Numerical investigation of flow dynamics and scalar transport in a wall-bounded turbulent jet

M Hrebtov¹, A Bazhenov¹ and K Borynyak¹

¹Institute of Thermophysics of SB RAS, 630090, Ac. Lavrentiev ave. 1, Novosibirsk, Russia

E-mail: weexov@ya.ru

Abstract. We present the results of Large Eddy Simulation of a turbulent jet discharging into a confined slot with the wall spacing to jet inlet width ratio = 0.1 and \( \text{Re} = 10^4 \). The jet exhibits a meandering motion accompanied by a formation of checkerboard pattern of large vortices in the mixing layer. A passive scalar transport was simulated with uniform inlet distribution of the scalar. It was found that the fluctuations of spanwise velocity component grow downstream and their maximum location is migrating from the lateral boundaries of the jet (free-shear layers) to the wall-boundary layers. Some evidence of counter-gradient turbulent scalar transport was found in the mixing layers of the jet, which may be attributed to the influence of observed large-scale checkerboard-type vortices.

1. Introduction

The plane turbulent wall-bounded jet is a flow with a lot of important practical applications in industries dealing with heat and mass transfer. It also retains of special importance in theoretical studies of turbulence, as a plane wall jet is a prototypical near-wall flow that is significantly more complex than a simple boundary layer. Its dynamics is an interplay of two wall boundary layers and two free shear layers in orthogonal planes.

The instabilities generated at the wall interact with the main Kelvin-Helmholtz type vortices in the shear layer. For different aspect ratios between the jet width and the distance between the walls the flow develops large-scale meandering oscillations and quasi-two-dimensional checkerboard pattern of vortices resembling von Karman’s vortex street. For large (~10) aspect ratios these features become more pronounced, especially in a turbulent regime for large enough Reynolds numbers (~10⁴).

The processes inside the jet are affected to a large degree by inflow conditions (the shape of the nozzle, harmonic forcing, etc.) and by the geometry of the flow. Such a big influence is caused by the high sensitivity of the shear layer to small flow perturbations that may lead to the change of dominant instability mode. Thus it is necessary to set the inlet conditions that are closest to physical ones.

This meandering behaviour of the flow must affect the turbulent diffusion of passive scalars enhancing the transport of the scalar in spanwise direction [1]. The present work was aimed to investigate this process.

For present numerical simulation we consider a wall-bounded shallow jet flow in a rectangular domain of size \( H \times L \times D \), where \( L = 267H \) and \( D = 200H \). L and D are much larger than H (see Fig. 1). This dimensions of the domain was taken to mimic the experimental setup [2].

The inflow channel is a short duct of the length \( 2\pi H \) (with the cross-section \( B \times H \) where \( B = 9.6H \)). The inlet velocity distribution was taken from a precursor DNS simulation of a fully developed
turbulent duct flow. The Reynolds number of the flow was $10^4$ based on the bulk velocity and the distance between the walls ($H$).

We used LES approach with the dynamic Smagorinsky model and the following equations for the momentum and scalar transport:

$$\frac{\partial \vec{\rho} \vec{u}}{\partial t} + \frac{\partial (\vec{\rho} \vec{u} \vec{u})}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \vec{p}}{\partial x_i} + \frac{\partial}{\partial x_i} \left( \left( \vec{v} + \nu_{sgs} \right) \vec{S}_{ij} \right),$$

$$\frac{\partial \vec{C}}{\partial t} + \frac{\partial (\vec{C} \vec{u})}{\partial x_j} = \frac{\partial}{\partial x_i} \left( \left( \vec{v} + \nu_{sgs} \right) \frac{\partial \vec{C}}{\partial x_i} \right).$$

The Schmidt number in the equation (2) was set to be 0.9, and its turbulent analogue $S_C$ was also set to this value.

We used a classical form of subgrid viscosity

$$\nu_{sgs} = \left( C_s \Delta \right)^2 \left| S_{ij} \right|,$$

where $\Delta$ is a grid spacing and $C_s$ is Smagorinsky coefficient, which should be evaluated dynamically.

![Figure 1](https://example.com/figure1.png)

**Figure 1.** The computational domain for numerical simulation (a); inverse mean streamwise velocity profile through the jet axis (b); instantaneous concentration distribution (c) (note the false coloring) compared to the experiment [2] (d).

The classical dynamic Smagorinsky [3] model computes $C_s$ after applying explicit spatial filtering to the velocity field and then minimizing the difference between the subgrid and residual stress tensors. This procedure may lead to negative turbulent viscosity (i.e., energy backscatter from the subgrid to resolved scales). This in turn was shown to cause a blow up of the solution because the amount of energy which may be transferred from the small scales is unlimited. It is common to use a clamping of Smagorinsky constant, leaving only positive values. But such procedure is an *ad hoc* and non-physical in nature. It was shown that averaging of the Smagorinsky constant over homogeneous directions resolves that issue, but only a few types of flow have suitable homogeneous directions for that averaging. Another way which was adopted in the present study was introduced in [4] and consists in averaging of the residual tensor magnitude over the lagrangian trajectories. The model has shown a good agreement with direct numerical simulations and can be adopted for almost every flow.

2. **Computational domain and numerical scheme**
The numerical grid consisted of $16 \times 10^6$ hexahedral cells with the wall-adjacent grid-cell height $(x^+)$ of about 1.5 in wall units. The grid was refined in the near-wall regions and also in the mixing layer. The grid resolution was chosen based on preliminary simulations with a set of grids with different number of cells.

The momentum and scalar transport equations (1), (2) were discretized using finite volume scheme with second order accuracy in time and space. The modified solver from OpenFoam [5] package was used.

3. Inlet boundary condition

Proper boundary conditions are necessary for complex flow simulations. Currently, there are several ways to generate turbulent initial conditions at the inlet.

First and the simplest way is a random fluctuations generation. This method is based on a white noise generation. Fluctuations are generated without temporal and spatial correlation, so the spectra of the signal will be governed by the grid spacing and the time step of the simulation. Also, continuity constraint might be violated, that will lead to the appearance of numerical artifacts in pressure fields.

The next way is precursor simulations. The main idea of the method is to let the fluctuations to evolve naturally in a separate simulation and then copy them to the inflow. This method provides the best quality inlet conditions but for many practical applications its computational costs may be too high.

To avoid costly preliminary computations we use the third way: the anisotropic divergence-free synthetic turbulence generation at inflow bound, introduced in [6].

The method utilizes the idea of obtaining the velocity from a vector potential field with the prescribed properties. Using the vector potential ensures the divergence-free condition of the inlet field. The second idea of the method is to represent the vector potential field in a form of “blobs” - constructed random objects of the given scales and intensities. The blobs’s characteristics are defined from known components of Reynolds stress tensor at the inlet and the two-point velocity correlations that represent the fluctuations scale. The method was shown to represent the turbulent statistics of the shear flows quite well.
In the present case for the inlet fields generation we used previously obtained distributions of Reynolds stress components and the two-point velocity correlations of the flow in a rectangular duct with the same geometry as the inlet. The mean inlet velocity was set to be unity and the inlet scalar concentration was set uniform with the value of 1.

4. Simulation results

The instantaneous velocity and concentration fields of the wall-bounded jet obtained in the simulation show the expected meandering behavior with a Strouhal number of the order of 0.1 in a good agreement with experimental observations. Largescale checkerboard-type vortex structures appear after \( y/H=50 \). The meandering behavior of the jet was shown \[7\] to be connected with instabilities caused by longitudinal vortices originating from the wall boundary layer interacting with a Kelvin-Helmholtz instabilities in the mixing layer.

The instantaneous scalar concentration distribution obtained in the simulation (Fig. 1) shows the appearance of large billow-like structures associated with Kelvin-Helmholtz instabilities. These structures start to form at \( y/D=2-3 \). The vortex structure lose lateral symmetry and form checkerboard pattern at \( y/H=50 \) where the two mixing layers collide. The comparison of the scalar distribution with experiment \[2\] shows a good agreement, with a similar type of structures observed in both the simulation and experiment.

![Figure 3](image-url) **Figure 3.** Mean velocity fluctuations components in the spanwise (a-c) and streamwise (d) cross-sections.

The longitudinal profile of mean velocity was compared to the high resolution LES of \[7\] and shown a satisfactory agreement. The difference may be due to the too-coarse mesh in the far region of the jet and an insufficient statistics to average over all the low frequency modes of the flow. In present case the averaging time to gain statistics was equal to twice the travelling time of the fluid particle.
over the computational domain. The expected behavior of the mean longitudinal velocity along the center line for plane jets is $V \sim y^{-1/2}$ in the self-similar region, however this regime was not exactly achieved in our simulations, which may be because the domain size was not large enough in longitudinal direction.

4.1 The mean fields
The mean velocity distribution along the streamwise and spanwise cross-sections are shown in Fig. 2. It can be seen that spanwise velocity component has a maximum in the mixing layer region close to the nozzle exit. As shown by [7], this is due to intense lateral vortex stretching of the "corner" vortices when they are exiting the nozzle.

The longitudinally directed vortices shed from the corners of the nozzle starts to interact by inducing the velocity fields that lead to the spreading of the jet in the mid-plane. However the three-dimensional instabilities that start to grow due to this motion lead to the loss of coherence of the motion of these vortices and eventually to their dissipation. This all is reflected by a maximum of the spanwise velocity close to the nozzle exit. The action of the "corner" vortices is reflected in the mean spanwise velocity in lateral cross-section by the appearance of zones with negative (i.e. directed toward the jet axis) velocity. The zones of negative mean spanwise velocity appear in a shape of stripes close to the wall with the width of several $H$.

The mean scalar concentration distribution has in general the behavior similar to the longitudinal velocity but the lateral spreading of the concentration is faster, which may be due to a significant role of turbulent scalar flux generated by a large-scale quasi-two-dimensional vortices.

4.2 Turbulent fluctuation intensities
The cross-sections of turbulent fluctuations are shown in Fig. 3. The longitudinal cross-sections (Fig. 3 d) are in accord with [7], showing strong anisotropy between the different components of turbulent fluctuations. The wall-normal component of the fluctuations has a peak in the near region of the jet, and then decays rapidly. This means that in the far field the flow should show two-dimensional features. However, the effect of the walls should lead to the appearance of longitudinal vortices so the three-dimensional features of the flow should also present. The meandering behaviour of the jet's core leads to growth of spanwise and streamwise fluctuations amplitudes downstream. We extended this observation by looking at the lateral cross-sections of the fluctuations components (Fig. 3 a-c).

The wall-normal ($x$) component of the fluctuations is the highest in the center of the flow which is expected due to the damping effect of the walls. The longitudinal ($y$) component is concentrated mostly in the near-wall region where the shear is maximal. While going downstream the longitudinal near-wall fluctuations become weaker while simultaneously growing in the inner part of the mixing layer. Quite surprising is the behaviour of the spanwise ($z$) component of the fluctuations. At the nozzle exit the most of spanwise fluctuations are concentrated in the mixing layer. When going downstream some fluctuations appear in the near-wall region of the jet. The magnitude of span-wise velocity fluctuations grow in the near-wall region and soon the location of the maximum of fluctuations moves to a thin layer close to the wall. This means that the meandering motion of the flow affects the near-wall longitudinal velocity fluctuations leading to their tilting, thus some of this fluctuations are transferred to a transversal component. The non-linear interactions of vortices that accompany this processes is the subject of future research.

The scalar fluctuations (Fig 2) are mostly affected by a large-scale two-dimensional vortices. This is evident from the scalar fluctuations distribution which does not change much with the distance from the walls. The scalar fluctuations are concentrated in the outer part of the mixing layer, and grow downstream with the growing intensity of the meandering motion of the jet.

4.3 The turbulent fluxes
The large-scale checkerboard-type vortices should affect the mass transfer in spanwise direction quite significantly. To investigate this we calculated the $z$ component of turbulent scalar flux ($<c'w'>$) and $yz$ component of Reynolds stress (Fig. 4). The streamwise distribution of the Reynolds stress shows two maxima, one close to the nozzle where the "corner" vortices play the major role, and the other further
downstream ($y/H=120$) where the meandering motion of the core become the most pronounced. The spanwise cross-section in the near field ($y/H=30$) shows the appearance of negative Reynolds stress in the near-wall region. This may be connected to the effect of "corner" vortices inducing the negative values of span-wise velocity in the same region. However the appearance of zone with negative stress means that turbulence act pushing the longitudinal fluctuations laterally toward the jet axis in the near wall region.

![Figure 4.](image)

**Figure 4.** Spanwise component of turbulent scalar flux (left); and $yz$ component of the Reynolds stress (right) in the lateral and longitudinal cross-sections.

The longitudinal distribution of turbulent scalar flux shows some negative values at the outer part of mixing layer. This is the evidence of counter-gradient transport of the scalar by the turbulent vortices. The distribution of scalar flux in a spanwise cross-section shows that negative values have two symmetric peaks close to both walls and are caused by the effect of longitudinal vortices that recapture some of the scalar concentration at the mid-plane section and advect it toward the walls and back in the direction of jet axis.

5. Conclusion

The confined jet is a complex flow with an interplay of two- and three-dimensional features. Large quasi-two-dimensional checkerboard vortices observed in the flow contain many smaller vortices with wall-parallel directions. The effect of the wall is very significant for that flow. The present study shows that spanwise velocity oscillations grow in the near-wall region as the flow progresses. This is caused by the interaction of wall-induced longitudinal vortices with mixing layer instabilities. The longitudinal vortices act to produce a counter-gradient transport of the passive scalar at the outer boundaries of the mixing layer. Also these vortices counteract lateral diffusion in the near-wall region of the flow. Further research is needed to investigate the nonlinear interaction between the vortices and the mean flow in the near-wall region.

Acknowledgements

This work was supported by Russian Science Foundation (grant №14-19-01685).

References

[1] Rowland J C, Stacey M T, and Dietrich W E 2009 *J. Fluid Mech.* 627 pp. 423-449
[2] Shestakov M V, Dulin V M, Tokarev M P, Sikovsky D P, Markovich D M 2015 *Int. J. Heat Fluid Flow* 51, 335–352
[3] Germano M, Piomelli U, Moin P, Cabot W 1991 *Phys. Fluids A.* 3 (7): 1760–1765
[4] Meneveau C, Lund T and Cabot W 1996 *J. Fluid Mech.* 319, pp. 353-385
[5] Weller H G, Tabor G, Jasak H, Fureby C 1998 *Comp. in Phys.*, 12, 6
[6] Kroger H, and Kornev N 2015 *ICHMT digital library online*. Begel House Inc.
[7] Mullyadzhanov R, Ilyushin B and Hanjalić K 2015 *Int. J. Heat Fluid Flow* 56: 284-289