**QUEEN'S
UNIVERSITY
BELFAST**

Blown Jet Vortex Generator Control of a Separated Diffuser Flow

Quinn, D., & Watterson, J. (2006). *Blown Jet Vortex Generator Control of a Separated Diffuser Flow*. Paper presented at 44th AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada, United States.

Document Version:

Early version, also known as pre-print

Queen's University Belfast - Research Portal:

[Link to publication record in Queen's University Belfast Research Portal](#)

General rights

Copyright for the publications made accessible via the Queen's University Belfast Research Portal is retained by the author(s) and / or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy

The Research Portal is Queen's institutional repository that provides access to Queen's research output. Every effort has been made to ensure that content in the Research Portal does not infringe any person's rights, or applicable UK laws. If you discover content in the Research Portal that you believe breaches copyright or violates any law, please contact openaccess@qub.ac.uk.

Blown Jet Vortex Generator Control of a Separated Diffuser Flow

D. Quinn^{*}, J. K. Watterson[†] and A. B. Crawford[‡]

School of Mechanical and Aerospace Engineering, The Queen's University of Belfast, Belfast, BT9 5AH, United Kingdom

The application of blown jet vortex generators to control flow separation in a diffuser with an opening angle of 10° has been studied using the computational fluid dynamics (CFD) code Fluent 6TM. Experimental data is available for the uncontrolled flow in the diffuser. The section of the duct upstream of the diffuser has a height H equal to 15 mm; its length and breadth are $101H$ and $41H$ respectively; the diffuser has an expansion ratio of 4.7:1. Fully developed flow is achieved upstream of the diffuser. Pipes of diameters equal to 1.5%, 2.5% and 5% of H were considered; pitch angle was constant at 45° and yaw angle was fixed at 60° ; velocity ratio was varied from 1.7 to 8.0; both co-rotating and counter-rotating arrays were studied. The best results were obtained with a counter-rotating array of generators with a hole diameter of 5% of H and a velocity ratio of 3.7.

Nomenclature

BJVG	=	Blown jet vortex generator
d	=	BJVG pipe diameter
H	=	depth of duct upstream of diffuser
U	=	freestream velocity
u	=	axial component of velocity
δ	=	local boundary layer thickness

I. Introduction

BOUNDARY layer separation occurs as the result of adverse pressure gradient. Provided the increase in static pressure is gradual, it is possible for the mixing processes in the boundary layer to transfer momentum from the free stream down to the wall region at a rate that balances the rate at which it is absorbed by the rising pressure. This process is orders of magnitude more rapid in turbulent boundary layers than laminar, such that the former are substantially more robust in the face of a diffusing flow, albeit at the cost of more rapid boundary layer thickening. If the mixing process is unable to sustain the loss of momentum in the wall region, the boundary layer will separate from the wall: on an aerofoil, this results in loss of flow curvature and reduction in lift and increase in drag; in a diffuser, there is strong mixing in the separated flow and pressure recovery falls below the ideal value. Boundary layer flow control is an attempt to interfere beneficially with the boundary layer momentum in the wall region. It may take the form of removal of "tired" boundary layer fluid, by suction, energizing of the boundary layer fluid by blowing, or enhancement of the rate at which free stream momentum is mixed with the boundary layer fluid. Each of these forms of flow control has been the subject of intensive and extensive research. In particular, enhancement of boundary layer mixing has been achieved by means of embedded longitudinal vortices, generated by a variety of mechanisms, such as wedges, bumps, vanes and blown jets¹. Vane-type vortex generators have been employed extensively on aircraft and to maintain attached flow for high lift applications on wings, maintenance of cross-wind control authority on vertical stabilizers and drag reduction on fuselage tail cones. Of these, the former two are essentially low speed applications; however, since the vanes remain deployed throughout flight, they entail a small drag penalty in cruise, though this is evidently offset by the improvement in low speed handling qualities that they bring and their mechanical simplicity. Blown jet vortex generation, on the other hand, requires a more complicated

^{*}Research student, School of Mechanical and Aerospace Engineering, Queen's University of Belfast, BT9 5AH, UK

[†]Senior lecturer, School of Mechanical and Aerospace Engineering, Queen's University of Belfast, BT9 5AH, UK

[‡]Research Assistant, Centre of Excellence for Integrated Aircraft Technology, Queen's University of Belfast

mechanical design (at the very least, pipe work connected at one end to a source of air and at the other to carefully manufactured holes or slots in the wing surface) and a source of air. This is not an insurmountable obstacle: a similar mechanical design for hybrid laminar flow control is being flight tested². The chief advantage of blown jet vortex generation is that it can be switched off during cruise: the only cruise drag penalty is the lift dependent drag associated with the additional weight of the system.

The idea of the generation of embedded streamwise vortices dates back to Wallis³. The concept has been extensively researched over the last couple of decades (e.g. References 4-9) and it was quickly established that the performance of blown jet vortex generators (BJVGs) depends on pitch and yaw angles, velocity ratio, hole spacing and array type. (The shape of the hole from which the jet issues may typically be either a slot or elliptical. The performance of the resulting BJVG does not appear to be strongly influenced by the form adopted¹⁰, though elliptical holes may be preferable⁷.) Pitch angle is defined as the angle formed between the axis of the jet pipe and the local plane of the boundary through which the jet issues; typically, pitch angles of between 20 deg and 45 deg have been studied. Yaw angle is the angle between the reference freestream direction and the axis of the jet pipe resolved into the local plane of the boundary; typically, the yaw angle has been varied between 0 deg and 90 deg, though Compton and Johnston⁶ considered values as high as 180 deg. It is generally agreed that the best pitch angle is about 45 deg, and performance (meaning strength of the vortex generated and its ability to prevent boundary layer separation) is best when the yaw angle is 60 deg.

Velocity ratio is the ratio of the mean velocity in the jet pipe to the reference freestream velocity: most studies appear to use either a low range of velocity ratio of something like 0.5 to 1.0^{4,6,8} or values that range well above unity^{5,7}. Milanovic and Zaman⁹ employed the idea of momentum-flux ratio, defined as mean momentum flux in the jet pipe divided by the freestream momentum flux, which is more general, allowing for a difference between the density of the fluid in the jet and the freestream flow: a circumstance that might arise in an aeronautical context or for film cooling flows. They used values in the range 1.5 to 20, which, for equal jet and freestream densities, corresponds approximately to velocity ratios between 1.2 and 4.5. Low velocity ratios are capable of reducing the extent of boundary layer separation in diffusing flows, but higher values of order unity are preferable, provided the vortices generated do not puncture through the boundary layer they are aimed at controlling.

As with all forms of vortex generators, BJVGs may be arranged in either co-rotating or counter-rotating arrays. Selby et al⁷ report that their experimental investigation demonstrated that co-rotating arrays perform better than counter-rotating arrays in a diffuser. This contradicts the experience of the present authors^{11,12} in whose experience counter-rotating arrays are to be preferred.

Using PIV, Rixon and Johari¹³ looked at the effect of velocity ratio on the development of vortices in a turbulent boundary layer. They held the pitch and yaw angles constant at 45 deg and 90 deg respectively and used a hole diameter to boundary layer thickness ratio of 14% (boundary layer thickness was measured upstream of the jet). Their range of velocity ratio was limited to 1, 2 and 3 but they did find that increasing the velocity ratio increased the circulation and peak vorticity of the vortex. They found that in the case of VR = 2 and 3 the vorticity decayed exponentially downstream of the jet while the decay rate was significantly slower in the case of VR = 1. This seems to imply that a lower velocity ratio will produce a vortex that penetrates deeper into the boundary layer but is weaker. This suggests the question: which has more influence on separation control, vortex strength or depth of penetration?

Compton and Johnston⁶ also report a numerical study of vortex production in a boundary layer using pitched and yawed jets and concluded that the jet generated vortices were very similar to weak vortices generated by solid generators. The one significant difference was that the vorticity in jet generated vortices decayed in a different manner: rapid decay at first but slower downstream. This indicated that while the vortices are weaker they exist further downstream than solid generated vortices. Also of note was the discovery that the maximum vorticity was dependent on jet velocity and yaw angle, but not pitch angle. Another early numerical study of vortex generator jets was carried out by Henry and Pearcey¹⁴ using the CFD tool FLOW 3D and the $k-\epsilon$ turbulence model. They modelled flow on a flat plate with an imposed adverse pressure gradient, and meshed one jet, with appropriate side wall boundary conditions to model counter-rotating and co-rotating configurations. For the jet outlets they used high aspect ratio rectangular slots instead of circular holes and argued that shape did not matter providing the same mass flow was emerging. The counter rotating results were generally better and more efficient, the co-rotating configuration required higher jet velocities to achieve any degree of success. They also noted that the co-rotating configuration was very sensitive to the jet spacing and recommended spacing between 3 and 6 times the height of the vortex core.

The work described in this paper is a numerical investigation of the effect of application of BJVGs to a diffuser flow. The work uses data published by Buice and Eaton¹⁵ for the flow in a simple diffuser of high aspect ratio: the upstream flow is highly two-dimensional, and the flow separates early in the diffusing section, thus providing a test

case for the performance of the numerical method for the clean, uncontrolled flow. Previous numerical studies of two-dimensional models of the Buice and Eaton geometry have highlighted the inadequacy of standard $k-\epsilon$ turbulence modelling and the superior performance of $k-\omega$, Spalart-Allmaras and $\bar{v}^2 - f$ models^{12,16,17}.

II. Numerical Method

A. Flow Solver and Numerical Conditions

This study employed the commercial CFD code Fluent¹⁸. The incompressible Reynolds-averaged Navier-Stokes equations were modelled. The software was run in its implicit segregated mode; the SIMPLE algorithm was used for pressure-velocity coupling and second order spatial discretisation was used for all the equations. Only steady state solutions were sought and calculations were iterated until all the residuals had 'flat lined'. The standard $k-\epsilon$, enhanced wall function $k-\epsilon$, SST $k-\omega$ and Spalart-Allmaras turbulence models were all tested.

Figure 1. Cross section of the diffuser, from Ref. 15.

B. Geometry, Mesh and Boundary Conditions

The geometry chosen is based on that of Buice and Eaton¹⁵ and is shown in Figure 1. The experimental duct upstream of the diffuser has a constant height H of 15 mm. The width of the duct is $41H$ and the duct extends $101H$ upstream and $56H$ downstream of the diffuser; the diffuser has a slope of 10 degrees and an area ratio of 4.7:1. In the numerical investigation, the upstream duct was limited to just $20H$ and preliminary two-dimensional calculations in a long constant area duct of height H were used to obtain suitable fully developed upstream boundary conditions for the three-dimensional calculations. Downstream of the diffuser, the numerical domain extended $50H$.

The BJVG pipes were modelled as having circular cross-section, with their orifice positioned immediately upstream of the start of the diffuser. Diameters of $0.015H$, $0.025H$ and $0.05H$ were considered. The pipes were modelled as 10 diameters long. The pitch and yaw angles were 45 degrees and 60 degrees respectively. Values of 1.7, 2.5, 3.7 and 8.0 were chosen for the velocity ratio, defined as the ratio of the mean velocity in the BJVG pipe to

the mean velocity in the tunnel section upstream of the diffuser. By selecting symmetry or periodic sidewall boundary conditions, the VGJ arrays could be modelled, respectively, as either counter-rotating or co-rotating. The configurations tested are tabulated in Table 1.

Table 1. Configurations tested

Case	Configuration	Pipe Diameter (% H)	Velocity Ratio
<i>A</i>	Counter-rotating	1.5	1.7
<i>B</i>	Counter-rotating	1.5	2.5
<i>C</i>	Counter-rotating	1.5	3.7
<i>D</i>	Counter-rotating	1.5	8.0
<i>E</i>	Counter-rotating	2.5	1.7
<i>F</i>	Counter-rotating	2.5	2.5
<i>G</i>	Counter-rotating	2.5	3.7
<i>H</i>	Counter-rotating	2.5	8.0
<i>I</i>	Co-rotating	2.5	1.7
<i>J</i>	Co-rotating	2.5	2.5
<i>K</i>	Co-rotating	2.5	3.7
<i>L</i>	Counter-rotating	5.0	1.7
<i>M</i>	Counter-rotating	5.0	2.5
<i>N</i>	Counter-rotating	5.0	3.7
<i>O</i>	Counter-rotating	5.0	8.0

The following boundary conditions were specified. A two-dimensional calculation for a constant area duct was performed with the inlet velocity magnitude set at 20m/s, turbulence intensity 5%, turbulent viscosity ratio = 10. All flow quantities were extrapolated to the outflow boundary, and the outflow mass flow rate was set equal to the inflow mass flow rate. The no-slip condition was applied on all solid surfaces. Fully developed profiles of velocity and turbulence quantities were obtained from this calculation and applied as inflow boundary conditions for the three-dimensional calculations. The side walls were defined as either symmetry boundaries (in the case of the counter-rotating generators) or periodic (in the case of the co-rotating generators).

This work employed structured meshes. Initial two-dimensional calculations indicated the need to resolve the flow down to the wall in order to predict the separation in the diffuser correctly. Meshes using $y^+ \approx 1$ were used in all subsequent calculations. The $k-\epsilon$ (with appropriate wall treatment), SST $k-\omega$ and Spalart-Allmaras turbulence models were all tested and the SST $k-\omega$ model was chosen for the vortex generator calculations. The final mesh for the flow control calculations employed the same wall normal node distribution as was determined to be satisfactory for the initial two-dimensional case: it comprised $256 \times 64 \times 32$ nodes in the x , y and z directions.

C. Mesh Dependence Study

A mesh dependence study was carried out for the two-dimensional clean diffuser. Three meshes (128×32, 256×64 and 512×128) were used. Care was taken to ensure that $y^+ \approx 1$ was maintained on all no-slip boundaries. Velocity profiles obtained on the coarse and medium meshes (using the $k-\omega$ turbulence model) are shown in Fig. 2 (the solution on the fine mesh is not shown for ease of presentation). The solutions are in very good agreement, suggesting that the medium mesh is sufficiently fine for a mesh independent solution. Subsequent three-dimensional meshes 256×64×32 were employed for the flow control cases.

III. Results and Discussion

A. Clean tunnel

Velocity profiles calculated on the two-dimensional 256×64 mesh for the uncontrolled diffuser are compared in Fig. 3 with the data published in Ref. 15. It is clear that the $k-\epsilon$ model with enhanced wall functions was incapable of predicting the separation (the same conclusion was reached for the standard $k-\epsilon$ model, in accord with the findings of Refs. 16 and 17). The Spalart-Allmaras model performed a little better, predicting a separation from the diffusing ramp; however, the separation is too late, too small, and the reattachment is too early. A much better prediction was obtained with the SST $k-\omega$ model. In this case, the separation point is predicted a little too early, but the size of the recirculation region agrees well with the measurements and the reattachment point is about right. In addition, in contrast with the $k-\epsilon$ and Spalart-Allmaras predictions, the form of the velocity profiles downstream of the diffusing ramp is correct: while the former turbulence models result in almost symmetrical profiles, the $k-\omega$ model predicts correctly that the flow is displaced towards the upper wall. All subsequent calculations for the three-dimensional flow control cases employ the SST $k-\omega$ model.

Figure 2. Results of mesh dependence study on coarse ($\diamond$) and medium meshes ($\square$).

B. Effect of flow control

Figs. 4, 5 and 6 show sample velocity profiles for the three-dimensional controlled flow cases, illustrating, respectively, the effects of velocity ratio, pipe diameter and array orientation.

Fig. 4 shows velocity profiles for the counter-rotating cases A and C, in the common-flow-down plane; case A has the lower velocity ratio. In case A the flow appears to be unaffected by the embedded vortices: there is a separation from the diffuser much the same as in the uncontrolled case. Just before the end of the diffuser is reached, the flow reattaches and there is a redistribution of flow towards the lower wall; so strong is this redistribution that the flow separates from the upper wall, resulting in a poor pressure recovery. In contrast, the flow in case C remains attached to both the upper and lower walls over the whole length of the diffuser. Indeed, case C gave the best pressure recovery of all the cases studied.

(a) Standard $k-\epsilon$ with enhanced wall treatment

(b) Spalart-Allmaras

(c) SST $k-\omega$

Figure 3. Clean tunnel predictions with different turbulence models: $\circ$ data from Ref. 15; — predictions.

Figure 4. Effect of velocity ratio: ○ data from Ref. 15; — Case A (VR = 1.7); — Case C (VR = 3.7).

The effect of pipe diameter is illustrated in Fig. 5 for cases C and N. As observed in Fig. 4, case C, with a pipe diameter of $0.025H$, results in flow attachment to both the upper and lower walls of the diffuser; however, the attachment might be regarded as tenuous, and a slightly stronger adverse pressure gradient might result in flow separation. Case N, on the other hand, employs a pipe of twice the diameter ($0.05H$), and therefore delivers four times as much momentum for the same velocity ratio. The velocity profile just downstream of the entry to the diffusing section shows what looks like a wall jet: this is the effect of the strong thinning of the boundary layer in the common-flow-down region between adjacent vortices. The wall jet ensures good attachment of the flow throughout the diffuser. However, this comes at the cost of a substantial flow separation from the upper wall. Clearly the design of the BJVG control must be tailored to the application: diffusers (internal flows) require attachment of the whole flow, while flaps (external flows) only require attachment to a single surface.

Figure 5. Effect of pipe diameter: ○ data from Ref. 15; — Case C (diameter = $0.025H$); — Case N (diameter = $0.05H$).

A comparison of co-rotating and counter-rotating cases is provided in Fig. 6. Case K, the co-rotating arrangement, shows evidence of the deterioration of the flow: the separation on the diffuser appears to be larger than in the uncontrolled case, but reattaches immediately downstream of the diffuser, resulting in a redistribution of the flow towards the lower wall and a separation from the upper wall – two separations for the price of one. The pressure recovery is poorer than in the uncontrolled case. This runs counter to the conclusion of Selby et al⁷ to the effect that co-rotating arrays performed best in diffusers. A possible explanation for this discrepancy lies in the inability of the numerical model to allow for merging of co-rotating vortices: this would happen in a physical array but is precluded by the application of periodic boundary conditions in the calculations. The resources necessary to test this hypothesis were not available within the constraints of this project. The counter-rotating array of case G

shows a much more effective re-attachment of the flow, although, in common with case N (Fig. 5), the attachment of the flow is sufficiently strong to cause a separation from the upper wall.

Figure 6. Effect of array orientation: $\circ$ data from Ref. 15; — Case K (co-rotating); — Case G (counter-rotating).

IV. Conclusions

The co-rotating arrays investigated failed to achieve flow attachment on the diffuser wall. This confirms work done by Henry and Pearcey¹⁴ on flat plate boundary layer control that concluded that counter-rotating configurations are generally more effective than a co-rotating; the latter require significantly higher velocity ratios for any degree of success. Although it has been reported that co-rotating configurations can be very effective, this was not confirmed by this investigation; it is hypothesized that the effectiveness of co-rotating arrays is due to the tendency of the co-rotating vortices to merge together, a phenomenon that could not be modelled in this investigation because of the boundary conditions imposed.

Examination of vortex development indicates that counter-rotating vortices form quicker than co-rotating vortices and appear to be more compact, resulting in higher levels of vorticity. The counter-rotating vortices also appear more circular than the co-rotating: the co-rotating vortex core does not move away from the wall as the vortex grows, resulting in a slight deformation of the vortex on the wall side. The co-rotating vortices fail to remain inside the boundary layer until they dissipate. They remain tight to the lower wall for approximately 10 – 20% of the diffuser at which point they move away from the wall and out of the boundary layer where they can no longer influence the level of attachment.

For the counter-rotating cases, increasing velocity ratio and pipe diameter increased the degree of the flow attachment achieved. However, too much flow attachment causes flow separation on the upper wall. The best case, as measured by a pressure recovery factor, achieved attachment on both upper and lower walls. This best case is achieved using a pipe diameter of $0.025H$ and velocity ratio of 2.5.

If the application were high lift, then the consideration of maintaining flow attachment on two surfaces is no longer relevant. Instead fully attached flow on the lower surface is required and this is achieved by increasing the velocity ratio and diameter further. Increasing velocity ratio and diameter results in vortices having higher levels of vorticity and which penetrate further into the diffuser, while remaining tight to the lower wall and inside the boundary layer. The nature of counter-rotating vortices means that the flow profiles across the diffuser are not uniform. In the common flow up regions the flow is less attached to the wall, and in some cases remains separated, and in order to achieve fully attached flow across the diffuser the velocity ratio and pipe diameter must be increased to the point where flow in the common flow up region is attached.

However, velocity ratio and pipe diameter should not be increased without regard to other considerations. Energy consumption rises with both velocity ratio and pipe diameter. Moreover, at large values of velocity ratio and pipe diameter, it was found that the vortices burst through the shear layer on the lower wall, and attached itself to the upper wall, resulting in the flow being more fully attached to the upper wall in relation to the uncontrolled case.

References

- ¹Pearcey, H.H., "Boundary layer control for airfoils and wings," in *Boundary Layer and Flow Control*, Volume 2, Pergamon Press 1961.
- ²Horstmann, K.H. and Meyer, J., "Flight test results with HLFC system in HYLTEC," *Proceedings of KATnet Flow Control Workshop*, Université de Poitiers-ENSEMA, October 2004.
- ³Wallis, R.A., "The Use of Air Jets for Boundary Layer Control," Aerodynamics Research Labs., Aero Note 110, N-34736, Melbourne, Australia, January 1952.
- ⁴Johnston, J. and Nishi, M. "Vortex generator jets-A means for flow separation control," *J. AIAA*, Vol 28, N0 6, pp. 989-994, 1990.
- ⁵Lin, J.C., Howard, F.G., and Selby, G.V., "Investigation of several passive and active methods for turbulent flow separation control," *AIAA paper* 90-1598, June 1990.
- ⁶Compton, D.A. and Johnston, J.P., "Streamwise vortex production by pitched and skewed jets in a turbulent boundary layer," *J. AIAA*, Vol. 30, No. 3, pp. 640-647, 1992.
- ⁷Selby, G.V., Lin, J.C. and Howard, F.G., "Control of low speed turbulent separated flow using jet vortex generators," *Experiments in Fluids*, Vol. 12, pp. 394-400, July 1992.
- ⁸Zhang, X., "Counter-rotating vortices embedded in a turbulent boundary layer with inclined jets," *J. AIAA*, Vol 37, No. 10, pp. 1277-1284, 1999.
- ⁹Milanovic, I.M. and Zaman, K.B.M.Q., "Fluid dynamics of highly pitched and yawed jets in crossflow," *J. AIAA*, Vol 42, No 5, pp. 874-882, 2004.
- ¹⁰Johnston, J.P., "Pitched and skewed vortex generator jets for control of turbulent boundary layer separation", ASME/JSME Fluids Engineering Division, FEDSM99-6917, 1999.
- ¹¹Leslie, M., "Experimental and computational investigation of sub-boundary layer vortex generators," PhD dissertation, The Queen's University of Belfast, Belfast, 2000.
- ¹²Ahmad, K.A., Watterson, J.K., Cole, J.C. and Briggs, I., "Sub-Boundary Layer Vortex Generator Control of a Separated Diffuser Flow," *AIAA paper* 2005-4650, June 2005.
- ¹³Rixon, G.S. and Johari, H., "Development of a Steady Vortex Generator Jet in a Turbulent Boundary Layer," *J. Fluids Engineering*, Vol. 125, pp. 1006-1015, 2003.
- ¹⁴Henry, F.S. and Pearcey, H.H., "Numerical model of boundary layer control using air jet generated vortices," *J. AIAA*, Vol 32, No 12, pp. 2415-2425, 1994.
- ¹⁵Buice, C.U., Eaton, J.K., "Experimental Investigation of Flow Through an Asymmetric Plane Diffuser," *Data bank contribution to the Journal of Fluids Engineering*, Vol. 122, pp433-435, 2000.
- ¹⁶Iaccarino, G., "Prediction of the turbulent flow in a diffuser with commercial CFD codes," *Center for Turbulence Research, Annual Research Briefs*, 2000.
- ¹⁷DalBello, T., "CFD study of Buice Diffuser (2D)," *NPARC Alliance Validation Archive*, 2003.
- ¹⁸Fluent, version 6.1, Fluent Inc., Lebanon, NH, 2004.