Impact of numerical method on a side jets formation in a round jet

K Wawrzak and A Boguslawski

1 Czestochowa University of Technology, Faculty of Mechanical Engineering and Computer Science, Al. Armii Krajowej 21, 42-201 Czestochowa, Poland

E-mail:wawrzak@imc.pcz.czest.pl

Abstract. Numerical analysis of a formation of side jets in an externally modulated round jet is presented. The research is performed applying Large Eddy Simulation method and high-order codes based on Cartesian and cylindrical coordinates. The main attention is paid to an impact of numerical approach on the formation of the side jets, their number and localisation. The results obtained suggest that on Cartesian meshes the number and directions of the side jets are dependent on the distribution of the mesh nodes.

1. Introduction

Contemporary Computational Fluid Dynamics (CFD) offers a wide range of numerical methods, which are capable to predict complex phenomena occurring in fluid flows with a very high accuracy. From the point of view of data validation, knowledge on how a numerical approach influences the physics of the flows and how to exclude such an artificial impact seems to be crucial issues. The present work focuses on Large Eddy Simulation (LES) of an externally excited round jet. A particular attention is devoted to the impact of the numerical approach on the physics of a flow and a possible formation of the so-called side jets. The side jets were observed experimentally in a globally unstable and externally modulated round jet, where very strong vortical structures appeared in the near field of the jet as streams released from the jet inclined at some angle to the jet axis [1,2]. Monkewitz et al. [2] claimed that the primary mechanism of the side jets formation was an azimuthal instability of the vortical structures, that caused a deformation of vortices. However, a complete mechanism driving this phenomenon is not fully understood so far and needs further analyses. The phenomenon of the side - jets formation was observed in a recent numerical investigation [3] devoted to the global instability in a variable density round jet. The LES predictions [3], using Cartesian meshes, showed four well visible side - jets oriented along the mesh lines. Whereas, experimental studies [2] demonstrated that the number of side - jets was between two and six and they were randomly oriented and approximately equally spaced in the azimuthal direction. According to [3] the mechanism leading to the formation of side - jet phenomenon was captured accurately and only the localization of the side - jets was forced by the Cartesian mesh since periodic variation in the azimuthal direction is unavoidable. As concluded in [3] the only difference between LES predictions and a real flow was the mechanism introducing the perturbations, namely random (real situation) versus determined by Cartesian meshes. The discrepancies between numerical and experimental observations were a motivation to perform detailed LES analysis consisting in a comparison of the results coming from two different numerical codes, one based on Cartesian meshes and the second on the cylindrical coordinate system.
2. Numerical methods
As mentioned above the LES predictions were performed employing two numerical codes differing in an application of the coordinate system namely Cartesian and cylindrical. The main difference between these numerical approaches is a mesh resolution in the azimuthal direction. The figures 1 and 2 show inlet planes with the nodes distribution for both numerical meshes. It is seen that the application of the cylindrical mesh ensures a uniform mesh resolution in the azimuthal direction.

2.1. Cartesian coordinate system.
The LES calculations in the case of the Cartesian coordinate system were carried out with the use of high-order solution algorithm based on the projection method for pressure-velocity coupling and a half-staggered mesh arrangement described in details in [4]. The time integration was performed by a predictor-corrector (Adams-Bashfort/Adams-Moulton) method and the spatial discretisation was based on 6th order compact difference scheme. The code used in this study was previously verified in isothermal, non-isothermal, exited and reacting jets [3,5,6,7].

2.2. Cylindrical coordinate system.
The application of the cylindrical coordinate system for the Navier–Stokes equations introduces the singularity at the axis which is due to presence of terms involving the term \(1/r\), where \(r\) – radial distance. Moreover, a definition of the computational domain as \((0,2\pi) \times (0, \hat{R})\), where boundary of the domain, forces to specify the boundary conditions at \(r = 0\), even if there is no physical boundary at the axis [8]. In order to avoid these problems the solution algorithm was formulated applying a coordinate transformation from \((0,2\pi) \times (0, \hat{R})\) to \((0, \pi) \times (\hat{R}, \hat{R})\) and the \(r = 0\) was omitted. The time integration was performed using a low storage Runge-Kutta method and the spatial discretisation was based on compact difference scheme for the radial and axial directions and the Fourier spectral method for the azimuthal one.

3. Boundary conditions and computational domain
The investigation presented in the paper is focused on an externally modulated jet. The computational domain is a simple rectangular box \(14D \times 14D \times 18D\) in the case of Cartesian coordinates and the cylinder \(7D \times 18D\) in the case of the cylindrical coordinate system as shown in the figures 3 and 4, where \(D = 2R\) is a jet diameter. In the case of the Cartesian coordinate system the mesh consists of 180 nodes in axial direction, whereas in normal and spanwise directions the number of mesh points is equal to 128. The cylindrical mesh contains \(64 \times 64 \times 180\) nodes in radial, azimuthal and axial directions, respectively. The inlet boundary condition is specified in terms of instantaneous velocity.
profile as $u(x,t) = U_{mean}(x) + u_{turb}(x,t) + u_{forcing}(x,t)$. The velocity fluctuations $u_{turb}(x,t)$ are computed using digital filtering method proposed in [9] in the case of Cartesian code and according to algorithm proposed in [10] for the calculations on cylindrical meshes. Both methods for the fluctuation velocity guarantee the proper turbulent velocity field. The mean velocity profile is described by the hyperbolic tangent function as follows:

$$U_{mean}(x) = \frac{U_0 + U_c}{2} - \frac{U_0 - U_c}{2} \tanh \left( \frac{1}{4} \theta \left( \frac{R}{r} - \frac{r(x)}{R} \right) \right)$$

where: $U_0$ - jet centerline velocity, $U_c$ - co-flow velocity, $R$ - radius of the jet, $\theta$ - momentum thickness. The co-flow $U_c = 0.01U_0$ is added to compensate lack of natural suction and entrainment from the lateral walls. The external excitation is modeled by adding the axial forcing $u_{forcing}(x,t)$ to the streamwise velocity. The axial forcing is defined as:

$$u_{forcing}(x,t) = A \sin \left( 2\pi St \frac{U_0}{D} t \right)$$

where: $A$ - amplitude of the excitation, $St = fD/U_0$ - the Strouhal number of the excitation. The axial velocity is equal to co-flow velocity while the rest of velocity components are set equal to zero at the side boundaries. The pressure is computed by the Neumann conditions $\nabla p \cdot \mathbf{n} = 0$ at the inlet and side boundaries. At the outlet plane velocity components are computed using the convective boundary condition $\partial U_i/\partial t + \partial U_{conv}/\partial n$, where $U_{conv}$ is a convection velocity which is
computed at every time step as mean velocity at the outlet plane. The pressure is assumed constant at the outlet plane.

4. Results
The LES calculations were performed at the Reynolds number $Re = U_0 D / \nu = 3000$. The jet with the boundary layer characterized by the parameter $R / \theta = 20$ is considered. Numerical analysis apart from an influence of the coordinate system is also focused on impact of inlet parameters on the formation of side-jets. The turbulence intensity at the nozzle exit varied in the range $Ti = 0.01%U_0 - 1%U_0$, the amplitude of the excitation is equal to $A = 1%U_0, 5%U_0$ and the Strouhal numbers are varying in the range $St = 0.4 - 0.6$. In figures 5, 6, 7, 8 and 9 the selected computations obtained in Cartesian coordinate system are presented. The results obtained for the cylindrical coordinate system are presented in figure 11 and 12.

**Figure 5.** Iso – surfaces of the axial velocity with contours of axial velocity at cross – sections perpendicular to the jet for $St = 0.4$, $A = 1%U_0$, and turbulence intensities equal to $Ti = 0.01%U_0$ (left), $Ti = 0.1%U_0$ (centre), $Ti = 1%U_0$ (right).

**Figure 6.** Iso – surfaces of the axial velocity with contours of axial velocity at cross – section perpendicular to the jet for $St = 0.4$, $A = 5%U_0$, and turbulence intensities equal to $Ti = 0.01%U_0$ (left), $Ti = 0.1%U_0$ (centre), $Ti = 1%U_0$ (right).
Figures 5 and 6 show iso-surfaces of the axial velocity with the cross-sections perpendicular to the jet for Strouhal number equal to $St = 0.4$ and for all turbulence intensities and amplitudes considered. The formation of the side–jets are very well visible in the case of the lowest turbulence intensity ($Ti = 0.01%U_0$) at the nozzle exit. In this case four side–jets equally spaced in azimuthal direction and located along diagonal of the computational domain are seen (left pictures in figures 5 and 6). An increase of the turbulence intensity at the inlet plane affects the localization and number of side–jets significantly. Only for the amplitude equal to $A = 5%U_0$ and $Ti = 0.1%U_0$ (figure 6 centre) results reveal a similar image to the lowest perturbation (cf left pictures in figures 5 and 6). For the same turbulence intensity ($Ti = 0.1%U_0$) but lower amplitude only two side – jets are observed (figure 5 centre). The results obtained from LES calculations with the $Ti = 1%U_0$ assumed at the nozzle exit, presented in right subfigures in figures 5 and 6, are characterized by insignificant impact of the amplitude of excitation on the side – jets formation. In this case three side – jets nearly equally

![Figure 7](image_url)

**Figure 7.** Iso – surfaces of the axial velocity with contours of axial velocity at cross – sections perpendicular to the jet for $St = 0.5$, $A = 1%U_0$, and turbulence intensities equal to $Ti = 0.01%U_0$ (left), $Ti = 0.1%U_0$ (centre), $Ti = 1%U_0$ (right).

![Figure 8](image_url)

**Figure 8.** Iso – surfaces of the axial velocity with contours of axial velocity at cross – sections perpendicular to the jet for $St = 0.5$, $A = 5%U_0$, and turbulence intensities equal to $Ti = 0.01%U_0$ (left), $Ti = 0.1%U_0$ (centre), $Ti = 1%U_0$ (right).
distributed in the azimuthal direction are obtained. However, even in this case strong dependence on Cartesian mesh is seen. There are one side–jet located along the diagonal of computational domain and two located along mesh orientation. The subsequent LES predictions, for the Strouhal number equal to \( St = 0.5 \), are presented in the figures 7-8. The assumption of the higher Strouhal number leads to a stronger dependence of the side – jets formation phenomenon on the mesh resolution. In this case four side – jets which cover diagonal of computational domain appear for the turbulence intensities equal to \( Ti = 0.01\% U_0 \) and \( Ti = 0.1\% U_0 \). However, in this case, for the lowest turbulence intensity, side–jets are much more pronounced than for the \( St = 0.4 \). They expand in the radial direction at the angle to the jet axis larger than for \( St = 0.4 \) and cover a larger region of the computational domain. The biggest differences are visible in the case of the highest perturbation at the inlet. In this case for both amplitudes of excitations only one side – jet is predicted. The last part of the LES calculations with Cartesian coordinate system are concentrated on the excitation of the jet characterized by \( St = 0.6 \).

**Figure 9.** Iso – surfaces of the axial velocity with contours of axial velocity at cross – sections perpendicular to the jet for \( St = 0.6 \), \( A = 1\% U_0 \), and turbulence intensities equal to \( Ti = 0.01\% U_0 \) (left), \( Ti = 0.1\% U_0 \) (centre), \( Ti = 1\% U_0 \) (right).

**Figure 10.** Iso – surfaces of the axial velocity with contours of axial velocity at cross – sections perpendicular to the jet for \( St = 0.6 \), \( A = 5\% U_0 \), and turbulence intensities equal to \( Ti = 0.01\% U_0 \) (left), \( Ti = 0.1\% U_0 \) (centre), \( Ti = 1\% U_0 \) (right).
The results obtained from these calculations are shown in the figures 9 and 10. The iso-surfaces and contours reveal the axial velocity field analogical to the lower Strouhal number. For the cases with $Ti = 0.01%U_0$ and $Ti = 0.1%U_0$ the round jet flow is characterized by occurrence of four side–jets located at the diagonal of the computational domain again. Only the results with $Ti = 1%U_0$ show weaker dependence of the formation of side–jets on the grid points distribution.

In order to verify the results obtained with the Cartesian coordinates the LES predictions were also performed for cylindrical meshes. In this case, the jet was stimulated with Strouhal number $St = 0.5$ chosen based on the previous analysis concerning the Cartesian mesh resolution whereas the rest of inlet parameters remained unchanged. Figures 11 and 12 present the results of LES calculations for the cylindrical coordinate system. The application of the cylindrical coordinates demonstrates significantly different results. In the case of the lowest turbulence intensity at the nozzle exit the formation of six side–jets is visible.

Figure 11. Iso-surfaces of the axial velocity with contours of axial velocity at cross–sections perpendicular to the jet for $St = 0.5$, $A = 1%U_0$, and turbulence intensities equal to $Ti = 0.01%U_0$ (left), $Ti = 0.1%U_0$ (centre), $Ti = 1%U_0$ (right).

Figure 12. Iso-surfaces of the axial velocity with contours of axial velocity at cross–sections perpendicular to the jet for $St = 0.5$, $A = 5%U_0$, and turbulence intensities equal to $Ti = 0.01%U_0$ (left), $Ti = 0.1%U_0$ (centre), $Ti = 1%U_0$ (right).
In the case with $Ti = 0.1\% U_0$ only one side – jet can be observed while in the case of the highest turbulence intensity no side – jet can be singled out. Moreover, it should be stressed that the side – jets are inclined at a smaller angle to the jet axis, compared to the results obtained for the Cartesian meshes. This seems to be caused by a decreased mesh resolution along the radial direction. It is also worth noting that in the case of the Cartesian mesh the side – jets remained at the same location and were persistent during the simulation. By contrast, in the case of the cylindrical coordinates the side – jet was rather an instantaneous phenomenon changing the orientation. This observation indicates strong dependence of this phenomenon on the mesh resolution. However, the mechanism leading to the formation of side – jets seems to be predicted correctly with both numerical approaches.

5. Conclusions
The investigation presented in the paper was focused on the influence of numerical approach on the formation of side – jets in the externally modulated round jet. The analysis showed strong dependence of the localization and number of side – jets on the mesh resolution. The LES predictions performed with the Cartesian meshes demonstrated formation of four side – jets, in most of the cases, which cover diagonal of the computational domain. This was especially well visible in the case of a very low turbulence intensity at the nozzle exit. Above-mentioned dependence was significantly less important with increasing the perturbation at the inlet. The application of the cylindrical coordinate system excluded the mesh influence and revealed random number and localization of the side – jets. However, the side – jets predicted on the cylindrical meshes were less inclined to the jet axis. Analysis concentrated on influence of inlet parameter on side – jets formation did not reveal significant impact of the amplitude and Strouhal number.

Acknowledgements
This work was supported by and statutory funds BS/PB-1-103-3010/11/P. PL-Grid infrastructure was used to carry out the computations.

References
[1] Raghu S, Lehmann B and Monkewitz P A 1990 Proc. 3rd European Turbulence Conference 221-6
[2] Monkewitz P A, Lehmann B, Barsikow B and Bechert D W 1989 Phys. Fluids A 1 446-8
[3] Boguslawski A, Tyliszczak A and Wawrzak K 2016 Phys. Fluids 28(2) 025108
[4] Tyliszczak A 2014 J. Comp. Phys. 276 438-67
[5] Wawrzak K Boguslawski A and Tyliszczak A 2015 Flow Turbul. Combust. 95 437-59
[6] Tyliszczak A and Geurts BJ 2014 Flow Turbul. Combust. 93 221-47
[7] Rosiak A and Tyliszczak A 2016 Int. J. Hydrogen Energ. 41 9705-17
[8] Constantinescu G S and Lele S K 2002 J. Comp. Phys. 183 165-86
[9] Klein M, Sadiki J and Janicka J 2003 J. Comp. Phys. 186 662-5
[10] Kempf A, Klein M and Janicka J 2005 Flow Turbul. Combust. 74 67-84