990 Investigation of Several Passive and Active Methods for Turbulent Flow Separation Control AIAA Journals Paper 90-1598
[5] Pearcey H H 1961 Shock Induced Separation and Its Prevention Boundary Layer and Flow Control 2 1170-344.
[6] Compton D A and Johnston J P 1992 AIAA Journal 30 640–7
[7] X Zhang July 1993 Computational analysis of co- and contra-rotating streamwise vortices in a turbulent boundary layer AIAA Journals Paper 93-3035
[8] Henry F S and Pearcey H H 1994 AIAA Journal 32 (12) 2415-25
[9] Akanni S D and Henry F S 1995 Numerical calculations for air-jet vortex generators in turbulent boundary layers CEAS European Forum on High Lift and Separation Control

[10] Kupper C and Henry F S 2001 Aeronautical Journal 105 255–65

[11] Guan X R and Xu C 2009 Numerical investigation of boundary-layer control using minute jet vortex generator (Nanjing university of science and technology)

[12] Menter F R 1994 AIAA Journal 32(8) 1598-605

[13] Wilcox D C 2008 AIAA Journal 46(11) 2823