Flow and Heat Transfer Characteristics of Multiple Impinging Jets: Large Eddy Simulation

Martin Draksler, Boštjan Končar, Leon Cizelj
Jožef Stefan Institute
Jamova cesta 39
1000 Ljubljana, Slovenia
martin.draksler@ijs.si, bostjan.koncar@ijs.si, leon.cizelj@ijs.si

Bojan Ničeno
Paul Scherrer Institute
5232 Villigen PSI
Switzerland
bojan.niceno@psi.ch

ABSTRACT

The flow and heat transfer characteristics of highly turbulent impinging jets are analyzed numerically by the means of Large Eddy Simulation. Experimental conditions at the nozzle plate (domain inlet) are not known, and therefore sensitivity analysis of the inlet boundary condition has been performed. Three different velocity profiles are tested, namely the flat velocity profile, the fitted profile from the precursor RANS simulation, and the flat velocity profile with additional pseudo-random fluctuations. The mean flow and heat transfer characteristics, obtained by time-averaging of the instantaneous results, are discussed and compared with available experimental data.

1 INTRODUCTION

Jet impingement cooling concept is based on heat removal by fluid jets, usually gaseous and highly turbulent, which impinge on the target surface after being dispensed from the nozzle. Single jet provides high heat transfer rates only in the vicinity of the impingement region, and therefore the configurations with multiple jets are usually employed to ensure the uniform heat transfer over wider area. Despite the fact, that the topic has been the subject of investigations for more than fifty years, the understanding of the involved physical mechanisms is still not complete. Recently, with the development of the sophisticated numerical models together with the increasing computational power, Computational Fluid Dynamics (CFD) is becoming an important tool for the fundamental research of turbulent flows, including impinging jets.

The concept with multiple impinging jets has been proposed also for the cooling of the divertor, a plasma-facing component of the future fusion reactor DEMO [1]. Since the component is in contact with hot plasma, very high heat loads are expected (in order of $10\,MW/m^2$) [2]. A conceptual design of the divertor is the subject of different optimization studies [3, 4, 5, 6], which require fast and efficient analyzing technique, and therefore a steady-state Reynolds-Averaged-Navier-Stokes (RANS) approach together with the two equation turbulence model is a preferable choice. The performance of the proposed cooling finger at the operating conditions has been predicted only by the means of numerical simulations, and therefore the estimation of the potential errors is very important.
All physical phenomena of turbulent impinging jets can be captured completely only by the Direct Numerical Simulation (DNS). The method requires very fine grid in order to resolve the smallest scales of motion, and therefore is not very convenient for simulations of turbulent flows at high Reynolds number. On the other hand, Large Eddy Simulation (LES) seems to be more appealing, since only the larger spatial scales are resolved. With the modelling of the flow at smaller, sub-grid scales, the computational demand is decreased, which enables the method to be used for the simulation of impinging jets at high Reynolds numbers [7, 8].

In order to improve the understanding of the physical phenomena of impinging jets, the LES simulation of the multiple impinging jets in hexagonal configuration has been performed. The experimental test case of Geers [9] has been selected to resemble as far as possible the divertor working conditions. The simulations have been carried out with a three dimensional numerical solver PSI-Boil (Parallel-SImulator of Boiling phenomena) [10]. Subgrid-scale turbulence is modelled by an explicit Wall-Adaptive Local Eddy-viscosity (WALE) subgrid-scale model [11].

The main objective of the study is to evaluate the accuracy of inlet boundary condition. The experimental velocity profile at the nozzle exit is not known, and therefore three different velocity profiles are tested, namely the flat velocity profile (case 1), the profile obtained from precursor RANS simulation (case 2), and the flat profile with additional pseudo-random fluctuations (case 3). The analysis is based on the comparison between time-averaged numerical results and experimental data.

2 GOVERNING EQUATIONS, NUMERICAL MODEL AND SIMULATION DETAILS

The flow is considered to be three-dimensional and incompressible. The properties of air are assumed to be constant. The governing equations consist of incompressible Navier-Stokes equations

\[ \nabla \cdot \mathbf{u} = 0, \]  
(1)

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

and of the heat transfer equation

\[ \frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla)T = \alpha \nabla^2 T + \dot{q}. \]  
(3)

Term \( \mathbf{u} \) is the velocity field, \( t \) is the time, \( \rho \) is the fluid density, \( \nu \) is the kinematic velocity, and \( \mathbf{f} \) is the source term representing external forces. In Eq. (3), \( T \) is the fluid temperature, \( \alpha \) is the thermal diffusivity and \( \dot{q} \) is the source term representing external sources, \textit{i.e.} wall heat flux.

Governing equations are numerically solved by a Finite Volume (FV) method on staggered grid. The time integration of the momentum equation is obtained by a semi-implicit projection method by Gresho [12] together with additive correction multi-grid solver. The detailed description of solving procedure may be found in [13]. Advection term \((\mathbf{u} \cdot \nabla)\mathbf{u}\) in momentum equation and the term \((\mathbf{u} \cdot \nabla)T\) in heat transfer equation are discretized by a second-order, explicit Adams-Bashfort method, while the Crank-Nicolson scheme is used for discretization of the diffusive term \(\nu \nabla^2 \mathbf{u}\) in momentum equation and the term \(\alpha \nabla^2 T\) in heat transfer equation. The linear systems resulting from the discretization of the momentum and heat transfer equation are solved with the Conjugate Gradient (CG) method with the Incomplete Cholesky preconditioner [14]. The size of the simulation time step is based on Courant-Friedrichs-Lewy (CFL) number which is kept below 0.4 in order to prevent the divergence of the solution.
The computational domain consists of a rectangular domain (Fig. 1(a)) which is bounded by two horizontal plates, i.e. nozzle plate at the top and heated surface at the bottom (target plate). The fluid enters the domain through the nozzle plate (inlet), and the jets deflect into radial direction after impinging on the target plate and leave the domain through four vertical boundary planes (outlet). The dimensions of the computational domain are similar to the experimental facility, i.e. $0.3 \text{ m} \times 0.052 \text{ m} \times 0.3 \text{ m}$ [9]. The nozzle diameter $D$ is equal to 0.013 m, the pitch-to-pitch distance $s/D$ is equal to two, and the nozzle-to-plate distance is equal to four diameters. Detailed description of the experiment can be found in [9]. Origin of the coordinate system is located at the geometric center of the target plate, with the y-axis pointing towards the nozzle plate. As such, the $U$ and $W$ denote the wall-parallel velocity components (in $x, z$ directions) while $V$ denotes the axial velocity (in $y$ direction).

The velocity profile and fluid temperature are prescribed at the domain inlet. The nozzle plate is modelled as an adiabatic no-slip wall, while the target plate is modelled as a no-slip wall with external heat load. Domain outlet is modelled by the convective outflow boundary condition [15] which allows re-entering of the fluid flow, if should it occur. The Neumann boundary condition for pressure is used at all boundaries where the velocity is prescribed.

Five different grids, all with the total number of elements equal to power of two, have been tested preliminary in order to assure the grid independent solution. Node clustering in the wall-normal direction is used to obtain the proper grid resolution near the target wall ($y^+ \approx 1.0$), while the node distribution in other two directions is kept uniform. The information about the total number of cells, and number of nodes in each direction is summarized in Table 1. Kuczaj et al. [16] have shown that the required grid resolution for LES should be in order of Taylor micro-scale [17]. Similar analysis shows that grid 4 and 5 fulfill the testing criterion [18]. The results herein are obtained with the grid 4.
Table 1: Information about tested grids. Notation Mega (M) is equal to $2^{10}$.

<table>
<thead>
<tr>
<th># nodes</th>
<th># cells</th>
<th>x</th>
<th>y</th>
<th>z</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid 1</td>
<td>9 M</td>
<td>384</td>
<td>64</td>
<td>384</td>
</tr>
<tr>
<td>Grid 2</td>
<td>18 M</td>
<td>384</td>
<td>128</td>
<td>384</td>
</tr>
<tr>
<td>Grid 3</td>
<td>20 M</td>
<td>768</td>
<td>64</td>
<td>768</td>
</tr>
<tr>
<td>Grid 4</td>
<td>74 M</td>
<td>768</td>
<td>128</td>
<td>768</td>
</tr>
<tr>
<td>Grid 5</td>
<td>600 M</td>
<td>1536</td>
<td>256</td>
<td>1536</td>
</tr>
</tbody>
</table>

2.1 Inlet boundary conditions

The flow through the nozzle plate is not simulated in our study. Therefore the phenomena which occur prior to the nozzle exit must be described by the inlet boundary condition. For instance, it is known that low aspect-ratio orifices produce vena contracta phenomena where the jet becomes more narrow and the fluid is additionally accelerated [9]. Moreover, experimental velocity profile at the nozzle exit is not known, and therefore the sensitivity study with three different inlet velocity profiles at the domain inlet (see Fig. 2) is performed. In the first case (case 1) the inlet velocity profile is simulated with the flat profile, i.e. with the uniform axial velocity $V_{cl}$ equal to 23.88 m/s (based on the Reynolds number at the nozzle [9]). In the second case (case 2) the velocity profile is adopted from the precursor RANS simulation where the flow through the nozzle plate is simulated and vena contracta is observed. In the third case (case 3) the fluctuations with the prescribed length scale ($10\%$ of nozzle diameter $D$) and time scale ($1/(1400\text{Hz})$) are superimposed in axial direction on the flat profile from the case 1. The fluctuations are obtained with the pseudo-random generator by Kraichnan [19].

Figure 2: Tested inlet profiles.

Pseudo-random generator generates the velocity field as follows [19]:

$$u(x, t) = \sum_{n=1}^{N} [v(k_n) \cos (k_n \cdot x + \omega_n t) + w(k_n) \sin (k_n \cdot x + \omega_n t)],$$

where

$$v(k_n) = \zeta_n \times k_n \quad \text{and} \quad w(k_n) = \xi_n \times k_n,$$

which ensures incompressibility

$$k_n \cdot v(k_n) = k_n \cdot w(k_n) = 0.$$
Vectors $\zeta_n$ and $\xi_n$ are picked independently from a three-dimensional Gaussian distribution. It is obvious that the inlet velocity profile affects the flow behavior in the vicinity of the nozzle plate, but more important question is whether the inlet velocity profile influences also the flow characteristics further downstream, near the target wall. The answer can be obtained from the cross-correlation function between pairs of points along the jet’s axis, i.e. between the inlet and the point at the centerline of the jet. The procedure is the following:

In the first step the cross-correlation function $\Phi_{ab}(\tau)$ as a function of time-lag $\tau$ between two considered points $a, b$ is calculated:

$$\Phi_{ab}(\tau) = \frac{\langle a(t)b(t+\tau) \rangle}{\langle a^2(t) \rangle^{1/2} \langle b^2(t+\tau) \rangle^{1/2}}.$$  

(7)

where $a(t)$ and $b(t)$ represent the fluctuating part of the variables from its mean value $\langle u_a(t) \rangle$ and $\langle u_b(t) \rangle$, respectively. Mean value ($\langle \rangle$) is obtained by time-averaging over the time interval equal to 0.5 second of physical time.

In the second step the maximum value $\Phi_{ab,\text{max}}$ of the cross-correlation function $\Phi_{ab}(\tau)$ for each pair of points is identified and presented in graph with respect to the distance between considered points $l/D$.

The results of two simulation cases (case 1 and 3) are shown in Fig. 3. Both of them confirm that the cross-correlation function is decreasing with the increasing distance from the nozzle, i.e. the correlation between the inlet velocity and the local fluid velocity is decreasing as the fluid flows downstream. The correlation is completely lost before the jet reaches the half-height of the channel ($l/D = 2$). Moreover, higher values of cross-correlation function are obtained when the additional fluctuations are imposed at the inlet (case 3) than in the simulation case with only flat velocity profile (case 1).

![Figure 3: Cross-correlation function between the inlet and considered point at the centerline of the central jet.](image)

3 RESULTS

The analysis of the inlet boundary condition is based on the mean flow and heat transfer characteristics which are obtained by time-averaging of instantaneous results over the time interval equal to at least 0.5 second of physical time. Jet needs approximately $2.1 \times 10^{-3}$ second to move from the nozzle plate down to the target wall. Moreover, time-averaged results show rotational symmetry of 60 degrees, and therefore additional averaging is used in order to increase the sample size for statistics.
Profiles of non-dimensional axial velocity $V/V_{cl}$ at different heights above the target plate for both vertical planes are shown in Fig. 4. The axis of the central jet is located at $r/D = 0$ (in both planes), the axis of its closest neighbor at $r/D = 2$ (in plane P-1), and the axis of the most outer one at $r/D = 2\sqrt{3}$ (in plane P-2). In the vicinity of the nozzle plate, at $y/D = 3.5$, all tested profiles predict wider jets as they occur in the experiment, resulting in a slower degradation of the axial velocity in the core of each jet as they move downstream. Moreover, bending of the neighbor and outer jet is causing a radial dislocation of the impingement region. The results show that precursor RANS profile (case 2) cause smaller bending of the outer jet comparing to the other two cases, while on the other hand the shape of the neighbor jet remains almost independent of the inlet profile. The results also show that additionally imposed fluctuations at the inlet do not affect the mean velocity profiles.

Figure 4: Profiles of the mean axial velocity $V/V_{cl}$ at different heights above the target wall in plane P-1 (a) and in plane P-2 (b). Symbol:Experiment, Black: Flat profile, Blue: Profile from precursor RANS simulation, Green: Flat profile + random generator.

In the regions between adjacent jets, the collisions between two or more wall jets (formed after the impingement) lead to the formation of the fountain flow where the reverse flow towards the nozzle plate is formed. In plane P-1 the fountain flow appear in the gap between the central jet and its closer neighbor (Fig. 4(a): at $r/D \approx -1.0$) while a much wider gap in plane P-2 (Fig. 4(b): at $r/D \approx -2.0$) is surrounded by four jets; two jets from plane P-2 (central and outer jet) and additional two neighbor jets (N) from two planes P-1, one from each side (see Fig. 1(b)). Agreement between the simulation results and the experimental data is rather good, except near the target plate where the bending of the jets affects also the formation of the fountain flow.

The profiles of the mean wall-normal $\langle v'v' \rangle$ and shear stresses $\langle u'v' \rangle$ in plane P-1 are shown in Fig. 5. Increased levels of both stresses occur in the circumferential region around each jet (shear layer), i.e. at $r/D = \pm 0.5$ and at $r/D = -2.0 \pm 0.5$, while on the other hand stresses remain rather low within the jets’ core. Beyond the edge of each jet all stresses decrease rapidly. Comparison between cases 1 and 3 shows that additional fluctuations at the inlet (case 3) do not increase the production of stresses further downstream; same levels of stresses occur in both simulation cases. Such behavior could be expected from the correlation function presented in Fig. 3. On the other
hand, the narrower core of each jet in case 2 allows somehow faster penetration of the shear layer towards the jets’ axes as the jets move downstream. Initially smaller wall-normal stresses in the case 2 exceed the values of other two cases farther downstream. The case 2 shows the best agreement with the experiment regarding the the shape of the jets. On the other hand, the production of the shear stresses $<u'v'>$ occur mainly within the shear layer after the nozzle lip, and therefore the shear stresses are in much better agreement with experimental data (see Fig. 5(b)).

![Figure 5: Profiles of the wall-normal stresses $<v'v'>$ /$V_{cl}^2$ (a) and shear stresses $<u'v'>$ /$V_{cl}^2$ (b) at different heights above the impingement plate. Symbol: Experiment, Black: Flat profile, Blue: Profile from precursor RANS simulation, Green: Flat profile + random generator.](image)

Radial distribution of the mean Nusselt number is presented in Fig. 6. Numerical simulations confirm that the highest heat transfer rates occur at the locations where the jets impinge on the target wall.

![Figure 6: Radial distribution of the Nusselt number in plane P-1 (a) and in plane P-2 (b). Symbol: Experiment, Black: Flat profile, Blue: Profile from precursor RANS simulation, Green: Flat profile + random generator.](image)
Comparing to the experiment, approximately 20% higher Nusselt number is predicted in the stagnation region of each jet by the simulation when the flat velocity profile is used (case 1). Similar result is obtained by the simulation case 3. On the other hand, approximately 50% higher Nusselt number than in experiment is obtained by the simulation case 2 due to the higher local velocity and higher stresses in the vicinity of the target wall. Away from the stagnation region, the discrepancy between the simulation results and experimental data is smaller.

4 CONCLUSIONS

Large Eddy Simulation is conducted in order to obtain the flow and heat transfer characteristics of multiple circular turbulent impinging jets at Reynolds number equal to 20000. The study of three inlet velocity profiles, i.e. flat profile with the uniform axial velocity (case 1), fitted velocity profile from the precursor RANS simulation (case 2), and the flat profile with additional axial fluctuations generated with pseudo-random generator (case 3), is based on the comparison between mean (time-averaged) simulation results and experimental data.

Mean velocity and stress profiles show that neither of tested inlet profiles is able to mimic the experimental conditions near the nozzle plate completely. The fitted profile from the precursor RANS simulation (case 2) produces the most similar shape of the jets comparing to the experiment, but with too high axial velocity if the axial velocity is re-scaled in order to keep the desired Reynold number. The size of the jet’s core is the most relevant parameter concerning the mean flow characteristics. The thinner core of the jets, which is formed due to the narrower inlet profile in case 2 (RANS profile) results in the smaller bending of the neighbor and outer jets, i.e. dislocation of the impingement region, as well as in the stronger development of the shear layer, i.e. faster penetration towards the jet axis as the jets move downstream.

Results show that additionally imposed turbulence (case 3) is rapidly dissipated from the flow, and that the levels of the stresses downstream the nozzle do not exceed the values that occur anyway (with the flat profile without additional fluctuations). Similar conclusion can be drawn also from the cross-correlation function between the inlet velocity and the local fluid velocity further downstream which is decreasing rapidly with the increasing distance from the inlet. The correlation between the inlet and local velocity is completely lost before the jet reaches the half-height of the channel.

Approximately 20% higher Nusselt number than in experiment is obtained by the simulation case 1 in the stagnation region of each jet, most probably due to the slightly higher fluid velocity in the vicinity of the target wall. Compared to the case 1 (flat profile), additionally imposed fluctuations at the inlet (case 3) do not affect the heat transfer characteristics since they are rapidly dissipated from the flow. On the other hand, higher fluid velocity near the target wall in the case 2 results in much higher heat transfer rate.

ACKNOWLEDGMENTS

Current work, supported by the European Commission and the Ministry of Education, Science and Sport of the Republic of Slovenia was carried out within the framework of European Fusion Development Agreement (EFDA).

REFERENCES


