International Journal of Heat and Fluid Flow 50 (2014) 240–253

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Large eddy simulation of a coaxial jet with a synthetic turbulent inlet

Alex W. Abboud *, Sean T. Smith

Institute for Clean and Secure Energy, Department of Chemical Engineering, University of Utah, 155 S 1452 E, room 350, Salt Lake City, UT 84112, USA

ARTICLE INFO

Article history:
Received 26 August 2013
Received in revised form 25 July 2014
Accepted 18 August 2014
Available online 6 September 2014

Keywords:
LES
CFD
Coaxial jet
Turbulence
Turbulent inlet conditions

ABSTRACT

This study analyzed the effects of two methods of synthetic inlet turbulence on the accuracy of coaxial, circular jet simulations with regards to experimental data. The two methods that were utilized in the study were a digital filter method and a synthetic eddy method. This study examines the implementation of these methods into an academic LES code, with extensive evaluation of the simulation data compared to experimental results. The results in the paper are presented for two different mesh resolutions, 22 and 8 million cells. The simulations were performed for three different velocity ratios of the annular to the inner jet: 0.0, 1.0 and 1.5. The inlet methods were utilized in two ways, one by setting the velocity fluctuation profile based on experimental data of the inlet flow, the second by setting an estimated isotropic-fluctuation profile. In this way, using the estimated method can be compared to using the prescribed fluctuations, which may not be usable if experimental data of the inlet is not available.

When comparing the centerline profiles to the velocity data for the turbulent inlet, the data for velocity ratios 1.0 and 1.5 were improved compared to the experimental profiles. However, for the velocity ratio 0.0 case velocity data when using the turbulent inlet the data showed significant decay and did not match experimental profiles in the far field of the jet. The radial data for a velocity ratio of 0.0 showed better agreement with the experimental data in the jet spreading to develop the proper velocity profile. The velocity fluctuation data was the most accurate in the near field region when the turbulent inlet was used. A comparison can also be made between using a prescribed fluctuation inlet and just using the approximated inlet condition. In the near field region using the prescribed fluctuations from data is better, as is expected. However, in some cases the approximation can perform just as well in the fluctuation profile in the far field region. For velocity ratios of 0.0 and 1.0, simply using an approximation for the inlet with isotropic fluctuations performed almost as well as the prescribed fluctuation method in the far-field region. At a velocity ratio of 1.5 this improvement was not present, possibly due to the higher Reynolds number in the annular region.

1. Introduction

In computational fluid dynamics (CFD), poor selection of the boundary conditions can have significant effect on the outcome of the solution. For outlet boundary conditions, these effects can often be mitigated by placing the outlet sufficiently far from the field of interest. The inlet of the domain then becomes the main boundary condition to be prescribed. In the context of Reynolds-averaged Navier Stokes (RANS) simulation, this inlet can often be set as a stationary velocity along with prescribed length scale and turbulent kinetic energy profiles. A channel-flow inlet, in RANS, lends itself most often to a power law profile for this stationary velocity. However, the inaccuracy of RANS in many types of flows has led, in recent years, to an increased use of large-eddy simulation (LES). For LES, setting a fully developed power-law profile with constant velocity is not sufficient to characterize the properties of the turbulent flow. This insufficiency is due to an absence of fluctuations in the resolved velocity. One method to approximate fully developed inlet flow in LES is to use a periodic boundary condition, but this is limited to simple geometries such as constant-cross-section channels. For other cases, inlet flow conditions need to be investigated.

Several different types of turbulent inlet conditions have been demonstrated in the technical literature, a naive approach consists of superimposing time-dependent random data to a stationary velocity profile. However, this approach yields poor results since the uncorrelated fluctuations are quickly dissipated (Tabor and Baba-Ahmadi, 2010). The inlet condition in LES must be prescribed in a way to produce physically realistic velocity structures. Coherence of structures downstream from the inlet have been shown in

* Corresponding author. Tel.: +1 720 320 1457.
E-mail addresses: awabboud@yahoo.com (A.W. Abboud), sean.t.smith@utah.edu (S.T. Smith).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.08.007
0142-727X/

two categories of inlet turbulence: synthetically generated turbulence (Jarrin et al., 2006; Lund et al., 1998; Montomoli and Eastwood, 2011; Klein et al., 2003a); and generation of data with a precursor simulation (Wang et al., 2004; Baba-Ahmadi and Tabor, 2009; Schluter et al., 2004). Alternative implementations for inlet conditions can be found in the review by Tabor and Baba-Ahmadi (2010). Tabor and Baba-Ahmadi (2010) acknowledges that synthetic methods can be inherently less accurate as they only provide turbulent-like properties to the inlet. Synthetic inlet methods do provide an easy way to specify the desired parameters of turbulence, and are quick to generate. Precursor simulation methods generate true turbulent structures, and are thus inherently more accurate than synthetic methods. While using data from precursor simulations can be simple to implement, it may be argued that this approach is undesirable for large-scale codes. A precursor simulation may be undesirable for problems with complex geometry. In addition, the computational cost associated with a large-scale precursor simulation for an LES inlet is often not justifiable. Due to these arguments, the choice of implementation (in our specific academic codebase) is a synthesized turbulent inlet.

The studies by Pedel et al. have been performed with the same research LES code used in this study (Pedel et al., 2012, 2013, 2014). This LES code, known as ARCHES, is a finite-volume, low-Mach package that is constrained to an equally spaced cartesian grid. It has been optimized for highly parallel computations of applied combustion problems. The papers by Pedel et al. compared the results of LES simulations for circular coaxial jets with the experimental results of Budilarto (2003). The focus of those studies was on the particle–fluid interactions. While the simulations performed reasonably well, there were some discrepancies between the experimental data and the results in the fluid flow. Those simulations were run using a stationary velocity inlet condition. This study introduces to the ARCHES codebase an implementation of a generalized turbulent inlet condition for use in arbitrary geometries and inlets. Furthermore, this study applies the synthesized turbulent inlet to the conditions of the previous simulations in order to provide more accurate fluid-phase results in relation to the experimental data from Budilarto (2003).

The research presented here focuses on external jets. These types of flow fields are important in many combustion applications – notably flares, burners and boilers. Velocity fields in turbulent free-flow jets have been studied for several decades, allowing for a good definition of the characteristics (Wygnanski and Fiedler, 1969; Champagne and Wygnanski, 1971; Chigier and Beer, 1964). These characteristics include features such as a core region in the central jet (where the velocity decay occurs slowly), the entrainment of an annular jet decreasing the centerline velocity, and the downstream development of a self-preserving profile. Early LES studies have examined the flow of these single circular jets (Olsson and Fuchs, 1996; Askselvoll and Moin, 1996). Hassel et al. (2006) and Tkatchenko et al. (2007) have run simulations using both RANS and LES codes for comparison between the methods in a coaxial jet mixer. Another LES study showing how varying the Reynolds number can affect the growth of the shear layer in the near field region has been performed in (Kim and Choi, 2009). Zhdanov et al. (2006) has experimentally measured confined coaxial jets to study the recirculation zone development. Jung et al. (2004) has carried out experiments for single circular turbulent jets at very high Reynolds numbers. Comparisons between experimental data and RANS simulations of other reacting flows in a confined coaxial jet have also been studied (Chorny and Zhdanov, 2012). Kornev et al. (2008) has studied the development of scalar fields in experiments with weakly confined coaxial jets.

Recent data has also been produced with significantly more information in the velocities and fluctuations for a single circular jet and under two strong coflow conditions (Budilarto, 2003). In addition to experimental data in recent years many CFD results of these types of flows have been produced, which give better insight into the fine structures of these flows (Apte et al., 2003; Pedel et al., 2012, 2013, 2014).

The experiment used for comparison in this paper does not utilize a swirl generator, however we point out significant developments in this area. Experimental results in weak coaxial jets have been produced with an impeller to generate a swirl condition in Petersson et al. (2000). Other inlet methods in LES have been developed based on using a swirl condition at the inlet first used in (Pierce and Moin, 1998) and shown to work well in coaxial flow of combustion (Apte et al., 2003), and in combustion reactors to mix the air and fuel streams (Pierce and Moin, 2004). Another development of a swirl inlet condition was created by (Baba-Ahmadi and Tabor, 2008) to remap data from the domain to the inlet.

This paper examines the effects of using the synthesized turbulence inlet condition with comparison against experimentally measured flow statistics of a circular coaxial jet. For reference, comparison is made to a stationary velocity inlet. A few additional configurational options were investigated with results being summarized in the text. Another study (Keating et al., 2004), which is similar to the current work, compared three inlet conditions for channel flow simulations. The size of the domain in that paper was much smaller than that presented here. Two of the methods presented in the Keating paper are based on the approach of using prior results over a periodic domain. Again, these methods are usually not desirable for a large-scale simulation or complex geometry. The Keating paper also presents one method based on the generation of turbulence with a near-wall momentum source term. In contrast, the simulation presented in the current study is a free-flow jet with little wall interaction. The Keating paper only compares a few different simulation results and neglects any inclusion of experiment data.

Only two inlet types are examined in this study: the digital filter method suggested by Klein et al. (2003a) and the synthetic eddy method of Jarrin et al. (2006). The digital filter generation method was used for a few major reasons. First, it is easy to adapt for arbitrary geometry inlets, second the simple algorithm does not require much adaptation to fit within our specific LES codebase. In addition, the relative computational cost of this method with a pre-generated table is small. The digital filter method was first shown to work well in DNS plane jets (Klein et al., 2003b), and has been demonstrated to closely match DNS data of a plane jet when utilized in LES (di Mare et al., 2006). Veloudis et al. (2007) has also shown a variation of the method to work well in channel flow for LES. The synthetic eddy method has been shown to work well in channel flows (Jarrin et al., 2006, 2008, 2009). With a small adaption the synthetic eddy method can be constructed with the same advantages that the digital filter method has.

This paper first reviews the theory for the two inlet methods that were utilized, and describes the experimental data set that is used for comparison. The numerical simulation and numerical details are next described. These details are followed by a comparison of the numerical and experimental data sets over the mean velocities and fluctuations. The comparisons are given both on the centerline and at downstream positions radially for three different flow configurations with three different inlet conditions.

2. Theory and experimental data set

2.1. Digital filter inlet

The creation of a synthetic turbulent inlet requires the recreation of the flow statistics. The digital filter method presented by