International Journal of Heat and Fluid Flow 46 (2014) 147–157

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Large eddy simulation of multiple impinging jets in hexagonal configuration – Mean flow characteristics

Martin Draksler [a,*], Bojan Ničeno [b], Boštjan Končar [a], Leon Cizelj [a]

[a] Jožef Stefan Institute, Reactor Engineering Division, Ljubljana, Slovenia
[b] Paul Scherrer Institute, Nuclear Energy and Safety, Villigen, Switzerland

ARTICLE INFO

Article history:
Received 8 August 2013
Received in revised form 3 January 2014
Accepted 20 January 2014
Available online 14 February 2014

Keywords:
Large eddy simulation
Multiple impinging jets
Validation study
Mean flow characteristics

ABSTRACT

The highly turbulent flow of 13 air impinging jets in hexagonal arrangement is analyzed numerically by the means of Large Eddy Simulation (LES). All important flow phenomena, *i.e.* the formation of the fountain flow as well as the negative production of normal stresses near target wall are successfully predicted by the simulation. The mean velocity field and turbulent stresses are validated against existing experimental data. Numerical results show good agreement with experiment. Minor discrepancies can be attributed to the inaccurate modeling of the inlet boundary conditions. A thorough comparison between the simulation results and experimental data is presented and discussed.

© 2014 Elsevier Inc. All rights reserved.

1. Introduction

Impinging jets are characterized by high momentum and heat transfer rates and therefore used in many industrial applications, e.g. for paper and textiles drying, for cooling of turbine blades and electronic components, for annealing and tempering of materials (Weigand and Spring, 2011). Since single jet impingement provides high heat transfer rate only in the vicinity of the impingement region, configurations with multiple jets are usually employed to ensure the uniformity of the heat transfer over wider area. Additional interactions between adjacent jets, which occur in the configurations with small separation distances between jets, increase the complexity of physics involved (Geers, 2003; Weigand and Spring, 2011).

The research on impinging jets started in nineteen sixties, mainly with experimental studies of flow and heat transfer characteristics of both, single (Bradshaw and Love, 1961; Gardon and Cobonpue, 1962; Gardon and Akfirat, 1965) and multiple impinging jets (Koopman, 1975; Goldstein and Timmers, 1982). Recently, thanks to the increase in computational power and the improvements of numerical models, Computational Fluid Dynamics (CFD) is gaining importance as one of the basic tools for fundamental research of turbulent flows, including impinging jets. Numerous studies on impinging jets can be found in the literature. The older ones deal mostly with experimental analyses (Livingood and Hrycak, 1973; Martin, 1977; Viskanta, 1993; Han and Goldstein, 2001), while the later ones include also numerical investigations. Zuckerman and Lior (2006) and Weigand and Spring (2011) provided the most recent reviews of the experimental and numerical research on impinging jets, including single and multiple configurations.

The present study is motivated by the research of cooling performance of the divertor, a plasma-facing component for the future fusion power plant (Norajitra et al., 2007). A conceptual design of the divertor has been developed at Karlsruhe Institute of Technology (Kruessmann and Norajitra, 2004), and is based on cooling by multiple high-pressure helium jets. In order to optimize the design and improve the heat removal capability of the divertor, extensive numerical studies were carried out (Končar et al., 2010, 2011, 2013; Simonovski et al., 2010). Pre-conceptual and optimization studies require efficient analyzing technique, and therefore a steady-state Reynolds-Averaged-Navier–Stokes (RANS) approach together with two-equation turbulence model was used for testing of different design options. In RANS modeling only the mean flow characteristics are resolved by governing equations while the turbulence is strictly modeled by a semi-empirical turbulence model. The success of RANS simulation therefore depends on the accuracy of the model used. Models are usually based on assumptions, and therefore it is very important that the assumptions in the model derivation apply to the process under investigation.

The predictive character of different turbulence models in simulations of impinging jets has been investigated by many authors (Craft et al., 1993; Thielen et al., 2003, 2005; Xing et al., 2010; Draksler and Končar, 2011). A thorough survey was given by Polat et al., 1989; Zuckerman and Lior, 2006; Weigand and Spring, 2011.

* Corresponding author. Tel.: +386 1 588 5320.
E-mail address: martin.draksler@ijs.si (M. Draksler).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.01.005
0142-727X/© 2014 Elsevier Inc. All rights reserved.

Nomenclature

Roman

$\boldsymbol{f}$	source term in momentum equation
$\boldsymbol{n}$	unit normal vector for the cell surface
$\mathbf{u}$	velocity vector
C_W	model constant
D	nozzle diameter
g_{ij}	velocity gradient tensor
H/D	nozzle-to-plate distance
H_u^n	convective term of momentum equation in time level n
n	time level
p	pressure
r/D	distance from the center-line of the model in radial direction
s/D	pitch-to-pitch distance
S_{ij}	strain rate tensor
s_{ij}^d	traceless symmetric part of the square of the velocity gradient tensor
t	time
u_i	ith velocity component
V_{cl}	mean axial velocity, evaluated at the center-line of the nozzle exit
x_i	ith Cartesian coordinate
y/D	vertical distance from the target wall
y^+	dimensionless wall distance
Re	Reynolds number

Greek

$\boldsymbol{\tau}$	stress tensor
Δ	cube root of cell volume
δ_{ij}	kronecker delta
Γ	boundary of cell domain
∇	operator for spatial derivative
ν	kinematic viscosity
ν_t	eddy viscosity
Ω	cell volume
ρ	fluid density
τ	separation time
φ	scalar potential (pseudo-pressure)

Other

$\langle \ldots \rangle$	mean value, obtained by time-averaging

Majority of their studies pointed out that present turbulence models fail to accurately simulate both, wall-normal and wall-parallel flow of impinging jets. The reasons are different, for instance eddy-viscosity models assume turbulence isotropy which is far from being true in highly turbulent impinging jets. Even more complex Second Closure Models, *i.e.* anisotropic Reynolds Stress Models, do not provide much better results in the near wall region (Craft et al., 1993; Weigand and Spring, 2011; Zuckerman and Lior, 2006) since they are still based on model constants. Up to the present, the elliptic relaxation model $\overline{v^2} - f$ by Durbin (1991), and its modified version $\zeta - f$ by Hanjalić et al. (2004) provide one of the best results in simulations of impinging jets.

All physical phenomena of turbulent flows can be captured completely only by the Direct Numerical Simulation (DNS) which requires a very fine grid in order to resolve the smallest scales of motion. As such, DNS is usually used for simulations of flows at low and moderate Reynolds numbers (up to 10,000) and simple geometries, e.g. two dimensional slot jets. Though a higher numerical accuracy is achieved by Spectral Methods (Ferziger and Perić, 1996; Tiselj et al., 2001), use of Finite Volume (FV) method prevails in simulations of impinging jets. So far, DNS was used for analysis of single jet impingement, to study the unsteady flow dynamics (Chung et al., 2002), to study the dynamics and formation process of coherent structures (Tsubokura et al., 2003), and for the studies related to the heat transfer process (Hattori and Nagano, 2004; Satake and Kunugi, 1998). Some authors used DNS for analysis of the entrainment process at the turbulent/nonturbulent interface (Silva and Pereira, 2009; Mathew and Basu, 2002; Ooi et al., 1999).

LES seems to be more appealing for modeling of the flow of round jets at high Reynolds numbers since only large spatial scales are solved directly while the small ones are being modeled. Despite the additional sub-grid-scale modeling LES still requires fine mesh in order to properly predict the fluid flow and heat transfer near the solid walls (Hadžiabdić, 2006; Gant, 2010). With careful use, LES can provide very accurate and meaningful results. For instance, Hadžiabdić (2006); Hadžiabdić and Hanjalić (2008) successfully reproduced the dynamics of coherent structures and heat transfer characteristics of single round impinging jet at Reynolds number around 23000. On the other hand, Hällqvist (2006) performed the LES simulation of single jet impingement with the model where more dissipative numerical scheme was used instead of using the explicit sub-grid-scale model. To the best of our knowledge, the only existing LES study of multiple round impinging jets at high Reynolds number has been performed by Kharoua and Khezzar (2011). Their simulation successfully predicted mean flow characteristics although only one quarter of the geometry was simulated, by applying the symmetry boundary condition in the middle of the jets, which is unusual for LES.

In this work, the LES simulation results of the selected test case with 13 highly turbulent air impinging jets in hexagonal arrangement are presented. The experimental case by Geers (2003) and Geers et al. (2004, 2005) has been carefully selected to resemble as far as possible the divertor cooling conditions. A three-dimensional parallel numerical solver PSI-Boil (Parallel SImulator of Boiling phenomena) by Ničeno (2011) is used to carry out the calculations. In addition, an explicit Wall-Adaptive Local Eddy-viscosity (WALE) sub-grid-scale (sgs) model is used to model the sub-grid-scale turbulence (Nicoud and Ducros, 1999).

Main objective of the present paper is to discuss the accuracy of LES simulation for predictions of flow field of multiple impinging jets. Mean flow characteristics, obtained by time-averaging of instantaneous flow field are compared with available experimental data at two characteristic planes.

2. Governing equations and numerical method

An incompressible and three-dimensional gaseous flow is considered for LES simulation of multiple impinging jets. Physical properties of air are assumed to be constant.

The governing equations consist of continuity equation

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

and incompressible momentum equation

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \mathbf{u} + \boldsymbol{f}, \tag{2}$$

Download English Version:

https://daneshyari.com/en/article/655500

Download Persian Version:

https://daneshyari.com/article/655500

Daneshyari.com