On the Mechanisms of Secondary Flows in a Gas Vortex Unit

Kaustav Niyogi, Maria M. Torregrosa, Guy B. Marin, and Geraldine J. Heynderickx
Laboratory for Chemical Technology, Ghent University, Technologiepark 914, Gent B-9052, Belgium

Vladimir N. Shtern
Shtern Research and Consulting, Houston, TX 77096

Published online February 1, 2018 in Wiley Online Library (wileyonlinelibrary.com)

The hydrodynamics of secondary flow phenomena in a disc-shaped gas vortex unit (GVU) is investigated using experimentally validated numerical simulations. The simulation using ANSYS FLUENT® v.14a reveals the development of a backflow region along the core of the central gas exhaust, and of a counterflow multivortex region in the bulk of the disc part of the unit. Under the tested conditions, the GVU flow is found to be highly spiraling in nature. Secondary flow phenomena develop as swirl becomes stronger. The backflow region develops first via the swirl-decay mechanism in the exhaust line. Near-wall jet formation in the boundary layers near the GVU end-walls eventually results in flow reversal in the bulk of the unit. When the jets grow stronger the counterflow becomes multivortex. The simulation results are validated with experimental data obtained from Stereoscopic Particle Image Velocimetry and surface oil visualization measurements. © 2018 The Authors AIChE Journal published by Wiley Periodicals, Inc. on behalf of American Institute of Chemical Engineers

Keywords: gas vortex unit, secondary recirculation flows, computational fluid dynamics, Reynolds stress model, swirling flow

Introduction

A Gas Vortex Unit (GVU) is the basic design unit for several engineering applications ranging from nuclear rocket propulsion,1 vortex diodic valves,2 vortex scrubbing,3 nano-precipitation reactors,4-6 vortex amplifiers,7 and combustion.8 The unit, schematically shown in Figure 1a, consists of a disc-shaped (aspect ratio, $L/D_R \ll 1$) geometry confined on either side by two normal-to-unit-axis parallel flat plates, referred to as the end-walls. The distance between the two end-walls constitutes the length of the unit. A series of azimuthally inclined rectangular gas injection slots are located along the circumferential wall, through which the gas is introduced into the unit. The inclination of the slots imparts a strong azimuthal component to the inflowing gas. As a result, a strong swirling flow is established within the unit as the gas spirals toward the central unidirectional exhaust located on the front end-wall. The extended exhaust line (not shown in the schematic) finally directs the gas toward the outlet into the atmosphere. The degree of swirl imparted to the flowing gas in the GVU is determined by the injection slot angle ($\gamma$), highlighted in Figure 1 (inset: zoomed-in view of the slots where $\gamma = 10^\circ$ for the presented unit).

Although the primary flow in the GVU is swirling in nature, secondary flows may develop under specific operating conditions and geometrical design of the unit. Figure 1b schematically shows the secondary flow patterns that may appear in the GVU in form of in-plane velocity streamlines in an azimuthal plane ($\theta = \text{constant}$). The figure reveals that two distinct recirculation regions can develop: one in between the two end-walls referred to as Counterflow Region (CR) and another in the core of the exhaust region along the exhaust line, referred to as Backflow Region (BR). As the gas swirls toward the central exhaust axis, the low aspect ratio of the unit causes the two end-walls to significantly affect the flow topology resulting in the formation of a toroidal recirculation region (Counterflow Region, CR in Figure 1b) in between the two end-walls. In the core of the CR, the radial gas velocity is found to be directed radially outward, opposite to the main throughput flow direction in the GVU. The radially converging gas entering from the injection slots, thus splits in two parts (referred to as through-flow) flowing close to the two end-walls of the unit, while the bulk region is occupied by a flow reversal region. Remark that the gas flowing in the CR possesses a strong azimuthal velocity component as well. This azimuthal velocity component causes the CR to stretch along the entire circumference of the unit, in the form of a toroidal flow reversal zone. Figure 1b also shows that in the vicinity of the central axis, in the core of the exhaust line, the gas flows axially downward, from the exhaust outlet toward the GVU rear end-wall. Meanwhile the gas flowing in from the injection slots, leaves the unit through a peripheral annular region near the exhaust wall surrounding this recirculation zone. This second recirculation zone constitutes the backflow region, (BR in Figure 1b) and extends along the entire exhaust line. In the BR, the gas flows downward, sucked in from the ambient atmosphere.
The near-axis flow reversal in the form of the BR is a well-known hydrodynamic phenomenon commonly applied for solid–liquid and liquid–liquid separation in hydro-cyclones and effluent treatment by cavitation. The formation of the backflow region in swirling flows, discovered around the 1960s, is known as vortex breakdown (VB). The effects of VB on flow hydrodynamics can be considered positive or negative depending on the application of the GVU. For instance, in delta-wing aircrafts, VB can prove to be dangerous as it may cause abrupt variations in lift and drag forces on the aircraft wings. In combustion chambers, however, VB can be beneficial to stabilize flames. In natural swirling flows such as tornadoes, VB is known to decrease the destructiveness of the twister. A number of explanations for the appearance of the VB have been proposed in literature: (a) inertial wave roll-up, (b) collapse of the near-axis boundary layer, (c) flow separation, (d) transition from convective to absolute instability. A recent view is that VB develops via the swirl-decay mechanism. The present study also shows that the swirl-decay mechanism can explain the formation of the BR, as discussed in more detail in the results section.

The counterflow region (CR) is a comparatively less researched hydrodynamic phenomenon. It was accidentally detected in a series of experimental studies conducted on a vortex unit for the development of a nuclear rocket propulsion engine conceived in 1960s. The presence of the CR was experimentally detected using techniques such as pitot tubes and hot wire anemometry. The reversal of the radial velocity in the CR was unexpected and was initially considered to be an artifact of the measurement. A detailed experimental study however confirmed the radial velocity reversal in the bulk of the disc part of the GVU. It also revealed that the CR occupies a major part of the GVU disc volume (Figure 1b). The authors postulated that the CR develops in the GVU due to entrainment of the bulk flow gas by the near-wall jets that develop in the boundary layers adjacent to the two end-walls of the unit. Initial tracer experiments were performed using air bubbles and concentrated milk powder to visualize the CR in between the end-walls of the GVU. It was observed that when a strong swirl component is imparted to the injected gas, a persistent “donut”-shaped, toroidal ring of tracers is formed in the bulk disc part of the GVU. The author concluded that, as the tracer particles were retained inside the cavity for prolonged times, the bulk region of the GVU is devoid of any net radially inward gas flow. That is, all injected gas actually flows through two boundary layers formed near the end-walls of the unit. As the observations were mostly visual the author associated the CR with zero radial velocity rather than flow reversal. Laser Doppler Anemometry (LDA) measurements of GVU flow turbulence confirmed that at high Reynolds numbers (Re~7,000–13,000) the swirling flow in the GVU is highly turbulent. The turbulent kinetic energy of the gas increases radially inward along the disc part of the unit and peaks on entering the exhaust region. To better understand the complex GVU flow hydrodynamics, analytical models describing the azimuthal and radial velocity distributions in the GVU were formulated and compared with experimental data. Although the model predicted GVU bulk flow quantities well, the disagreement between the model and experimental data near the end-walls resulted in speculation on the accuracy of the experimental measurements close to the end-walls. Intrusive flow measurement techniques such as pitot tubes or yaw probes disrupt the flow while techniques such as LDA have limited accuracy in the near-wall regions due to wall reflections. Hence, a numerical investigation into the complex secondary flow hydrodynamics in the GVU becomes essential. Initial simulations were performed using a laminar approximation of the GVU flow and a finite difference methodology. The simulations qualitatively showed that the swirl structure in the GVU is irrotational in nature in the bulk flow rather than solid-body rotational. The effect of varying gas injection slot angles was investigated. With increasing swirl, the radial flow reversal was located close to the chamber outer periphery. However, as the model used a laminar approximation, its accuracy was limited to low Reynolds numbers. Turbulent flow simulations using the two-parameter k-ε turbulence model in radically converging confined flows
demonstrated that turbulence modeling is crucial in predicting the flow field. Radially converging nonswirling flow results in flow laminarization due to acceleration. In the presence of swirl, the boundary layers in the GVU for a given Re become comparatively thinner, intensifying the local turbulence production near the walls and increasing the numerical complexity of turbulence modeling in the GVU. One major drawback of the two-parameter eddy viscosity models is that the scalar eddy viscosity cannot account for the turbulent anisotropy arising from the strong streamline curvature in the GVU. Second-order turbulence models such as the Reynolds Stress Model (RSM) prove to be highly applicable in this regard. A turbulence model such as RSM directly solves the Reynolds stresses in the flow field and captures the effect of streamline bending on turbulence. A successful implementation of RSM turbulence modeling in GVU simulations demonstrated the capability of Computational Fluid Dynamics (CFD) software ANSYS FLUENT to predict both bulk and secondary flows in the GVU. The numerical results quantitatively agreed with the experimental data thus validating the numerical code.

The present work provides an in-depth analysis of swirling flows and more specifically the associated secondary flow phenomena in GVUs. Two-dimensional (2D) axisymmetric simulations of an azimuthal plane of the GVU are performed with the commercial finite volume software package ANSYS FLUENT. Incompressible steady-state simulations are performed using the RSM turbulence modeling approach. The numerical model is first validated with experimental data provided by Stereoscopic Particle Image Velocimetry (SPIV) and surface oil flow visualization in a GVU setup in the authors’ laboratory. Additionally, a comparison is made between swirl-free purely radially converging flow and swirling flow in the GVU.

**Methodology**

**GVU setup description**

The schematic of the GVU experimental setup simulated in the present work is shown in Figure 1a. The GVU consists of a disc-shaped confined static geometry positioned along a horizontal axis. The geometrical dimensions of the experimental GVU can be found in Table 1. Pressurized gas (in the present study, air: $\rho = 1.225 \text{ kg/m}^3$, $\mu = 1.75 \times 10^{-5} \text{ kg/m s}$) is sent through 12 feeding pipes into a distributor jacket. From the jacket, the gas is directed into the main unit through 36 equilibrium injection slots uniformly located along the circumferential wall of the unit. The slots are azimuthally inclined at an angle of 10°, thereby imparting a strong azimuthal velocity component to the injected gas. The gas spirals inward in the disc part and leaves the unit axially through a centrally located unidirectional exhaust. Under the specified operating conditions, the gas velocities obtained in the geometry are lower than 0.3 Mach number (Ma), and hence the flow is considered to be incompressible.

Figure 1c shows the Stereoscopic Particle Image Velocimetry (SPIV) measurement configuration with two cameras that are angularly positioned to measure the three gas velocity components on a single laser sheet illuminating a 2D azimuthal plane passing through the GVU. To measure the gas flow field, tracer oil droplets are injected along with the gas from the injection slots in the GVU. The size of the droplets is chosen such that the Stokes number is less than 1, ensuring that the droplets follow the azimuthal gas flow. The SPIV technique is useful to obtain a visual proof of the presence of the counterflow region and validating the numerical technique used in the present study. Also, it is the first time in literature that a 2D visualization of the CR on an azimuthal plane in the GVU is attempted using the SPIV technique. However, as shown in the present study, the experimental technique has its own limitations. In fast-swirling flow the large centrifugal force increases the measuring error in the radial direction. The significantly denser oil tracer droplets are subject to stronger centrifugal forces as compared to the gas molecules, causing a radially outward shift in the PIV measured radial and axial flow fields. This is further explained in the results and discussions section.

**Numerical model**

In the present study, steady 2D axisymmetric flow simulations in an azimuthal ($\theta = \text{constant}$) plane of the GVU are performed, using the commercial CFD software package ANSYS FLUENT. The simulated plane corresponds to a GVU section shown in Figure 1b. The effect of the gravitational force on the flow hydrodynamics is negligible due to the low gas density and the dominant centrifugal force. Hence gravity is not considered in the present study. The experimental unit has 36 discrete gas injection slots. In the present study the 2D axisymmetric flow assumption results in a hypothetical situation where the gas uniformly enters the geometry over the entire circumference, at an angle corresponding to the actual experimental slot angle. This gas entry is henceforth referred to as circumferential injection. The axisymmetric assumption is considered valid, based on previous work, where it was shown that the use of multiple equidistant injection slots (36) uniformly distributes the gas in the GVU and makes the flow nearly axisymmetric even in the vicinity of the injection slots. The no-slip boundary condition is imposed at the end-walls of the GVU. The Reynolds Averaged Navier-Stokes (RANS) turbulence modeling approach is adopted, as the main goal of the present work is to obtain ensemble-averaged GVU hydrodynamics. Turbulence modeling for highly swirling flows can be challenging. Swirling flows often encounter high streamline curvature and the Reynolds stresses exhibit anisotropy owing to this curvilinear motion. Two-parameter eddy-viscosity turbulence models quantify turbulence using a scalar in the form of turbulent viscosity and fail to account for the directional dependence of the turbulent stresses. Hence, the Reynolds Stress Model (RSM) is used in the present study as it directly calculates the Reynolds stresses in the flow domain, and captures the turbulence anisotropy. The governing conservation and turbulence model equations are presented in Table 2.

By virtue of an initial systematic mesh study, the optimal mesh for resolving the primary and secondary flow characteristics in the GVU is found to consist of 200,000 quadrilateral cells as shown in Figure 2. The cell size varies from 1 mm in the bulk region to approximately 0.001 mm inflation layer cells near the GVU end-walls. The inflation layers are added near the end-walls to accurately capture the near-wall flow. Mesh resolution results in wall $y^+$ values of the order of 1.

| Table 1. Geometrical Dimensions of the Experimental GVU |
|-----------------------------|---------|
| GVU Jacket diameter         | 0.7 m   |
| GVU Circumferential wall diameter ($D_{e0}$) | 0.54 m  |
| GVU exhaust diameter ($D_{ex}$)    | 0.2 m   |
| GVU exhaust diameter ($D_{ex0}$)  | 0.15 m  |
| GVU length ($L_{e0}$)          | 0.1 m   |
| Number of injection slots (N)  | 36      |
| Slot width ($L_s$)             | 0.002 m |
for both the near-wall and bulk flow quantities in a previous study is validated in great detail with experimental data. The turbulence model and wall modeling used in the pre-treatment for resolving the flow near the walls of the geometry. Stress-omega RSM turbulence model applies enhanced wall treatment, implemented through the automatic wall treatment, which highlights the inflation layers

dynamics in the GVU. The velocity streamlines obtained from the GVU bulk flow were shown to quantitatively agree, indicating the numerical model correctly predicts bulk flow hydrodynamics in the GVU. The velocity streamlines obtained from simulations were compared with the oil droplet tracks on the rear end-wall of the GVU. The qualitative and quantitative agreement between the two sets of data validated the applicability of the numerical model in the boundary layers in the near-wall regions as well. Thus, the simulations of GVU flow were validated both in the bulk region and close to the end-walls in the boundary layers in the unit.

### Solution methodology

A second-order accurate spatial discretization scheme is applied to solve the momentum and turbulence equations. Pressure corrections are computed using the body force weighted Pressure Staggering Option (PRESTO!) scheme. To solve the set of equations the segregated pressure-based Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm is used. The scaled residuals in the mass and momentum balances are set to $10^{-5}$ as condition for convergence. The simulations are performed on an AMD-based Linux 16-core clusters. One simulation for the 2D axisymmetric GVU flow requires about 2 h of CPU time. The simulation data are exported to MS Excel and Tecplot v.2015 for post-processing and further analysis.

### Results and Discussion

#### GVU flow characterization

The azimuthally inclined injection of the gas in the GVU imparts a strong swirling motion in the disc part of the GVU while the overall gas mass flow rate through the unit is radially inward. Moreover, as the circumferential flow surface area of the GVU decreases with decreasing radius, the superficial radial gas velocity increases as the gas approaches the central exhaust to maintain constant mass flow rate through consecutive cross-sectional flow areas. Meanwhile, the azimuthal gas velocity in the bulk flow increases with decreasing radius exhibiting a flow pattern similar to a potential vortex.
The no-slip condition at the end-walls causes both the radial and azimuthal velocity components to drop to zero in the near-wall boundary layers. The azimuthal velocity decrease from the bulk flow value to zero in the boundary layers shows a monotonic smooth profile. In contrast, the radial velocity profile changes counter-intuitively. Sharp peaks of the radial velocity in the form of near-wall jets appear near the end-walls. The converging radial flow, the high degree of swirl and the presence of the end-walls result in the formation of near-wall boundary layers, thus resulting in a highly complicated flow pattern in the GVU geometry. To better understand all these flow phenomena, it is helpful to decouple the effects of the converging radial throughput and the swirl. This is achieved by defining two control parameters in the form of characteristic numbers describing the GVU flow.

The first parameter is the radial Reynolds number \( Re \) which is defined as

\[
Re = \frac{\rho U_{r,i} L_R}{\mu} = \frac{G_M L_R}{A_i \mu}
\]

where \( U_{r,i} \) is the superficial radial gas velocity component at the GVU circumferential injection slots, \( G_M \) is the gas mass flow rate, \( L_R \) is the length of the unit, \( A_i \) is the circumferential area at injection, and \( \rho \) and \( \mu \) are the density and dynamic viscosity of the operating gas respectively. For a given aspect ratio of the geometry and a given gas mass flow rate, \( Re \) remains constant, independent of the degree of swirl in the flow.

The second control parameter is the swirl ratio \( S \), defined as

\[
S = \frac{U_{\theta,i}}{U_{r,i}}
\]

where, \( U_{\theta,i} \) is the azimuthal gas velocity component at the GVU injection. An \( S \) value of 0, for a given \( Re \), corresponds to a swirl-free flow through the unit. Remark that \( S \) is determined by the injection angle \( \phi \) of the injection slots in the actual experimental geometry. An increase in \( S \), at constant \( Re \), imparts a higher degree of swirl to the flow in the GVU for a constant gas mass flow rate. It is interesting to mention here that at high \( Re \) values, the flow becomes predominantly convective and the viscous contribution of gas becomes negligible making the flow topology independent of \( Re \). In contrast, a variation of \( S \) can significantly change the flow topology even at high \( Re \) values.

As the main focus of the present work is to investigate highly swirling flow features in the GVU, \( S \) is set to be larger than 1 for most simulation cases. However, it is instructive to first investigate a swirl-free flow \( (S = 0) \) case for reasons of comparison.

**Swirl-free flow \( (Re = 13700; \ S = 0) \)**

Figure 3a shows the calculated radial gauge pressure profile along the centerline \((z = 0.05 \ m)\) of the GVU for swirl-free, purely radially converging flow. The positive pressure gradient \((\partial P/\partial r > 0)\) from the circumferential wall to the central exhaust of the disc \((0.06 < r < 0.27 \ m)\) directs the injected gas toward the exhaust. The figure shows that in the exhaust region \((r < 0.06 \ m)\) an adverse pressure gradient develops. To understand the origin of this adverse pressure gradient, the in-plane gas velocity vector field along the azimuthal plane is plotted in Figure 3b. It can be seen from the figure, that the radially converging gas experiences a strong streamline curvature from the radial to the axial direction in the exhaust region to align the flow with respect to the central exhaust line. This streamline bending compounded with the high \( Re \) flow results in the formation of two local recirculation regions, highlighted in Figure 3b. As the radially converging gas reaches the exhaust line, the bulk flow makes a 90° anticlockwise turn, causing a local flow separation just downstream of the point of intersection of the exhaust wall and the front end-wall \((z = 0.1 \ m)\) of the unit. This results in the development of an adjacent thin recirculation region (Recirculation zone 1, Figure 3b). The drastic change in the flow direction causes the flow to locally accelerate in accordance with the inviscid flow theory. The color change of the velocity vectors in the vicinity of the abovementioned intersection, shown in Figure 3b, highlights this increase in the gas velocity. The flow acceleration decreases the local static pressure in the vicinity of the intersection, and generates an adverse pressure gradient after the exhaust bend, resulting in the formation of a local Recirculation zone 1.

The second recirculation zone develops near the intersection of the rear end-wall of the disc and the axis of the unit (Recirculation zone 2, Figure 3b). The gas approaches the axis at high \( Re \) condition which causes flow separation near the axis.
due to the streamline curvature of the flow. The flow separation generates an adverse pressure gradient and a local pressure maximum near the axis-wall intersection, corresponding to the inviscid fluid theory, as seen in the radial pressure profile ($r < 0.06 \text{ m}$) in Figure 3a. The elevated pressure values direct the gas away from the rear end-wall ($z = 0 \text{ m}$), resulting in the formation of a local recirculation zone. Remark that this zone remains confined to the vicinity of the GVU rear end-wall and does not extend extensively into the exhaust line. Recirculation zone 2 does not significantly constrict the area for gas flow in the exhaust.

Figure 4a shows the axial profile of the radial velocity at different radial positions in the disc part of the unit. As the flow approaches the exhaust the radial velocity of the gas increases with decreasing radius. This flow acceleration causes the radial velocity profile to be nearly uniform over almost the entire length of the unit (at $r = 0.23$ and $0.18 \text{ m}$). Only in the thin boundary layers formed near the two end-walls, the no-slip boundary condition causes the radial gas velocity to monotonically decrease to zero, as can be seen more clearly in the zoomed-in near-wall profiles in Figure 5. Further downstream, closer toward the exhaust, the axial symmetry of the radial velocity profile with respect to the centerline ($z = 0.05 \text{ m}$) breaks. Suction generated due to the flow acceleration near the intersection of the front end-wall ($z = 0.1 \text{ m}$) and the exhaust increases the radial velocity near the front end-wall ($z = 0.1 \text{ m}$) as compared to the rear end-wall ($z = 0 \text{ m}$).

For swirl-free flow in the GVU, the radial flow acceleration significantly affects the turbulent characteristics of the gas as well. In converging flows, fluid acceleration results in relaminarization of the turbulence in the downstream direction. The kinetic energy required by the mean flow to accelerate is obtained from its turbulent counterpart. Hence the turbulence gradually decreases downstream. The gradually reducing circumferential area in the GVU geometry in the direction of flow represents a similar converging flow scenario. Hence, the acceleration of the radial velocity is expected to cause flow relaminarization in a swirl-free GVU flow. To test this hypothesis, the turbulent intensity field in the disc part of the unit is shown in Figure 4b. The turbulent intensity, set at 5% at the injection, is seen to decrease downstream in the GVU disc part, confirming flow relaminarization. The noncolored region in Figure 4b near the rear end-wall of the unit toward the central axis corresponds to the high turbulence region due to the presence of Recirculation zone 2. As turbulence significantly increases in this part of the geometry, Recirculation zone 2 is excluded from the turbulence intensity color map to capture the lower values in the turbulence field in the disc part of the GVU. Owing to this laminar nature (turbulence intensity <5%) of swirl-free flow in the GVU, an analytical solution of the velocity profile in the near wall boundary layer regions can be obtained.

For radially converging sink flow, mass conservation yields
\[
U_r(\text{bulk}) = -\frac{Q}{2\pi r LR} = -\frac{f_0}{r} \tag{3}
\]

where \(Q\) is the volumetric gas flow rate and \(f_0\) is a constant for a given gas flow rate and unit length. This equation remains valid for the bulk flow inside the GVU disc part, except in the vicinity of the end-walls where boundary layers develop as the radial velocity, \(U_r\), drops to zero. The pressure field in the unit is linked with the velocity field through the Navier-Stokes equation in the radial direction as shown in Table 2, Eq. 1.2. In radial direction the equation reduces to

\[
\rho U_r \frac{\partial U_r}{\partial r} = -\frac{\partial P}{\partial r} + \mu \left[ \frac{1}{r^2} \frac{\partial}{\partial r} \left( r \frac{\partial U_r}{\partial r} \right) - \frac{U_r}{r^2} + \frac{\partial^2 U_r}{\partial z^2} \right] \tag{4}
\]

for swirl-free steady flow.

At high \(Re\), the bulk flow is dominantly convective in nature and viscous contributions can be neglected, simplifying Eq. 4 to

\[
\frac{\partial P}{\partial r} = -\rho U_r \frac{\partial U_r}{\partial r} = \frac{\rho f_0^2}{r^3} \tag{5}
\]

Over the boundary layer, the radial velocity can be expressed as \(U_r = -\frac{f(z)}{r}\). As \(dP/dr\) is nearly uniform across the boundary layer, its bulk-flow value, \(dP/dr = \rho f_0^2/r^3\) can be used henceforth. Combining all, Eq. 5 results in

\[
\frac{(f_0^2 - f^2)}{r^3} = -\frac{\nu f z}{r} \tag{6}
\]

where \(\nu = \mu/\rho\) is the kinematic viscosity of the gas and the subscript “zz” denotes the second-order derivative with respect to \(z\). As the boundary layer thickness is small compared to the radial coordinate in the GVU, \(r\) can be approximated by a local value \(r_0\) (say, \(r_0 = 0.23\) m). Introducing the dimensionless variables \(\varphi = \frac{f(z)}{f_0}\), and \(\zeta = \left(\frac{f(z)}{f_0}\right)^{1/2}\) transforms Eq. 6 into

\[
\varphi \zeta = \varphi^2 - 1 \tag{7}
\]

where the subscript “\(\zeta\)” denotes the differentiation with respect to \(\zeta\). The boundary conditions are \(\varphi = 0 \) at \(\zeta = 0\) (no-slip) and \(\varphi \to 1\) as \(\zeta \to \infty\) (the radial velocity tends to its bulk value). Multiplying all terms of Eq. 7 with \(\varphi \zeta\) and integrating results in

\[
\frac{\varphi^2}{2} = \frac{\varphi^3}{3} - \varphi + \frac{2}{3} \tag{8}
\]

where the last term on the right-hand side of the equation is an integration constant satisfying the boundary condition, \(\varphi \to 1\) as \(\zeta \to \infty\). One more integration, satisfying the no-slip condition, yields

\[
\zeta = \int \left( \frac{4}{3} - 2\nu + \frac{\nu^3}{2} \right) \, du \tag{9}
\]

where the integration runs from 0 to \(\varphi\).

In Figure 5 this analytical solution (red line) for \(r = 0.23\) m is compared with the corresponding radial velocity profile obtained from the numerical simulations (dashed). As can be seen from the figure, the two profiles match quantitatively, validating that laminarization holds for swirl-free (\(S = 0\)) flow. Equation 9 demonstrates that for swirl-free flow in the GVU, due to the relaminarization phenomenon, an analytical solution of the near-wall transformation of the radial velocity can be obtained.

**Swirling flow (\(Re = 13700\); \(S = 5, 12\))**

**Bulk Flow Hydrodynamics.** Adding an azimuthal velocity component to the injection of the GVU significantly alters the flow pattern in the unit. As the superficial radial velocity, or the gas mass flow rate for incompressible flow, at injection remains constant, \(Re\) remains unchanged and the net throughput in the unit remains constant. However, a finite swirl ratio imparts additional azimuthal momentum to the gas causing it to start spiraling toward the exhaust. The larger the swirl ratio, the higher is the number of rotations the gas completes inside the disc part of the geometry before exiting through the central exhaust. As the value of \(S\) is always considered to be higher than 1 in the present study, the gas undergoes multiple complete rotations in the unit before reaching the exhaust. Figure 6 shows the radial profile of the azimuthal velocity component along the centerline (\(z = 0.05\) m) in the steady-state swirl flow inside the unit. In the disc part of the unit (\(0.075 < r < 0.27\) m) the azimuthal velocity component increases with decreasing radius. This flow behavior has been shown previously to qualitatively represent free-vortex flow, where the gas angular momentum is nearly conserved in radial direction. The quantitative deviation of the azimuthal velocity profile from the hyperbolic free-vortex flow profile arises from the fact that the friction losses encountered by the gas at the end-walls reduce \(U_0\) to some extent. However, the qualitative trend of increasing \(U_0\) with decreasing radius is retained. In the exhaust region of the unit (\(0 < r < 0.0075\) m), the BR and an intense turbulence develop causing the swirling structure to break. This explains the steep drop in the azimuthal velocity component for \(r < 0.06\) m observed, see Figure 6.

Figure 7a shows the radial profile of the static gauge pressure along the unit centerline (\(z = 0.05\) m). Compared to the swirl-free flow (Figure 3a), the pressure drop over the disc part significantly increases for when swirl is imparted to the flowing gas. This increase in pressure drop across the unit partly arises from a balance between the centrifugal force generated on the gas and the radial pressure drop, and partly arises due to the formation of an extended BR in the exhaust line as explained below.

The velocity vector field along the azimuthal plane for swirling flow in the GVU is shown in Figure 7b for \(S = 12\). Two interesting phenomena can be observed. First, Recirculation zone 1 near the exhaust wall as seen in Figure 3b
The velocity profile suggests that in the initial section of the exhaust line, near \( z = 0.1 \) m, the gas is swirling comparatively stronger compared to the gas near the outlet of the exhaust line (\( z = 0.6 \) m). A swirling vortex possesses a static pressure minimum at its center on the axis of rotation.\(^{12}\) The stronger the rotation of the elements, the lower is the local pressure at the axis. Naturally it follows that the stronger swirling structure near \( z = 0.1 \) m reduces the local static gauge pressure value at the GVU axis at \( z = 0.1 \) m to a lower value than the near-axis pressure value at \( z = 0.6 \) m. This reverse pressure gradient along the axis of the unit causes flow reversal by sucking ambient gas into the exhaust line and pushing it toward the rear-end wall of the unit. This flow reversal constitutes the BR in the GVU as seen in Figure 7b. One of the major consequences of this BR is that its presence considerably constrains the net flow surface area in the exhaust line available for the gas to leave the unit. This flow area constriction is one of the causes for the overall pressure drop increase across the unit, as seen in Figure 7a.

As previously mentioned, another reason for the pressure drop increase can be attributed to the influence of the centrifugal force on the pressure-velocity coupling in the GVU swirling flow. To better understand this mechanism, the radial steady-state Navier-Stokes equation in cylindrical coordinates is analyzed. The Navier Stokes equation (Table 2, Eq. 1.2) can be expressed in radial coordinates as

\[
\rho \left( U_r \frac{\partial U_r}{\partial r} + \frac{U_\theta U_r}{r} \frac{\partial U_r}{\partial \theta} + \frac{U_z U_r}{r} \frac{\partial U_r}{\partial z} \right) = -\frac{\partial P}{\partial r} + \mu \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial U_r}{\partial r} \right) - \frac{U_r^2}{r^2} - 2 \frac{\partial U_\theta}{\partial \theta} - \frac{\partial^2 U_r}{\partial z^2} \right]
\]

As the GVU flow can be considered to be axisymmetric, the azimuthal velocity and the azimuthal gradients can be neglected. The viscous contribution can also be neglected as, given the high Re, the flow is highly convective in nature. For axisymmetric swirl-free flow through the GVU (\( S = 0 \)), the centrifugal acceleration term \( U_r^2/r \) is absent and the pressure drop over the GVU unit can be approximated as

\[
\frac{\partial P}{\partial r} = -\rho \left( U_r \frac{\partial U_r}{\partial r} + U_z \frac{\partial U_r}{\partial z} \right)
\]

Figure 9a validates these assumptions by comparing the pressure gradient obtained from numerical simulations to the pressure drop over the GVU unit calculated using the above approximation.

Figure 7. (a) Radial profile of static gauge pressure along \( z = 0.05 \) m, calculated by solving the set of Eqs. 1.1–1.5 given in Table 2 and (b) In-plane velocity vector field along axisymmetric azimuthal plane, calculated by solving the set of Eqs. 1.1–1.5 given in Table 2 under operating conditions (\( G_m = 0.4 \) kg/s, \( Re = 13700, S = 12 \)). Color indicates the velocity magnitude values of the gas.

Figure 8. Axial profile of azimuthal velocity along \( r = 0.06 \) m, calculated by solving the set of Eqs. 1.1–1.5 given in Table 2 under operating conditions (\( G_m = 0.4 \) kg/s, \( Re = 13700, S = 12 \)).
gradient as computed from Eq. 11. Excellent agreement is seen between the two curves.

When swirl is introduced in the GVU flow ($S = 12$), the centrifugal acceleration term is retained in Eq. 10, resulting in

$$\rho \left( U_r \frac{\partial U_r}{\partial r} + U_z \frac{\partial U_r}{\partial z} - \frac{U_r^2}{r} \right) = -\frac{\partial P}{\partial r} \quad (12)$$

To understand the extent of influence of the centrifugal term on the overall pressure drop of the unit, Figure 9b is plotted. The figure separately compares the pressure drop obtained from simulations of swirling GVU flow with the contribution from the centrifugal acceleration term ($\frac{U_r^2}{r}$) in Eq. 12 on the one hand, and the radial and axial velocity gradient terms ($U_r \frac{\partial U_r}{\partial r} + U_z \frac{\partial U_r}{\partial z}$) in Eq. 12 on the other hand. The figure shows that the contribution of the centrifugal term significantly exceeds its radial and axial counterparts, and singularly accounts for the entire pressure drop over the disc part of the unit. This helps to further simplify Eq. 12, resulting in the cyclostrophic balance, given by

$$\frac{U_r^2}{r} = -\frac{\partial P}{\partial r} \quad (13)$$

Thus, the high azimuthal velocity imparted to the flowing gas in the GVU increases the unit pressure drop as compared to swirl-free flow ($S = 0$). This effect, compounded with the reduction of the cross-sectional flow surface area near the exhaust due to the formation of the BR, elevates the total pressure drop across the GVU for the swirling flow case.

Formation of Counterflow in the Disc Part of the GVU.

In the bulk flow through the disc part of the GVU, the cyclostrophic balance (Eq. 13) between the radially inward pressure force and the radially outward centrifugal force on the gas holds well. However, the cyclostrophic balance does not hold in the close vicinity of the end-walls due to the boundary layer formation. Imposing the no-slip boundary condition sets the azimuthal velocity component to zero at the end-walls, as seen in Figure 10 where the axial profile of the scaled azimuthal velocity is plotted for $r = 0.21$ m. As $U_0$ diminishes in the boundary layers, there is a corresponding drop in centrifugal force exerted on the gas elements. In contrast, the static gauge pressure remains unaffected by the near-wall boundary layer. The scaled static gauge pressure profile in the near-wall region in Figure 10 illustrates this. For convenient comparison, both the static gauge pressure and the azimuthal velocity in Figure 10 are scaled with respect to their respective maximum values along the GVU length at $r = 0.21$ m.

![Figure 9. (a) Radial profile of static pressure gradient along $z = 0.05$ m: (—) calculated by solving Eq. 11; full line, from numerical simulation, calculated by solving the set of Eqs. 1.1–1.5 given in Table 2 under operating conditions ($G_M = 0.4$ kg/s, $Re = 13700$, $S = 0$) and (b) Radial profile of static pressure gradient along $z = 0.05$ m: (—) calculated by solving Eq. 11; (—) calculated by solving Eq. 13; full line, from numerical simulation, calculated by solving the set of Eqs. 1.1–1.5 given in Table 2 under operating conditions ($G_M = 0.4$ kg/s, $Re = 13700$, $S = 12$).](image)

![Figure 10. Axial profiles of scaled static gauge pressure and azimuthal velocity and normalized radial velocity at $r = 0.21$ m: (**) azimuthal velocity, scaled by its maximum value along GVU length at $r = 0.21$ m, calculated by solving Eq. 3; (—) radial velocity, normalized by the superficial velocity at $r = 0.21$ m cross-sectional surface area in the GVU; full line, static gauge pressure, scaled by its maximum value along GVU length at $r = 0.21$ m, calculated by solving the set of Eqs. 1.1–1.5 given in Table 2 under operating conditions ($G_M = 0.4$ kg/s, $Re = 13700$, $S = 12$).](image)
maximal values along the length of the unit at $r = 0.21$ m. The drop in the centrifugal force near the end-walls causes the cyclostrophic balance to break. The gas in this region is directed radially inward by the unbalanced pressure gradient.

The monotonic near-wall profile of the radial velocity in swirl-free flow ($S = 0$), seen in Figure 4a, is lost resulting in the development of a local radial velocity peak in the vicinity of each end-wall boundary layer. This local radial velocity peak near the rear end-wall can be seen in Figure 10, where the radial velocity, normalized by the superficial radial velocity at $r = 0.21$ m, is plotted along the length of the unit. The near-wall peaks in the radial velocity profile when swirl is introduced in the GVU flow are referred to as the near-wall jets.29

The axial profiles of the radial velocity component of the gas at $r = 0.21$ m are shown in Figure 11a for two different swirl ratio cases $S = 5$ (solid line) and $S = 12$ (dashed line). The figure shows that for a comparatively lower degree of swirl ($S = 5$), the effect of the near-wall jets remains confined to the close vicinity of the end-walls. The radial velocity in the bulk flow for $S = 5$ is slightly lower than the superficial gas velocity magnitude (2.47 m/s) at the given cross-sectional area ($r = 0.21$ m) to account for the excess gas entrained by the near-wall jets. However, the radial velocity along the entire length of the unit remains negative as seen in Figure 11a indicating that the gas flow throughout the entire length of the unit remains radially inward. For an increased value of the swirl ratio ($S = 12$), the near-wall jets become stronger as indicated by the radial peak velocity magnitudes in Figure 11a. As a consequence, the jets entrain a higher volume of gas along with them toward the central exhaust, and the radial velocity in the bulk region for $S = 12$ becomes positive causing a local flow reversal, from the central axis toward the GVU injection. This radial flow reversal between the two end-walls of the GVU constitutes the CR. Figure 11b, showing the in-plane velocity vector field in an azimuthal plane, helps to visualize this CR. It can be seen that a pair of counter-rotating vortices develop in the bulk region of the flow. Remark that, in the entire disc part of the GVU, the azimuthal velocity component still remains the dominant velocity component. It is one order of magnitude higher than the corresponding local radial velocity component (not shown). This implies that the 2D representation of the CR as seen in Figure 11b is actually an in-plane projection of a toroidal ring-like 3D structure in the GVU geometry.

The CR in the GVU has previously been indirectly observed in literature through pitot tube measurements.22 However, this intrusive experimental technique could provide velocity data only at specific sections of the geometry, and at the expense of disrupting the local flow phenomena. The authors clearly demonstrated how immensely challenging it is to experimentally quantify the CR and produced an in-plane velocity field as shown in Figure 12b. Nonetheless, obtaining experimental data on secondary flow phenomena in the GVU is highly crucial for the validation of simulated GVU hydrodynamics as in the present study. Hence, the more rigorous SPIV technique is adopted for the first time in literature to validate the numerical model used to simulate counterflow fields. Figure 12 compares the experimental in-plane (axial and radial) velocity streamlines along the azimuthal plane of the GVU generated using SPIV (Figure 12a) with those obtained from the simulations (Figure 12b). The experimental and simulated streamlines both show the formation of counter-rotating vortices and radially outward reverse flow in the bulk region in between the end-walls of the unit. However, the quantitative agreement between experiment and simulation is not completely satisfactory. The authors believe that this discrepancy can be explained as follows. The Stokes number for the tracer droplets being less than 1 is a strong indication that the droplets will follow the gas along the main velocity component direction which, in the case of the swirling GVU flow, is azimuthally oriented. It is questionable, however, whether the tracer droplets will accurately follow the bulk gas flow in the radial and axial directions. The axial and radial velocity components (0.5-2 m/s) in the bulk region of the GVU are two orders of magnitude smaller than the azimuthal velocity (45–100 m/s). The magnitude of the radial and axial velocities in the bulk flow are so low that they may be in the inaccuracy range of
SPIV measurements (3% for azimuthal velocities). It is known from literature that the SPIV error can be high for radial velocity measurements in highly swirling flows. In the bulk region, the centrifugal force on a gas element is balanced by the radial pressure gradient (Eq. 13). The centrifugal force acting on an oil droplet, however, is significantly higher and is not balanced by the radial pressure gradient. The latter results in a radially outward shift toward the circumferential wall of the in-plane velocity field (axial and radial) of tracer droplets in an azimuthal plane. Comparing the in-plane (radial and axial) velocity fields in Figures 12a, b confirms this conjecture. Nevertheless, the SPIV data qualitatively capture the presence of a CR in the bulk of the GVU without any measurement intrusion into the flow field, validating the numerical prediction of the recirculation region. Moreover, the numerical results compare well with the velocity flow fields obtained in previous numerical studies.

The presence of near-wall jets significantly alters the turbulence characteristics of the swirling GVU flow. Figure 13a shows the profile of the turbulent intensity along the length of the unit at \( r = 0.21 \, \text{m} \) for \( S = 12 \). It can be seen that close to the two end-walls of the unit, high turbulence production takes place. These regions correspond to the locations where the near-wall jet peaks are located. The jets result in high velocity gradients in the end-wall boundary layers. The corresponding intense shear between the gas layers results in turbulence production. Due to such high values of turbulence, the near-wall axial profile of the radial velocity can no longer be analytically derived, as was possible for the swirl-free \( (S = 0) \) GVU flow (Eq. 9). Furthermore, the presence of the CR significantly alters the turbulence characteristics in the bulk flow as well.

Figure 4b has already demonstrated the relaminarization effect of flow acceleration on swirl-free \( (S = 0) \) flow in the GVU. Figure 13b, comparing the turbulence intensities of swirl-free \( (S = 0) \) and swirling \( (S = 12) \) GVU flow, attests to this observation. The turbulent intensity, set at 5% at injection, decreases with decreasing radius for swirl-free flow \( (S = 0) \). In strongly swirling flow \( (S = 12) \), however, the CR prevents the downstream flow from laminarizing. The turbulent intensity increases with decreasing radius as the solid curve in Figure 13b illustrates. The turbulent intensity shows a local maximum near \( r = 0.22 \, \text{m} \) for \( S = 12 \). For reasons of comparison this cross-section is highlighted in Figure 11b. By comparing Figures 11b and 13b, it is seen that the turbulent intensity maximum coincides with the GVU region where the injected gas stream meets the reverse flowing (radially outward flowing) gas that is brought in by the counterflow vortices. This collision of oppositely directed streams results in the formation of a flow stagnation saddle point and generates high shear rates resulting in a significant turbulence production. As the gas flow approaches the central exhaust it collides with the...
boundary of the BR causing turbulence to increase once again at the interface.

Figure 11a indicates that for a given Re, the swirl ratio $S$ will have a threshold value above which the CR develops (no CR for $S = 5$, Figure 11a). To investigate the operating conditions for the emergence of the counterflow phenomenon, it is instructive to compare the injection gas flow rate in the GVU on the one hand, and the gas entrainment flow rate by the near-wall jets on the other hand. To this end, Figure 14 shows the cumulative gas flow rate calculated from the rear to the front end-wall of the GVU plotted along the unit length, at $r = 0.21$ m for $S = 0, 5,$ and 12. The cumulative gas flow rate at the given radius ($r = 0.21$ m), for each swirl ratio, has been normalized by the injected gas flow rate. For swirl-free flow ($S = 0$), the cumulative gas flow rate shows a linear monotonic growth starting from one end-wall ($z = 0$) to the other ($z = 0.1$ m). The monotonicity of the curve indicates that the gas flows in one direction only: radially inward and without any flow reversal.

When limited swirl is introduced in the flow ($S = 5$), it can be seen from Figure 14 that the formation of the near-wall jet increases the local gas flow rate near the end-wall due to jet entrainment, causing the cumulative gas flow rate profile to increase more sharply than in the absence of swirl ($S = 0$). As more gas flows near the walls of the unit, the local flow rate of the gas in the bulk region decreases (Figure 11a), explaining the central plateau in the $S = 5$ curve in Figure 14.

For higher swirl ratios ($S = 12$), the cumulative gas flow rate curve loses its monotonic behavior and two extrema are formed on both sides of the centerline ($z = 0.05$ m) as seen in Figure 14. The formation of an extremum in the cumulative gas flow rate curve is indicative of the flow reversal resulting in the formation of the CR in the bulk flow through the disc part of the GVU. The near-wall jets entrain the adjacent gas thus causing a steep rise in the cumulative gas flow rate near the end-wall until a maximum is reached. Counterflow in the bulk causes the local radial gas velocities to reverse direction and change sign, such that the local gas flow rates are “subtracted” from the cumulative gas flow rate. This explains

Figure 14. Axial profile of cumulative normalized gas flow rate along $r = 0.21$ m for different swirl ratios: (---) $S = 12$; (-) $S = 5$; full line $S = 0$, calculated by solving the set of Eqs. 1.1–1.5 given in Table 2, under operating conditions ($G_M = 0.4$ kg/s, Re = 13700).

$Q_1$ and $Q_2$ denote the jet entrainment flow rate in the GVU.

Figure 15. In-plane velocity streamlines along the azimuthal plane ($\theta = 20^\circ$) for different swirl ratios at constant Reynolds number (Re = 13700), from numerical simulations calculated by solving the set of Eqs. 1.1–1.5 given in Table 2.
the decrease in the cumulative gas flow rate profile seen for \( S = 12 \) in Figure 14. After crossing the centerline (\( z = 0.05 \) m), the radial gas velocities again change direction on reaching the other boundary of the CR toward the front end-wall, and the second extremum appears in Figure 14. The peak value \( Q_1 = 0.572 \) achieved at around \( z = 0.027 \) m, equals the normalized gas flow rate entrained by the jet located near the rear end-wall, while \( Q_2 = 0.672 \), achieved at \( z = 0.069 \) m, is the normalized gas flow rate entrained by the jet adjacent to the front end-wall. The axial distance in between the two peaks corresponds to the CR where the gas flows radially outward. The sum \( Q_1 + Q_2 \) is larger than one, indicating that the total jet entrainment flow rate exceeds the injection gas flow rate. This physically implies that at high swirl ratios (\( S = 12 \)), the near-wall jets have become so strong that the injected quantity of gas is not sufficient to feed the jets. Gas is sucked back into the jets from the downstream bulk region resulting in the formation of a CR. The above discussion provides a proper physical explanation regarding the origin of the CR in the GVU flow. A counterflow rate \( Q_{ct} \) can be further quantified as \( Q_{ct} = (Q_1 + Q_2 - 1)^*100 \) (= 24.4\%). \( Q_{ct} \) gives a percentage measure of the jet entrainment overshoot over the injection gas flow rate and characterizes the flow rate of the GVU counterflow.

Next, some light is shed on the effect of swirl ratio and Reynolds number values on the formation of these GVU secondary flow features. Figure 15 shows the evolution of flow topology in the GVU with increasing swirl ratio, for a constant Re number of 13700. At low values of the swirl ratio (\( S = 5 \)) the gas streamlines become packed near the two end-walls indicating the formation of near-wall jets. The swirl strength is not yet sufficient to form a CR in the bulk flow of GVU. At \( S = 7.5 \), two counter-rotating vortices are seen to develop in the GVU disc part close to the rear end-wall. The counter-rotating vortices become more elongated and shift toward the circumferential wall, penetrating further into the GVU disc part as the degree of swirl increases, as observed for the \( S = 10 \) case. This shift of the CR from the exhaust region toward the circumferential wall of the unit has also been experimentally reported in previous literature and helps strengthen the prediction from the simulations.\(^{23}\)

For \( S = 12.5 \), the CR further grows and occupies nearly half of the GVU disc part volume. The flow topology is identical to that for \( S = 10 \), but becomes more complicated at \( S = 17 \), where four vortices are observed in the CR of the GVU. Nevertheless, all vortices constitute a united recirculation domain in the GVU disc part separating the through-flow of the

Figure 16. In-plane velocity streamlines along the azimuthal plane (\( \theta = 20^\circ \)) for different Reynolds numbers at constant swirl ratio \( S = 12.5 \), from numerical simulations calculated by solving the set of Eqs. 1.1–1.5 given in Table 2.
injected gas into two major branches: one adjacent to the rear end-wall and another adjacent the front end-wall.

At \( S = 27 \), Figure 15 shows that the overall size of the CR has reached saturation, being bounded: in the axial direction by the two near-wall jets, in the radial direction by the incoming gas flow near the circumferential wall and by the BR near the exhaust. Larger swirl ratios further increase the number of vortices within the CR. The plurality of vortices and saddle points indicates the intense mixing within the CR which can be beneficial for applications in combustion and chemical vortex devices.

The variation of the GVU flow topology with respect to Re is shown in Figure 16. For all studied cases, the swirl ratio is set at a value of 12.5. At small Re, no counterflow occurs (Figure 16, at Re = 3.5). The flow is viscous in nature. As Re increases (\( Re = 7 \)), the streamlines downstream of the injection slots in the bulk region of the disc part of the GVU seem to shift toward the end walls, with an increased packing of streamlines near the two end-walls. This shift of the streamlines suggests entrainment of the bulk gas by the near-wall jets and is a precursor for the emergence of the CR at \( Re = 10.5 \). As Re further increases (\( Re = 14.5 \)), the counterflow expands radially inward Additionally a small recirculation region at the GVU central axis develops. The small recirculation is a precursor for the BR. As Re increases (\( Re = 28 \)) the BR develops by the swirl decay mechanism, and extends over the entire exhaust line. The vortices in the CR become separated by a through-flow branch (\( Re = 700 \)). Beyond this Re however, the flow topology saturates with respect to Re, as can be seen by comparing the in-plane streamlines for \( Re = 700 \) and 13000. As Re is usually high in technological applications, \( Re = 13000 \) is the range in focus in this study where the flow topology becomes independent with respect to Re, and in turn the gas flow rate through the GVU.

**Conclusions**

This article studies secondary flow phenomena arising due to swirling motion of gas in a vortex unit with the help of experimentally validated numerical simulations. Both the formation of a backflow region in the central exhaust and a counterflow region in the bulk flow in the disc part of the unit are investigated in detail. The backflow region develops due to the swirl decay mechanism along the exhaust line of the unit. As the gas proceeds through the exhaust line, it loses its azimuthal momentum. An axial pressure gradient is established along the exhaust line due to this difference in swirl structure and causes gas to flow back in the unit. The counterflow region in the disc part of the unit appears due to the strengthening of the near-wall jets observed in the unit. The near-wall jets form as a result of the breakdown of the cyclostrophic balance between the radial pressure gradient and the centrifugal force exerted on the gas elements. As the near-wall jets get stronger they entrain the adjacent gas from the bulk causing a local flow reversal. This constitutes the counterflow region. A first ever experimental validation of the presence of a counterflow region using Stereoscopic Particle Image Velocimetry is performed to further validate the applied numerical model. To highlight the effect of swirl on the flow topology, first a swirl-free fast flow is numerically explored. It is shown that the flow acceleration in the GVU disc part causes the injected turbulent flow to get laminarized downstream of the unit without the formation of a counterflow or backflow region. The evolution of the flow topology in the GVU is explored for (a) increasing swirl ratio \( S \) at a fixed Re number, and (b) increasing Reynolds number at a fixed swirl ratio \( S \). With increasing Re the flow topology shifts from a laminar to a turbulent regime through a short transition zone, and finally becomes viscosity-independent at large Re values (\( Re > 13000 \)), typical of technological applications. With increasing \( S \) at a constant Re number the counterflow region becomes multicellular with multiple vortices appearing. The plurality of vortices and related saddle stagnation points of the motion improve mixing of flow ingredients, which is beneficial for a variety of technological applications.

**Acknowledgments**

This work was supported by the European Research Council FP7/2007–2013/ERC grant agreement n° 290793. We are grateful for the use of the STEVIN Supercomputer Infrastructure at Ghent University, funded by Ghent University, the Flemish Supercomputer Center (VSC), the Research Foundation - Flanders (FWO) and the Flemish Government—Department EWI. Pieter Reyniers and Pieter Verhees are acknowledged for editorial support.

**Notation**

\[ A_i \] = circumferential area at injection, \( m^2 \)  
\[ D_k \] = GVU final exhaust diameter, \( m \)  
\[ D_{Em} \] = GVU exhaust diameter at the front wall, \( m \)  
\[ D_k \] = GVU circumferential wall diameter, \( m \)  
\[ G_{2x} \] = gas flow rate, \( kg/s \)  
\[ I_0 \] = slot width, \( m \)  
\[ I_{turbulent} \] = turbulence intensity, -  
\[ k \] = turbulent kinetic energy, \( m^2/s^2 \)  
\[ L_k \] = GVU length, \( m \)  
\[ N \] = number of injection slots, -  
\[ P \] = gas pressure, Pa  
\[ P_{gauge} \] = gauge gas pressure, Pa  
\[ P_{scaled} \] = scaled gauge gas pressure  
\[ Q \] = volumetric gas flow rate, \( m^3/s \)  
\[ Q_{normalized} \] = cumulative normalized gas flow rate, -  
\[ r \] = radial coordinate, \( m \)  
\[ Re \] = Reynolds number, -  
\[ S \] = swirl ratio, -  
\[ U \] = mean gas velocity, \( m/s \)  
\[ U_r \] = superficial radial gas velocity, \( m/s \)  
\[ U_{r,s} \] = superficial axial gas velocity at the injection slots, \( m/s \)  
\[ u' \] = fluctuating gas velocity component, \( m/s \)  
\[ u[0]' \] = Reynolds stress, \( m^2/s^2 \)  
\[ x \] = coordinate, \( m \)  
\[ z \] = axial coordinate, \( m \)

**Greek letters**

\( \gamma \) = injection slot angle, deg  
\( \varepsilon \) = turbulence dissipation rate, \( m^2/s^3 \)  
\( \mu \) = gas viscosity, Pa s  
\( \nu \) = kinematic viscosity, \( m^2/s \)  
\( \rho \) = gas density, \( kg/m^3 \)  
\( \theta \) = angular coordinate, rad

**Literature Cited**

1. Ragsdale RG. NASA Research on the hydrodynamics of the gas vortex reactor, NP-9150 NSA-15-001058. Washington, DC, 1959.
2. Priestman GH. A study of vortex throttles part 1: experimental proceedings of the institution of mechanical engineers, Part c. J Mech Eng Sci. 1987;201(5):331–336.
3. Loftus PJ, Stickler DB, Diehl RC. A confined vortex scrubber for fine particulate removal from flue gases. Environ Prog. 1992;11(1): 27–32.
4. Cheng JC, Olsen MG, Fox RO. A microscale multi-inlet vortex nanoprecipitation reactor: turbulence measurement and simulation. Appl Phys Lett. 2009;94(20):204104.

5. Liu Z, Passalacqua A, Olsen MG, Fox RO, Hill JC. Dynamic delayed eddy simulation of a multi-inlet vortex reactor. AIChE J. 2016;62(7):2570–2578.

6. Liu Z, Ramezani M, Fox RO, Hill JC, Olsen MG. Flow characteristics in a scaled-up multi-inlet vortex nanoprecipitation reactor. Ind Eng Chem Res. 2015;54(16):4512–4525.

7. Parker D, Birch MJ, Francis J. Computational fluid dynamic studies of vortex amplifier design for the nuclear industry—I. Steady-state conditions. J Fluids Eng. 2011;133(4):041103.

8. Volchkov EP, Dvornikov NA, Lukashov VV, Borodulya VA, Teplitskii YS, Pitsukha EA. Study of swirling gas-dispersed flows in vortex chambers of various structures in the presence and absence of combustion. J Eng Phys Thermophys. 2012;85(4):856–866.

9. Schütz S, Gorbach G, Piesche M. Modeling fluid behavior and droplet interactions during liquid–liquid separation in hydrocyclones. Chem Eng Sci. 2009;64(18):3935–3952.

10. Pandare A, Ranade VV. Flow in vortex diodes. Chem Eng Res Des. 2015;102:274–285.

11. Council AR. Preliminary Results of Low Speed Wind Tunnel Tests on a Gothic Wing of Aspect Ratio 1.0: H.M. Stationery Office; 1960.

12. Shtern V. Counterflows: Paradoxical Fluid Mechanics Phenomena. UK: Cambridge University Press, 2012.

13. Claypole TC, Syred N. The effect of swirl burner aerodynamics on NOx formation. Symp Int Combust. 1981;18(1):81–89.

14. Escudier MP, Bornstein J, Maxworthy T. The dynamics of confined vortices. Proc R Soc Lond A Math Phys Sci. 1982;382(1783):335–360.

15. Benjamin TB. Theory of the vortex breakdown phenomenon. J Fluid Mech. 1962;4(4):593–629.

16. Hall MG. Vortex breakdown. Annu Rev Fluid Mech. 1972;4(1):195–218.

17. Leibovich S. Vortex stability and breakdown - Survey and extension. AIAA J. 1984;22(9):1192–1206.

18. Olendraru C, Sellier A. Absolute/convective instabilities in the batchelor vortex: viscous case. Paper presented in: Seventh European Turbulence Conference. France: Saint-Jean Cap Ferrat, 1998.

19. Herrada MA, Shtern VN, Torregrosa MM. The instability nature of the Vogel–Escudier flow. J Fluid Mech. 2015;766:590–610.

20. Kendall JMJ. Experimental study of a compressible viscous vortex, JPL-TR-32–290. NASA, 1962.

21. Donaldson CduP, Williamson GG. An experimental study of turbulence in a driven vortex. Aerodynamic Research Associates of Princeton Incorporated, AD-0609460, 1964.

22. Savino JM, Keshok EG. Experimental profiles of velocity components and radial pressure distributions in a vortex contained in a short cylindrical chamber, TN D-3072, NASA, 1965.

23. Wormley DN. An analytical model for the incompressible flow in short vortex chambers. J Basic Eng. 1969;91(2):264–272.

24. Yan L, Vatistas GH, Lin S. Experimental studies on turbulence kinetic energy in confined vortex flows. J Therm Sci. 2000;9(1):10–22.

25. Kwok CCK, Thinh ND, Lin S. An investigation of confined vortex flow phenomena. J Basic Eng. 1972;94(3):689–696.

26. Hombeck RW. Viscous flow in a short cylindrical vortex chamber with a finite swirl ratio: NASA Lewis Research Center, TN-D-5132, 1969.

27. Leschziner MA, Hogg S. Computation of highly swirling confined flow with a Reynolds stress turbulence model. AIAA J. 1989;27(1):57–63.

28. Singh A, Vyas BD, Powle US. Investigations on inward flow between two stationary parallel disks. Int J Heat Fluid Flow. 1999;20(4):395–401.

29. Niyogi K, Torregrosa MM, Pantzali MN, Heynderickx GJ, Marin GB, Shtern VN. On near-wall jets in a disc-like gas vortex unit. AIChE J. 2016;63:1740–1756.

30. Vatistas GH, Fayed M, Soroardy JU. Strongly swirling turbulent sink flow between two stationary disks. J Propulsion Power. 2008;24(2):395–301.

31. Patankar SV. Numerical Heat Transfer and Fluid Flow. New York: McGraw-Hill, 1980.

32. Batchelor GK. An Introduction to Fluid Dynamics. UK: Cambridge University Press, 2000.

33. Volchkov EP, Terekhov VI, Kaidanik AN, Yadykin AN. Aerodynamics and heat and mass transfer of fluidized particle beds in vortex chambers. Heat Transf Eng. 1993;14(3):36–47.

34. Birch DM, Martine N. Tracer particle momentum effects in vortex flows. J Fluid Mech. 2010;723:665–691.

Manuscript received Aug. 30, 2017, and revision received Nov. 28, 2017.