Instantaneous Heat Transfer Characteristics of Multiple Impinging Jets

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

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

ABSTRACT

In this paper, the instantaneous flow and heat transfer characteristics of highly turbulent impinging gaseous jets in a hexagonal configuration are numerically studied by a means of large eddy simulation. This study is conducted in order to identify the key physical phenomena that govern efficient heat transfer of impinging jets. Instantaneous flow field is examined, with emphasis on propagation of coherent structures and their influence on instantaneous heat transfer characteristics. Two mechanisms that contribute to enhanced heat transfer have been identified: (i) a convective cooling by the tangential flow near impingement point, and (ii) an intense quasi-periodic quenching by large-scale vortical structures. A discussion is based on a thorough analysis of instantaneous velocity and temperature fields.

1 INTRODUCTION

The cooling with impingement of gaseous jets is one of the most effective single phase heat removal techniques. This is due to the highly unsteady jet dynamics near the target wall that results in intense flapping of jets, i.e. movements of the stagnation/impingement point that broadens the region of intense heat transfer around the geometrical center of the jets. In configurations with multiple jets, additional enhancements in heat transfer are observed in regions where adjacent wall-jets interact, and therefore the use of multiple jets in practice is preferable choice. Simple geometry together with the high heat removal capability are the main attributes that make the jet impingement cooling so attractive for practical applications. Our interest in impinging jets and associated phenomena originates from the research within the optimization studies of the divertor's cooling module [1-3] for the future fusion demonstrator plant DEMO.

With a continuous growth of computational resources, Computational Fluid Dynamics (CFD) has gained recognition as one of the fundamental research tools for analyses of turbulent flows and associated phenomena. As such, it is increasingly adopted also for the research of jet impingement cooling. All physical phenomena of turbulent flows can be captured completely only by the Direct Numerical Simulation (DNS). The method requires a very fine grid and small time step in order to resolve the smallest scales of turbulent motion.
Therefore, present DNS simulations of impinging jets are limited to relative low Reynolds numbers (up to 10000), and relatively simple geometries (slot jets). On the other hand, Large Eddy Simulation (LES) seems to be more appealing since only larger scales are resolved while the smallest ones are being modelled. LES has proven to be a reliable approach for studying the flow and heat transfer characteristics of single jet impingement [4-6]. On the other side, the use of DNS and LES for systems of multiple impinging jets at high Reynolds number is not so frequent. 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 [7]. Their simulation successfully predicts mean flow characteristics although only a quarter of the geometry was simulated, which is unusual for LES.

In this work, the instantaneous heat transfer characteristics of the selected test case with 13 highly turbulent air impinging jets in hexagonal arrangement are studied numerically by the means of LES. A three-dimensional parallel numerical solver PSI-BOIL (Parallel Simulator of BOiling phenomena) [8] is used to carry out the simulations. The sub-grid-scale turbulence is modeled by an explicit Wall-Adaptive Local Eddy viscosity (WALE) model. A validation study that is based on the mean flow characteristics, obtained by time-averaging of instantaneous flow fields shows very good agreement between numerical results and experimental data by Geers [9].

Main objective of the present paper is to identify key physical mechanisms that contribute to efficient heat transfer. Discussion is based on instantaneous flow and temperature fields.

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 independent of temperature and pressure. The governing equations consist of incompressible Navier-Stokes equations

\[ \nabla \cdot \mathbf{u} = 0, \]  
\[ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + f, \]  
and the heat transfer equation

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

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

The governing equations are numerically solved by the Finite Volume (FV) method on staggered grid. Central differencing scheme is used for spatial discretization. The time discretization of the advection terms is obtained with the Adams-Bashfort scheme, while the Crank-Nicolson scheme is used for discretization of the diffusive terms. The time integration of the momentum equation is obtained by a semi-implicit projection method together with the additive correction multi-grid solver (for details see [10]). The linear systems resulting from the discretization of the momentum and heat transfer equation are solved with the Conjugate
Gradient method with the Incomplete Cholesky preconditioner. The size of the 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. This results in time step equal to $5 \times 10^{-7}$ s.

The computational domain (see Figure 1) consists of a box-shaped domain that is bounded by two horizontal planes. The jets enter the domain through the nozzle plate at the top (inlet). After impingement onto the target plate fluid leaves the domain through four vertical planes (outlet). The nozzle diameter $D$ is equal to 0.013 m, the jet-to-jet spacing $s/D$ is equal to two, and the nozzle-to-plate distance $H/D$ is equal to four. 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, $u$ and $w$ denote both wall-parallel velocity components (in $x,z$ directions) while the $v$ stands for axial velocity (in $y$ direction).

![Figure 1: Computational domain with a characteristic plane where numerical results are presented. Plane P-1 intersects geometric axes of the central nozzle and its closest neighbour.](image)

The velocity profile and the fluid temperature are prescribed at the 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 [11] 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.

The numerical grid consists of 576 M cells ($1 \ M = 2^{20}$). Node distribution in both wall parallel directions is kept uniform, while the node clustering in the wall-normal direction is used to obtain the proper grid resolution near the target wall ($y^+ \sim 1$). The results of grid refinement study as well as other details on computational grids can be found in [10].

## 3 RESULTS

The following discussion is based on instantaneous flow and heat transfer characteristics of multiple impinging jets at Reynolds number equal to 20000 and nozzle to plate spacing $H/D = 4$. All flow field variables are expressed in dimensionless form. For instance, $v/V_{e1}$ denotes a normalized axial velocity with respect to the jet exit velocity at the nozzle centerline. Geometry related parameters are expressed in terms of nozzle diameter $D$. 
As such, $x/D$ denotes the distance along the x-axis from the centerline of the model. All data, presented in present work, are extracted at the same time step.

### 3.1 Instantaneous flow characteristics

Jets enter the domain through the nozzle plate at the top of the domain (see Figure 2). Initially sharp interface between each jet and surrounding fluid becomes disturbed with the increasing distance from the nozzle. Exponentially growing disturbances, attributed to the occurrence of the Kelvin-Helmholtz instability, results in formation of a circumferential shear layer. Initially thin shear layers develop as the jets move downstream and start to penetrate towards the axis of each jet. The study of mean flow characteristics [10] showed that the levels of mean wall-parallel stresses $(u'\bar{u}', w'\bar{w}')$ around central jet at $|x/D| \sim 0.5$ are doubled between vertical locations $y/D = 3.5$ and $y/D = 2.5$. Further downstream, a very unstable and unsteady dynamics of jets is observed. Flapping of the jets causes movements of the impingement point, and at certain moments of time jets may completely break up.

Collisions between two or more jets after the impingement lead to the formation of the reverse flow towards the nozzle plate. This fountain flow additionally disturbs incoming jets and promotes the growth of stresses. Smaller levels of stresses are detected at sides where fountain flow is absent. In plane P-1 (see Figure 2), the fountain flow occurs in the gap between the central jet and its closest neighbor. Part of the fluid may re-enter the jet and re-impinge the target wall, while the remaining fluid flows out of the configuration, creating a so-called self-induced cross flow. All these phenomena cause a strong dispersion of the impinging jets in the near wall region, and therefore multiple stagnation points are formed at the same instance of time. As it will be shown later, this complicated impingement pattern is also reflected in heat transfer characteristics.

![Figure 2: Snapshot of the axial velocity field in plane P-1](image)

### 3.2 Identification of coherent structures

Visualizations of virtually all flows, regardless of the flow regime, reveal the evolution of various flow structures of different sizes and shapes. Their formation is attributed to some-kind of instability that evolves in the flow and cause the formation of regions with locally concentrated vorticity. If the instantaneous vorticity is phase-correlated within the structure [12] for a longer time compared to its characteristic time, the structure can be identified as a coherent one. Therefore, it can be distinguished from other phase-independent vortical...
motions, like eddies, for example. In flows of free-shear jets, strong velocity gradients in the shear layer cause the occurrence of the instabilities of the Kelvin-Helmholtz type. The exponential growth of initially small disturbances leads into a roll-up of shear layer that eventually results in the formation of three-dimensional large-scale structures. As these structures propagate downstream, different mechanisms distort their initial shape [12].

Coherent structures mostly develop in turbulent ambient, and therefore an identification method is required in order to distinguish these quasi-periodic vortical structures from the turbulent background. Jeong and Hussain [13] have proposed a $\lambda_2$-criterion, which presumes that occurrence of the coherent structures coincides with the occurrence of the pressure minimum. They have also shown that in some flow types, a pressure minimum can be caused by unsteady straining, and that sometimes a pressure minimum is eliminated by the viscous effects. Therefore, those effects should be excluded from the eduction criteria. Existence of a pressure minimum in a plane requires two positive eigenvalues of the hessian of a pressure ($\nabla^2 p$) [13]. They have shown that hessian of a pressure is equal to $-(S^2 + \Omega^2)$, where $S$ and $\Omega$ are the symmetric and antisymmetric parts of velocity gradient tensor $\nabla u$. Therefore, two negative eigenvalues of the symmetric tensor $(S^2 + \Omega^2)$ are required for existence of pressure minimum. Moreover, if eigenvalues are ordered by size ($\lambda_1 < \lambda_2 < \lambda_3$), identification of the negative $\lambda_2$ is sufficient.

Visualization of isosurfaces of $\lambda_2$ shows that vortical structures are mostly concentrated in the circumferential shear region around each jet. Similar distribution of structures can be observed also in the near wall region. This result is in good agreement with the experimental observation by Geers [14].

![Figure 3: Vortical structures, visualized by isosurfaces of $\lambda_2 = -40$, and coloured by the distance from the target wall. Both vertical planes show the instantaneous axial velocity.](image_url)
3.3 Instantaneous heat transfer characteristics

A typical heat transfer distribution of a hexagonal array is presented in Figure 4. The contours of the Nusselt number (\(Nu\)) show a very heterogeneous pattern, which reflects a very unstable and irregular flow dynamics near the impingement surface. The highest values of instantaneous \(Nu\) are mostly found in the stagnation region of each jet and in the gaps between jets where the secondary stagnation zones are formed by the collisions of adjacent wall-jets.

![Figure 4: Contours of instantaneous Nusselt number. Red line shows the location of the characteristic plane P1.](image)

Figure 5 shows the \(Nu\) profile in parallel to the wall-normal and wall-parallel velocity plots. In addition, the contour plots of instantaneous temperature and axial velocity fields in near wall region are shown, and the coherent structures are visualized (\(\lambda_2 = -15\)). All data are extracted from a single vertical plane along the x-axis (see Figure 4). The axial velocity is shown at \(y/D = 0.005\), while the wall-parallel velocity is extracted from the first near-wall nodes (\(y/D = 10^{-5}\)).

Radial profile of \(Nu\) exhibits approximately ten local maxima between \(x/D = -0.5\) and \(x/D = 3.5\). Those local enhancements in heat transfer can be attributed either to the “impingement mechanism” (visible also in velocity-vector plots) or to the occurrence of the coherent structure in the boundary layer. However, both situations result in an instantaneous
supply of a cold fluid into already heated boundary layer which results in locally enhanced heat transfer.

The peak at $x/D \sim 0.2$ ($Nu \sim 200$) is attributed to the efficient heat transfer by a cold jet that impinges onto the target surface. Radial wall-jet that is formed immediately after deflection from target wall contains already heated boundary layer. This results in a rapid drop of the heat transfer rate, although the wall-jet velocity is high. Similar mechanism is identified at $x/D = 2.4$.

Figure 5: (a) Contours of instantaneous axial velocity field, isolines of $\lambda_2 = -15$; (b) Instantaneous velocity-vector field (coloured by the velocity magnitude), isolines of $\lambda_2 = -15$; (c) Contours of instantaneous temperature field, instantaneous velocity-vector field, isolines of $\lambda_2 = -15$.

At $x/D \sim 0.2$, a peak of Nusselt number ($Nu=250$) coincides with the occurrence of the vortical structure in the near wall region (at $y/D \sim 0.02$). This structure provides a cold air into already heated boundary layer, and additionally accelerates the fluid that is trapped
between the structure and the wall. Both mechanisms enhance the heat transfer. At \( x/D < -0.2 \), a weak vertical impinging jet cannot penetrate into already wide boundary layer. Consequently, a reduction of heat transfer occurs as the fluid temperature continuously rises. Similar mechanism applies to the peaks at \( x/D \sim 0.5, \ 0.8, 1.8 \) and \( x/D \sim 3 \).

Heat transfer characteristics of the secondary stagnation zones are “defined” by the instantaneous flow characteristics of colliding jets. For this instance of time, a peak in Nusselt number at \( x/D \sim 1.5 \) is caused by the occurrence of a vortical structure.

4 CONCLUSIONS

Large eddy simulation was carried out in order to study the instantaneous heat transfer characteristics of multiple gaseous impinging jets at Reynolds number equal to 20000. Discussion starts with the analysis of the instantaneous axial velocity field which shows a very unsteady flow dynamics, especially in the near wall region. Initially well-defined jets become quickly disturbed by the evolving disturbances from the circumferential shear layer. Visualization of flow structures by the \( \lambda_2 \)-criterion reveals that the vortical structures evolve mostly in the shear layer of each jet, which is in good agreement with experimental observations.

The contours of instantaneous Nusselt number show a very heterogeneous pattern which reflects a very unsteady flow dynamics near the target wall. Comparison between the instantaneous Nusselt number distribution and the flow characteristics in the near-wall region reveals two mechanisms that cause local enhancements in heat transfer: (i) a convective cooling by the tangential flow near the impingement point, and (ii) an intense quasi-periodic quenching by large-scale vortical structures.

ACKNOWLEDGMENTS

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

Part of the simulations has been performed on Monte Rosa computational cluster of Centro Svizzero di Calcolo Scientifico (CSCS) which is gratefully acknowledged.

REFERENCES


