International Journal of Heat and Fluid Flow 000 (2016) 1–8

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Effect of turbulent model closure and type of inlet boundary condition on a Large Eddy Simulation of a non-reacting jet with co-flow stream

Raul Payri*, J. Javier López, Pedro Martí-Aldaraví, Jhoan S. Giraldo

CMT - Motores Térmicos, Universitat Politècnica de València, Edificio 6D, 46022 Valencia, Spain

ARTICLE INFO

Article history:
Received 9 September 2014
Revised 1 June 2016
Accepted 30 June 2016
Available online xxx

Keywords:
Large Eddy Simulations
Inlet boundary
Smagorinsky
One Equation Eddy
OpenFoam

ABSTRACT

In this paper, the behavior and turbulence structure of a non-reacting jet with a co-flow stream is described by means of Large Eddy Simulations (LES) carried out with the computational tool OpenFoam. In order to study the influence of the sub-grid scale (SGS) model on the main flow statistics, Smagorinsky (SMAG) and One Equation Eddy (OEE) approaches are used to model the smallest scales involved in the turbulence of the jet. The impact of cell size and turbulent inlet boundary condition in resulting velocity profiles is analyzed as well. Four different tasks have been performed to accomplish these objectives. Firstly, the simulation of a turbulent pipe, which is necessary to generate and map coherent turbulence structure into the inlet of the non-reacting jet domain. Secondly, a structured mesh based on hexahedrons has been built for the jet and its co-flow. The third task consists on performing four different simulations. In those, mapping statistics from the turbulent pipe is compared with the use of fluctuating inlet boundary condition available in OpenFoam; OEE and SMAG approaches are contrasted; and the effect of changing cell size is investigated. Finally, as forth task, the obtained results are compared with experimental data. As main conclusions of this comparison, it has been proved that the fluctuating boundary condition requires much less computational cost, but some inaccuracies were found close to the nozzle. Also, both SGS models are capable to simulate this kind of jets with a co-flow stream with exactitude.

1. Introduction

Nowadays, research in combustion is linked to applications that can provide alternatives to reduce emissions and increase process efficiencies. Taking advantage of the gases produced by combustion is a good way to achieve those targets. Recirculating gas combustion products have shown to be useful in order to reduce NO*x* emissions by diluting the mixture and thus controlling temperature levels (Domingo et al., 2008). Flame stabilization is improved as well as NO*x* emissions due to the thermal energy carried by these gases, which act as the enthalpy source needed for ignition (Bourlioux et al., 2000; Domingo et al., 2006). Cabra et al. (Cabra et al., 2002, 2005), in their proposal on lifted flames with a co-flow based on combustion products seems to be a successful implementation in order to study flame stabilization by burnt gases. Due to the large experimental database, besides the sensitivity of the flame characteristics to operating conditions, this flame configuration has gained particular interest in the computational combustion community, and is frequently used for validation and development of combustion models (Gordon et al., 2007). The studies on Large Eddy Simulations (LES) in this kind of flame have been reported in literature (Ihme and See, 2009; Jones and Navarro-Martinez, 2007; Navarro-Martinez and Kronenburg, 2009; Rigopoulos and Navarro-Martinez, 2009; Stankovic and Merci, 2011, 2013; Vervisch and Trouvé, 1998), most of them focusing on Smagorinsky turbulence model closure. LES simulations are not common on these flames due to the cost of implementing detailed chemistry and the inaccuracy of infinitely fast chemistry approaches to simulate lifted flames (Navarro-Martinez and Kronenburg, 2009). Avoiding this problem and considering that the study of turbulent flows in inert environments turns out to be a key point to understand the fuel–air mixing process, some works make an effort to study several applications that involve non-reacting turbulent flows (Banaeizadeh et al., 2013; Kim et al., 2012; Yang and Kær, 2012). Inert studies are of great importance in many industrial processes which include combustion systems, such as rocket engines, gas turbines, industrial furnaces and internal combustion engines (Lucchini et al., 2011). The inert study of this flame helps to focus only on the problem of turbulence, which is one of the

* Corresponding author. Tel.: +34963877650; fax: +34 963877659.
E-mail addresses: rpayri@mot.upv.es (R. Payri), jolosan3@mot.upv.es (J.J. López), pedmar15@mot.upv.es (P. Martí-Aldaraví), jghi@posgrado.upv.es (J.S. Giraldo).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2016.06.016
0142-727X/

Nomenclature

τ_{ij}	sub-grid scale stress tensor (m^2/s^2)
ν_t	turbulent viscosity (m^2/s)
S	rate of strain tensor (1/s)
C_s	coefficient for SMAG model closure
C_k, C_ϵ	coefficients for OEE model closure
R_{xx}	spacial autocorrelation
x	axial distance from the nozzle (mm)
r	radial distance (mm)
D	nozzle exit diameter (mm)
U_0	jet velocity at the nozzle exit (m/s)
U	axial velocity (m/s)
U_m	maximum value of U (m/s)
U_{m0}	maximum value of U at the nozzle (m/s)
u'	axial velocity fluctuation (m/s)
v'	radial velocity fluctuation (m/s)
$u'v'$	Reynolds shear stress (m^2/s^2)
Re	Reynolds number
OEE	One Equation Eddy model
$SMAG$	Smagorinsky model
_nbc	referred to simulations performed with the mapping strategy
_ti	referred to simulations performed with the fluctuating boundary condition
_c	referred to simulations performed with the coarse mesh
_r	referred to simulations performed with the refined mesh
$r_{1/2}$	radial distance at which the excess velocity is half of the value of U_m (mm)

most influential phenomena in combustion. Turbulence increases the mixing process and enhances combustion (Peters, 2000). Inert calculations are the first step before simulating reactive cases.

This paper carries out LES on a non-reacting jet with a co-flow stream that emulates an inert Cabra's experiment considering two different ways of turbulence modeling closure, Smagorinsky (SMAG) and One Equation Eddy (OEE). A turbulent pipe is simulated in order to map its fields in the non-reacting jet domain. The results gathered by this strategy are contrasted with resulting velocity profiles from the simulation using a fluctuating inlet boundary condition. Also, the impact of the cell size is analyzed. Since turbulence is a chaotic phenomenon the solution of two LES calculations should be different. Nonetheless, its velocity statistics, e.g. perturbation velocity root mean square, can be comparable (Kempf, 2007). The simulations are also compared with experimental data.

2. Description of the study

The burner consists of a round fuel jet issuing into a co-flow of H_2 combustion products. The vitiated stream is obtained from hydrogen/air lean premixed combustion and it is composed of H_2O and air (Cabra et al., 2002). The central jet mixture consist of 30% H_2 and 70% N_2, by volume. The bulk velocity of the fuel jet and of the co-flow velocity are of the order of 100 m/s and 5 m/s, respectively. Table 1 summarizes the boundary conditions used in this work as well as the boundary conditions used in the experimental work developed by Wu et al. (2006), who studied the turbulence phenomena related with the experiment in non-reacting and reacting conditions. LES results are compared with experimental data from Cao et al. (2005) as well. For simulations, the main flow and the co-flow are considered to be the same specie with the same kinematic viscosity ($2.07 \times 10^{-}5$ m^2s). In order to reach an equivalent Reynolds number of $Re = 18,600$ in the co-flow stream, the velocity is calculated with the aforementioned viscosity and results $U_0 = 1.84$ m/s.

Table 1
General boundary conditions.

	Experimental Wu et al. (2006)		Experimental Cao et al. (2005)		This work	
	Flow	Co-flow	Flow	Co-flow	Flow	Co-flow
Re	31,500	17,300	23,600	18,600	23,600	18,600
U_0 (m/s)	106	1.4	107	3.5	107	1.84
ϕ (mm)	4.57	190	4.57	210	4.57	210

3. Turbulence modeling

The simulations have been performed with the open-source code OpenFoam. The solver for transient incompressible flows resolves Navier–Stokes equations enforced with a merged PISO-SIMPLE algorithm. It is based on an Eulerian formulation. A finite-volume discretization with second-order central schemes for convection and diffusion terms is employed. Temporal discretization is performed with an implicit second order scheme. This solver first sets the boundary conditions, then solves the discretized momentum equation to compute an intermediate velocity field, computes the mass fluxes at cell faces and lastly the pressure equation is solved.

LES decompose the flow variables into resolved and sub-grid scale terms. The resolved scales are calculated by means of the transport equations, meanwhile the sub-grid scales terms are modeled (Piomelli, 1999; Pomraning and Rutland, 2002; Tyliszczak et al., 2014). Both filtered variables and sub-grid scale variables are dependent of the filter size and the impact of the modeling should decrease as the filter size decrease. With the filtering procedure the momentum equation becomes:

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_j \bar{u}_i}{\partial x_j} = -\frac{\partial \bar{P}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \quad (1)$$

where the variable $\bar{P}$ also includes volumetric forces, and the SGS stress tensor is:

$$\tau_{ij} = \overline{u_i u_j} - \overline{u_i}\,\overline{u_j} \quad (2)$$

The SGS tensor cannot be determined by the resolved scales, therefore it has to be modeled (system closure). This work uses two kind of turbulence model closures: the Smagorinsky approach (SMAG) (Smagorinsky, 1963) and the One Equation Eddy approach (OEE) (Pomraning and Rutland, 2002). A brief description of both is given in the following sections.

3.1. Smagorinsky approach (SMAG)

It is an algebraic model (or zero equation model), which means that there is no transport equation required to calculate the turbulent eddy viscosity (Chan et al., 2014). The model obtains the sub-grid stress term as a function of turbulent viscosity and the strain rate.

$$\tau_{ij} - \frac{1}{3}\delta_{ij}\tau_{kk} = -2\nu_t S_{ij} \quad (3)$$

where S_{ij} is the rate-of-strain tensor and ν_t is the turbulent viscosity, both given by:

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \quad (4)$$

$$\nu_t = C_s \Delta^2 \sqrt{2\,\overline{S_{ij}}\,\overline{S_{ji}}} \quad (5)$$