Numerical simulation of a turbulent flow in annular jet for different diameter ratio using the Reynolds Stress model

Ali Feraoun¹, Habib Merouane²

¹Departement de génie mécanique, Université de Mascara, Algérie, ali_feraoun@outlook.fr
²Departement de génie mécanique, Université de Mascara, Algérie, merhabib@gmail.com

ABSTRACT

In present study, a detailed investigation of an annular jet of different diameter ratio has been reported numerically. We present in this paper the influence of the diameter ratio on the initial region of an annular jet. Three different diameter ratios are investigated: the smaller diameter ratio ($r = 0.701$), intermediate diameter ratio ($r = 0.82$) and the high diameter ratio ($r = 0.905$). The numerical simulation was performed by making use of the commercial CFD “Fluent” code which discretizes the solution domain into quadrilateral elements and use a numerical finite volume method coupled with a multigrid resolution scheme. In this research the applications of the Reynolds stress model (RSM) for prediction of a turbulent flow in annular jet are described. The flow governing equations are solved by using a performed SIMPLE algorithm. The results of predicted axial velocity profiles are compared with the experimental data. The computations indicated that the results predicted by RSM model are in good agreement with the experiments.

Key Words: annular jet, turbulent flow, Reynolds stress model, bluff body

NOMENCLATURE

Symbols:

- $\varepsilon$: Turbulent energy dissipation rate, m².s⁻³
- $\mu_T$: Turbulent dynamic viscosity, kg. m⁻².s⁻¹
- $\delta_{ij}$: Kronecker symbol

Indices / Exponents:

- $S$: Surface
- $D_i$: Inner diameter
- $D_o$: Outer diameter
- $r$: Diameter ratio
- $X$: Axial location
- $Y$: Radial location

1. INTRODUCTION

Annular jet flows are encountered in many fields of practical interest, in combustion (burners, bluff bodies...) or in industrial treatment processes. In the case of turbulent flames, the existence of a zone dominated by intense recirculation, omnipresent in such flows, increases the mixing intensity and stabilizes the flame. Stability is a fundamental issue in gas turbine combustion, where extremely lean fuel-air mixtures are adopted for achieving low NOₓ emissions [1]. The geometry of the annular jet is determined by the blockage ratio $D_i/D_o$, where $D_i$ represents the internal diameter and $D_o$ the external diameter. Ko and Chan [2] have divided the annular jet into
three zones, which they called the initial merging, the intermediate and the fully merged zone. In the first one called initial zone, a recirculation zone appears downstream the central obstacle because of a lack of any air supply which creates a pressure gradient in this area. The stationary surrounding air is carried away by the jet from the nozzle and a shear appears. This shear gives rise to instabilities which form periodic vortices. At the boundary of the recirculation zone, there is the stagnation point located on the jet axis. This point is characterized by a longitudinal velocity equal to zero and a transversal velocity nil in average. The position of the stagnation point is independent of the velocity of the injected air but depends on the nozzle geometry.

Many experimental and numerical studies of the non-reacting time-averaged flow field in the recirculation zone of turbulent annular jets have appeared in the literature [3-6]. The experimentalists observed that the reverse mass flow rate increases with increasing blockage ratio, while the recirculation length decreases with increasing blockage ratio due to the decreasing ratio of inertia to pressure force. Moreover, an increased forebody angle (from the cylinder to the disk) leads to outward oriented streamlines and therefore to a longer and wider recirculation bubble, having again an increased reverse mass-flow rate. Behind the bluff-bodies high values of turbulent intensity were measured, which rapidly decay more downstream in the established flow region.

Nakazano et al. [7] studied the annular jets with diameter ratio ranging between 0.2 and 0.8. They use two loudspeakers, placed inside the nozzle, in a transversal way of the flow. The used frequencies correspond to inherent frequencies and they observe that they can influence the recirculation zone and decrease its length by exciting the flow.

Durão and Whitelaw [8] did LDA measurements on unconfined annular air jets. Disks of different sizes were used to study the effect of the different blockage ratios (0.20, 0.39, 0.50) for flows at Re=3,500-28,300.

The mean values of the velocities and their rms. were collected. The mean velocity measurements indicated that the reverse mass-flow rate increases with increasing blockage ratio, while the recirculation length decreases with increasing blockage ratio due to the decreasing ratio of inertia to pressure force. The recirculation length becomes independent from a high Reynolds number.

DeITaglia et al. [9] investigated the behaviour of an annular air jet with a large blockage ratio and found that the flow inside the recirculation zone is asymmetric. This was also observed by Pinho and Whitelaw [10].

Mc Guirk et al [11] Akselvoll and Moin [12] studied the annular jet combined with a central jet. It has been found that the ratio of recirculation decreases with increasing blockage ratio, the turbulence intensity in the recirculation zone has been found to be particularly high (at least 30% of the local mean velocity and strongly anisotropic, becoming isotropic further downstream, after the reattachment point. At the axial stagnation point, the radial stresses are larger than the axial stresses. This is due to the attenuation of the axial normal stresses in that region. Indeed, the production of axial normal stresses is a negative quantity at the stagnation point, since the axial velocity gradient is positive. In the recirculation zone, the measured shear stresses do not vanish in the locations of zero velocity gradient.

In contrast to the experimental works, the number of publications on annular jet simulations is very limited. The reason may be the complexity of the simulation of these flows, which commonly involve recirculation and large shear stresses.

Pope and Whitelaw [13] performed steady two-dimensional simulations of the wake behind an annular jet of blockage ratio 0.50. The k-ε model and two versions of a second moment closure turbulence model were used. The recirculation length were underpredicted and the axial velocity on the centerline downstream the stagnation point was lower than the measured values. The authors indicated the necessity of extending the domain of computations upstream of the bluff-body so that more accurate boundary conditions could be applied. Moreover,
they suggested to use second order closure turbulence models for capturing the different values of the Reynolds stresses.

Percin at al. [14] studied the spatial and temporal characteristics of the three-dimensional swirling annular jet flow using time-resolved tomographic particle image velocimetry technique. The measurements were performed in two modes: (1) a relatively low frequency double-frame mode to increase the measurement time and achieve a converged statistical analysis; (2) a high frequency single-frame mode to enable visualization of the time-series phenomenon.

Ryzhenkov and Mullyadzhanov. [15] studied a turbulent annular jet at the Reynolds number of 8900 Using high-order spectral element method and large eddy simulations, based on the bulk velocity in the annular channel and its half-width. It is shown that the main eigenmodes with low azimuthal wavenumbers rotate around the symmetry axis in both directions providing the explanation for the precessing motion of the recirculating zone.

M. Vanierschot at al. [16] studied the flow structures of an annular swirling jet flow using timeresolved tomographic PIV measurements. For turbulent swirling jet flow, 3D visualization of the vortical structures in the flow field has revealed a central vortex core along the central axis, which breaks up into a double helix.

Ryzhenkov and Mullyadzhanov. [17] studied a direct numerical simulations (DNS) of the turbulent annular jet flow with three different values of the inner-to-outer diameter ratio \( a = d / D = 0.3, 0.5, 0.7 \). The time-averaged velocity fields and fluctuations were analyzed. The low aspect ratio results in the shortest recirculation zone while for high a value the stagnation point features high level of turbulent kinetic energy.

M. Vanierschot at al. [18] studied the flow dynamics of an annular swirling jet flow in the transitional regime at \( Re=520 \) using three different swirl numbers \( Sw=0.38, 0.47 \) and 0.57. A vortex breakdown bubble appears/disappears periodically at a Strouhal number \( St=0.023 \). If the swirl is increased to \( Sw=0.47 \), this breakdown bubble is permanently present in the flow field. A large-scale vortical structure, the precessing vortex core, appears in the flow field and the strength of the PVC fluctuates randomly in time. The PVC precesses at a Strouhal number \( St=0.23 \) around the central axis of the jet. It was observed that the precessing frequency of this spiral vortex core decreases to \( St=0.146 \) as the swirl is increased to \( Sw=0.57 \).

In the present paper RSM model is used to study annular jet flows with different internal diameter and external diameter with different blockage ratio \( r=D_i / D_o = 0.701, 0.82, 0.905 \). The time-averaged Navier-Stokes equations are solved numerically by a control-volume finite-difference method together with the turbulence model equations. The governing equations were numerically solved using the commercial computational fluid dynamics (CFD) solver FLUENT. The SIMPLE algorithm described by Patankar [24] is utilized for pressure-velocity de-coupling and iteration. The under-relaxation iterative TDMA line-by-line sweeping technique is used for solving the resultant finite-difference equations. In obtaining the numerical solution the program GAMBIT was used to generate the computational mesh. Results were obtained with several different grids to determine that the grid used was fine enough to allow adequate details of the flow to be predicted with adequate accuracy but that was not so fine as to require excessive computational times.

2. MATHEMATICAL MODEL/EXPERIMENTAL METHOD

The incompressible time average Navier-Stokes equations in Cartesian coordinate system are written in following form:

2.1 Continuity equation.

\[
\frac{\partial u_i}{\partial x_i} = 0
\]
2.2 Momentum equation

\[ \overline{u}_j \frac{\partial (\rho \overline{u}_i)}{\partial x_j} = - \frac{\partial \overline{p}}{\partial x_i} + g_i + \frac{\partial}{\partial x_j} \left( \overline{r}_{ij} - \rho \overline{u}_i' \overline{u}_j' \right) \]  

(2)

Where \( \overline{u}_i \) and \( \overline{u}_j \) denote the time-average velocity in i and j direction, \( \overline{u}_i \) and \( \overline{u}_j \) stand for the fluctuating velocity, \( \overline{p} \) is the time-average pressure, \( \rho \) is the density of the fluid, \( \overline{r}_{ij} \) is the mean viscous stress tensor, defined as \( \overline{r}_{ij} = 2\mu \overline{S}_{ij} = \mu \left( \frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) \), and \( - \rho \overline{u}_i' \overline{u}_j' \) is the Reynolds stress tensor.

2.3 Reynolds stress model (RSM)

The Reynold's stress model (RSM) is a higher level, elaborate turbulence model. It is usually called a Second Order Closure. This modelling approach originates from the work by Launder [20]. In RSM, the eddy viscosity approach has been discarded and the Reynolds stresses are directly computed. The exact Reynolds stress transport equation accounts for the directional effects of the Reynolds stress fields.

The Reynolds stress model involves calculation of the individual Reynolds stresses, \( \overline{u}_i' \overline{u}_j' \), using differential transport equations. The individual Reynolds stresses are then used to obtain closure of the Reynolds-averaged momentum equation.

The exact transport equations for the transport of the Reynolds stresses, \( \overline{u}_i' \overline{u}_j' \), may be written as follows:

\[ \frac{\partial}{\partial x_k} \left( \rho \overline{u}_k \overline{u}_i' \overline{u}_j' \right) = - \frac{\partial}{\partial x_k} \left[ \left( \rho \overline{u}_i' \overline{u}_j' \overline{u}_k' \right) + p \left( \delta_{kj} \overline{u}_i' \overline{u}_k' \right) + \frac{\partial}{\partial x_k} \left( \rho \overline{u}_i' \overline{u}_j' \right) \right] + \frac{\partial}{\partial x_k} \left( \mu \frac{\partial}{\partial x_k} \left( \overline{u}_i' \overline{u}_j' \right) \right) - P_{ij} - G_{ij} + \phi_{ij} - \epsilon_{ij} + S_{ij} \]  

(3)

where:

\( C_{ij} \): is convection term, \( D_{T,ij} \) is turbulent diffusion term, \( D_{L,ij} \) is molecular diffusion term.

\[ P_{ij} = \rho \left( \frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) \]  

(4)

\( P_{ij} \): is the Stress Production.

\[ G_{ij} = \rho \beta \left( g_i \overline{u}_j' \theta + g_j \overline{u}_i' \theta \right) \]  

(5)

\( G_{ij} \): is the Buoyancy Production.

\[ \phi_{ij} = p \left( \frac{\partial \overline{u}_i'}{\partial x_j} + \frac{\partial \overline{u}_j'}{\partial x_i} \right) \]  

(6)

\( \phi_{ij} \): is Pressure Strain.

\[ \epsilon_{ij} = 2\mu \frac{\partial \overline{u}_i'}{\partial x_j} \frac{\partial \overline{u}_j'}{\partial x_k} \]  

(7)

\( \epsilon_{ij} \): is the dissipation rate.
2.3.1 Modeling Turbulent Diffusive Transport

The Turbulent Diffusive Transport $D_{ij}$ can be modeled by the generalized gradient-diffusion model of Lien and Leschziner [21]

$$D_{ij} = \frac{\partial}{\partial x_k} \left( \frac{\mu}{\sigma_k} \frac{\partial u_i}{\partial x_k} \right)$$  \hfill (8)

2.3.2 Modeling the Pressure-Strain Term

The classical approach to modeling $\phi_{ij}$ uses the following decomposition:

$$\phi_{ij} = \phi_{ij,1} + \phi_{ij,2} + \phi_{ij,w}$$  \hfill (9)

Where

$\phi_{ij,1}$ is the slow pressure-strain term, also known as the return-to-isotropy term and is closed using Rotta [22]

$$\phi_{ij,1} = -C_1 \frac{\rho \varepsilon}{k} \left[ u_i' u_j' - \frac{2}{3} \delta_{ij} \right]$$  \hfill (10)

$\phi_{ij,2}$ is the rapid pressure-strain term is closed with the isotropization of production model

The rapid pressure-strain term, $\phi_{ij,2}$, is modeled as

$$\phi_{ij,2} = -C_2 \left[ (P_j + F_j + G_j) - \frac{2}{3} \delta_{ij} (P + G - C) \right]$$  \hfill (11)

$\phi_{ij,w}$ is the wall-reflection term. The term consists of two different terms, which are modeled following by Gibson and Launder [23]:

$$\Phi_{ij}^w = \Phi_{ij,1}^w + \Phi_{ij,2}^w$$  \hfill (12)

$$\Phi_{ij,1}^w = C_1 \frac{\varepsilon}{k} \left( \frac{u_i' u_m n_k n_m}{\delta_{ij}} - \frac{3}{2} \frac{u_i' u_k n_k}{n_j} + \frac{3}{2} \frac{u_j' u_k n_k}{n_i} \right) c_{jk}^{3/2}$$  \hfill (13)

$$\Phi_{ij,2}^w = C_2 \left[ \phi_{km,2} n_k n_m \delta_{ij} - \frac{3}{2} \phi_{ik,2} n_j n_k - \frac{3}{2} \phi_{i(k,2} n_i n_k \right] c_{jk}^{3/2}$$  \hfill (14)

where $C_1' = 0.5$, $C_2' = 0.3$, $n_k$ is the $x_k$ component of the unit normal to the wall, $d$ is the normal distance to the wall, and $C_s = \frac{C_{\mu}^{3/4}}{\kappa}$, where $C_{\mu} = 0.09$ and $\kappa = 0.4187$ is the von Kármán constant.

The above Reynolds stress transport equation contains two new variables of turbulent energy $k$ and turbulent dissipation rate $\varepsilon$, so their transport equations are as follows:

$$\frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \left[ \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] \frac{\partial k}{\partial x_j} \right) + \frac{1}{2} (P_{ii} + G_{ii}) - \rho \varepsilon$$  \hfill (15)

$$\frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left( \left[ \mu + \frac{\mu_t}{\sigma_{\varepsilon}} \frac{\partial \varepsilon}{\partial x_j} \right] \frac{\partial \varepsilon}{\partial x_j} \right) + C_{\varepsilon 1} \frac{1}{2} (P_{ii} + C_{\varepsilon 2} G_{ii}) \frac{\varepsilon}{k} - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k}$$  \hfill (16)

For the incompressible fluid, the buoyancy production term $G_{ij}$ equals to zero and the constants in this model are...
given in table (1):

<table>
<thead>
<tr>
<th>$C_D$</th>
<th>$\sigma_k$</th>
<th>$\sigma_\varepsilon$</th>
<th>$C_l$</th>
<th>$C_{e1}$</th>
<th>$C_{e2}$</th>
<th>$C_\mu$</th>
<th>$C_2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1.3</td>
<td>1.8</td>
<td>1.44</td>
<td>1.92</td>
<td>0.009</td>
<td>0.6</td>
</tr>
</tbody>
</table>

TABLE 1. Constants for the RSM model

2.4 Computational details

(Fig. 1) show a schematic of the computational domain used for the model annular jet investigated by G. Lalizel [19]. The nozzle is geometrically characterized by the outer diameter equal to 53.88 mm and the inner diameter equal to 48.75 mm. The width of the nozzle output is thus equal to 2.565 mm and the diameter ratio is equal to 0.905.

The numerical simulation is performed by using a commercial CFD code which uses the finite-volume method to discretize the equations of fluid flow. The flow governing equations are solved by using the Reynolds stress model with a performed SIMPLE algorithm to treat the pressure terms in the momentum equations. In the simulations, the convergence is defined when the residuals are reduced below $10^{-4}$.

(Fig. 2) shows the computational grid used. The configuration of the jet is considered to be axisymmetric. The definitions of the geometry and mesh generation were performed by using the Gambit mesh with fine and quadrilateral cells. The area is modelled for dimensions of radius 100 mm and 400 mm in length that corresponds to a half cylinder. The mesh consists of 100 nodes in the radial direction and 300 nodes in the axial direction.

![Figure 1. Schematic diagram of the computational domain](image1)

![Figure 2. Schematic of the computational grid](image2)

2.5 Boundary Conditions

There are three faces bounding the calculation domain namely: the inlet boundary, the wall boundary and the outlet boundary. The experimental data of the mean and rms velocity are applied in the inlet conditions. At the outlet, a large stagnation region is attached at the end of the test section so that convective outflow conditions can be applied.

3. RESULTS

(Fig. 3) illustrates the results of the mean axial velocity contours obtained by RMS model. Three different regions can be distinguished: the quiescent surroundings, the core flow of the jet and the small central recirculation. On
the borders of the three regions shear layers exist (the outer between core flow and surroundings and the inner between core flow and the central recirculation). In these shear layers eddies originate from shear layer instabilities of the Kelvin–Helmholtz type. Behind the center body, in the absence of external flow, fluid is entrained from the jet itself forming a standing vortex, called the central recirculation. The central recirculation closes at the stagnation point.

(Fig.4) show the turbulent kinetic energy contours obtained by RMS model, the turbulent kinetic energy is concentrated in the recirculation zone in the central region of the flow near the jet and in the outer shear layer and the turbulent kinetic energy is quickly dissipated further downstream.

(FIGURE 3. Axial mean velocity contour (m/s) FIGURE 4. Turbulent kinetic energy contour (m²/s²))

The results of (Fig. 5) show the profiles of the mean axial velocity obtained by RMS model at different axial locations X/D₀=0.05, 0.2 and 0.5. The velocity profiles is negative near the jet axis at Y/D₀<0.2 due to the backflow of the recirculation. The maximum axial velocity has a peak velocity around 1 at the radial distance Y/D₀=0.47 in location X/D₀=0.05 and the asymptotic decrease slowly further downstream the jet axis. In general, the results obtained numerically by the RSM model are in good agreement with the experimental data.

The results of (Fig. 6) show the profiles of the mean radial velocity obtained numerically by RSM model at different axial locations. In these figures the radial mean velocity is positive near the jet axis at Y/D₀<0.4 in location X/D₀=0.05 and 0.2. The radial mean velocity is negative at Y/D₀<0.5 in location X/D₀=0.5 due to the recirculation. The maximum downward velocity is situated at Y/D₀ = 0, 34 with the value are about - 0, 34 U₀

Good agreement with measurements is observed by RSM model

(Fig.7 and 8) show the results of the axial and radial rms velocity at different axial locations, the rms fluctuations present two oscillations or two peaks formed by the jet at X/D₀=0.05 and 0.2 the first peak it’s caused by the core flow and central recirculation, the second peak it’s caused by the core flow and surroundings co-flow, these oscillations are important for the axial rms velocity and decrease further downstream. In general, the magnitudes of the radial rms fluctuations were smaller than those for the axial fluctuations. The predicted results by RSM model agree satisfactorily with the measurement

(Fig.9 to 14) shows the view of mean axial velocity and turbulent kinetic energy contours for different diameter ratio, different zone of recirculation it is formed due to the bluff-body, this recirculation zone is less important for
the small diameter ratio and became more significant when the diameter ratio increase. The turbulent kinetic energy decreases when the diameter ratio decreases and it is concentrated in the recirculation zone in the central region of the flow and shear layer.

The influence of the inner to outer nozzle diameter ratio on the centerline is demonstrated in (Fig.15) the profiles clearly illustrate the impact of the diameter ratio r on the jet evolution behavior, length and intensity of the inner recirculation region, and flow redevelopment length.

The recirculation length depends only on the blockage ratio, we observe in this figure that the length of the recirculation decreases with blockage ratio. The length of the recirculatio
FIGURE 7. Axial rms velocity at different axial locations

FIGURE 8. Radial rms velocity at different axial locations
FIGURE 9. Axial mean velocity contour (m/s) with $r = 0.905$

FIGURE 10. Axial mean velocity contour (m/s) with $r = 0.82$

FIGURE 11. Axial mean velocity contour (m/s) with $r = 0.701$

FIGURE 12. Turbulent kinetic energy contour ($m^2/s^2$) with $r = 0.905$

FIGURE 13. Turbulent kinetic energy contour ($m^2/s^2$) with $r = 0.82$

FIGURE 14. Turbulent kinetic energy contour ($m^2/s^2$) with $r = 0.701$
4. CONCLUSIONS

The present study investigated the flow characteristics of an annular jet at different diameter ratio. A first model of annular jet with a small diameter ratio is investigated numerically into the ambient air by the Reynolds Stress Model model. The result obtained numerically show good agreement with the experimental. For the small diameter ratio a small circulation zone is observed; for the high diameter ratio the circulation it is very important and the recirculation length depends only on the blockage ratio, we observe that the length of the recirculation increase with diameter ratio. The turbulent kinetic energy decreases when the diameter ratio decreases and it is concentrated in the recirculation zone in the central region of the flow and shear layer.

The turbulence analysis of the flows tested shows that the flow inside the centre recirculation zone is very anisotropic, especially near the jet axis and the results predicted by the Reynolds Stress Model agree satisfactorily with the measurements.

REFERENCES


