International Journal of Heat and Fluid Flow 67 (2017) 1–12

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

journal homepage: www.elsevier.com/locate/ijhff

# Numerical analysis of the impact of exit conditions on low Mach number turbulent jets

Mehmet Onur Cetin[a,*], Matthias Meinke[a,b], Wolfgang Schröder[a,b]

[a] Institute of Aerodynamics, RWTH Aachen University, Wüllnerstr. 5a, Aachen, 52062 Germany
[b] JARA – High-Performance Computing, Forschungszentrum Jülich, 52425 Jülich, Germany

ARTICLE INFO

*Article history:*
Received 23 February 2017
Revised 23 May 2017
Accepted 3 July 2017

*Keywords:*
Large-eddy simulation
Subsonic jet
Inflow forcing
Non-generic nozzle

ABSTRACT

Large-eddy simulations (LES) of axisymmetric turbulent hot jets at a Reynolds number Re = 316,000 and a Mach number M = 0.12 are computed using an unstructured hierarchical Cartesian mesh method. To analyze the impact of exit flow conditions on the turbulent free jet two formulations of the flow distributions at the nozzle exit are considered. First, the flow inside the nozzle is part of the overall nozzle-jet analysis such that the turbulent flow at the nozzle exit is determined by the solution of the full discrete conservation equations. This formulation is denoted free-exit-flow formulation. Second, the nozzle exit distributions which are used as inflow distributions for the free jet analysis are based on the mean flow profiles plus vortex rings. That is, for the latter case the impact of the nozzle geometry on the free jet is not directly taken into account, the nozzle exit flow distribution is imposed. This formulation is denoted imposed-exit-flow formulation. The LES analyses show that the flow field immediately downstream of the nozzle exit is strongly influenced by the flow structures induced by the nozzle geometry. Compared to the imposed-exit-flow case the free-exit-flow possesses an almost 30% higher peak turbulence intensity level which also results in a massively enhanced velocity decay. Moreover, the free-exit-flow case generates a spectral peak at St = 0.56, while no distinctive peak is observed for the imposed-exit-flow formulation.



## 1. Introduction

Turbulent jets are characterized by strongly sheared flow, which generates turbulent eddies and high frequency noise near the jet orifice, and pressure fluctuations of the fully turbulent core of the jet, which determine the low frequency noise in the streamwise direction (Lighthill, 1954). Both noise sources are strictly related to the turbulent flow in the exit section on the one hand, and to the transition from a wall-bounded to a free-shear layer on the other hand. This means that any jet noise analysis is highly impacted by the quality of the flow solution in the nozzle exit region. The investigation of the influence of various nozzle exit flow formulations is the purpose of this study.

To circumvent the challenge of computing the internal nozzle flow and to make the jet analysis more efficient, it is rather common in numerical investigations to use forcing methods to model the exit distributions of the real nozzle flow. Artificial disturbances are introduced into the shear layer of the jet to trigger the growth of the shear layer instabilities. However, it is questionable whether such an approach can reasonably approximate the real flow conditions especially if the real nozzle geometry, i.e., a chevron nozzle or a nozzle with internal components, generates a highly intricate flow structure. That is, the more demanding the flow field, the more difficult the model of the exit conditions.

In numerous studies, the influence of artificially forced inflow conditions on the development of cold jets was analyzed. Freund (2001) numerically investigated the flow and acoustic field of a low Reynolds number cold jet. To model the nozzle exit conditions random disturbances varying with time and angle were introduced in the shear layer of the inflow plane. The DNS computations showed an excellent agreement with the experiments conducted by Stromberg et al. (1980). Bogey and Bailly (2005) numerically studied the impact of vortex ring forcing. They varied several forcing parameters, e.g., the number of modes, force coefficients etc. They found that the inflow forcing using higher modes reduces the peak magnitude of the turbulence intensity. Moreover, the turbulence anisotropy, i.e., the ratio of the peak magnitude of the turbulent axial and radial velocity components downstream of the potential core, differs when the number of modes is changed in the inflow section. This is important since the turbulent scales characterize the jet development. Therefore, it was stated that the structure of the inflow forcing remarkably influences the jet devel-

* Corresponding author.
*E-mail address:* o.cetin@aia.rwth-aachen.de (M.O. Cetin).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2017.07.003
0142-727X/

opment. Furthermore, Bogey and Bailly (2010) numerically examined the effect of the flow state at the nozzle exit of initially laminar, isothermal jets on the flow and the resulting acoustic field. They showed that the shear layer development is strongly influenced by the variation of the initial momentum thickness of the jet, i.e., at decreasing momentum thickness the shear layer development is enhanced. Decreasing the momentum thickness and introducing random noise at the inflow cause a more rapid growth of the turbulence intensity along the lip line. Speth and Gaitonde (2015) numerically investigated the influence of the boundary layer thickness at the nozzle exit on the flow field of supersonic cold jets at a Reynolds number of $Re = 1.1 \cdot 10^6$. They observed that a jet with thick boundary layer thickness reveals a longer core length and reduces the turbulence intensity compared to a thin boundary layer thickness jet.

Further jet studies focused on hot flow conditions since the fluid temperature is another significant parameter not only for the flow field but also for the acoustic field. Bodony and Lele (2005) numerically explored jets at heated and unheated flow conditions using large-eddy simulation without including the nozzle geometry in the computational domain. Their results exhibited a smaller potential core length when the jet was heated at a constant velocity. Furthermore, the heated jet revealed approx. 20% higher peak turbulence intensity than the cold jet. Koh et al. (2010) numerically investigated single and coaxial jets at hot and cold flow conditions to determine the impact of the temperature gradient on the flow and the acoustic field. They observed that small scale turbulent structures are enhanced by the pronounced temperature gradients. That is, isotropic turbulence is promoted by the temperature gradients. Bogey et al. (2009) simulated a coaxial hot jet emanating from a two-concentric-pipe nozzle at a Reynolds number of $Re = 1.3 \cdot 10^6$ using 14 million cells. Artificial perturbations were introduced at the inflow plane to seed the turbulence. Overall, the results showed a fairly good agreement with the experiments.

The discussion of the findings using prescribed inflow conditions shows that convincing jet analyses can be performed for exit distributions of wall-bounded shear layers that can be described by a fully developed turbulent flow profile. This condition is, however, no longer satisfied when pronounced perturbations are generated by elements in the nozzle interior. Such configurations are encountered when non-generic nozzle and/or jet flows are to be investigated. Xia et al. (2012) numerically analyzed the impact of several complex nozzle geometries, e.g., serrated nozzles, coaxial nozzles with pylon, and the impact of wing flaps on the jet development. They stated that the intricacy of the geometry should be taken into account to perform realistic engine simulations. The impact of the internal nozzle geometry on the jet structure was discussed in some contributions in the literature. Recent efforts focused on the influence of the inner nozzle geometry, i.e., fan flow deflectors such as wedges, vanes etc., on the exhaust plume (Papamoschou and Liu, 2008; Xiong et al., 2010). It was found that introducing nozzle built-in components alters the jet development by increasing the turbulent mixing in the jet near field. Dippold et al. (2009), for example, investigated the offset stream nozzle concept. That is, the flow field was redirected by introducing built-in components inside the nozzle where they created multiple-shear-layer flow and modified the turbulent flow field which resulted in an earlier potential core collapse. Cetin et al. (2016) numerically analyzed the impact of the nozzle built-in components on the jet flow development. Their analyses for the various nozzle configurations, i.e., a clean nozzle, a centerbody nozzle, and a centerbody-plus-strut nozzle showed a similar flow development and spectral distributions in the outer free-shear layer. However, they had a dramatic difference in the inner flow region. In other words, for the outer free-shear layer development the influence of the flow condition inside the nozzle is less pronounced.

As discussed above, the impact of the various inflow conditions on the jet development was analyzed in detail in the literature. However, those studies, which showed convincing jet results, mostly focused on single-shear layer flows. A study which analyzes the influence of various inflow conditions in a multiple shear layer problem, e.g., an inner free-shear and an outer wall-bounded shear layer, is still missing. While the attached boundary layer of a nozzle flow can be convincingly approximated as an inflow condition for a subsequent jet computation, the formulation of a free-shear layer inflow distribution is more challenging since the free-shear flow possesses a higher degree of freedom. To study such a multiple shear layer problem a centerbody nozzle, which produces a free-shear layer emanating from the centerbody plus an outer wall-bounded-shear layer along the nozzle lip, is considered.

In this study, the impact of the exit flow distribution on the near-field jet structures is discussed for turbulent jets that are simulated by highly resolved large-eddy simulations (LES). The Reynolds number is $\mathrm{Re} = 316{,}000$ and the Mach number is $\mathrm{M} = 0.12$. In the first setup - denoted free-exit-flow (FEF) formulation - the analysis of the flow inside the nozzle belongs to the complete nozzle-jet flow computation. That is, the nozzle exit flow is determined by the LES solution of the conservation equations. In the second setup - denoted imposed-exit-flow (IEF) formulation - the nozzle exit distribution is given as an inflow profile, i.e., the flow inside the nozzle is not part of the computational domain. The imposed inflow profile is based on the mean flow distribution of the entire nozzle-jet solution plus additional perturbations that are related to ring-like vorticity distributions.

The study possesses the following structure. First, the numerical method to determine the flow field is briefly explained. Then, the flow problems and the different jet exit flow conditions are described. Subsequently, a thorough discussion of the flow field is given. Finally, a conclusion and an outlook are given.

## 2. Numerical method

To determine the flow field, an unstructured Cartesian finite-volume cut-cell method is used to solve the Navier-Stokes equations for compressible, unsteady, and turbulent flows. The large-eddy simulation is based on the monotone integrated large-eddy simulation (MILES) approach (Boris et al., 1992). That is, the truncation error of the numerical scheme mimics the dissipation of the unresolved subgrid scales.

The convective fluxes of the governing equations are formulated by a low dissipation version of the advection upstream splitting method (AUSM) (Meinke et al., 2002). The cell center gradients are computed using a second-order accurate least-squares reconstruction scheme (Hartmann et al., 2011) such that the overall spatial approximation is second-order accurate. An explicit second-order 5-stage Runge–Kutta method is used for the temporal integration of the conservation equations (Meinke et al., 2002).

A Cartesian unstructured fully parallel mesh generator with hierarchical mesh refinement is used for the grid generation (Lintermann et al., 2014). In the vicinity of the boundaries, the equidistant cells are reshaped into cut cells (Hartmann et al., 2011), where fully conservative boundary conditions are applied. Small cut-cells are treated using an interpolation and flux-redistribution method developed by Schneiders et al. (2013). Further details of the numerical methods, i.e., the discretization and the computation of the viscid and inviscid fluxes, are described by Meinke et al. (2002) and Hartmann et al. (2011). This solution method has been validated for several internal and external flow problems such as