Heat transfer characteristic of an impingement cooling system with different nozzle geometry

K Marzec¹ and A Kucaba-Pietal²
¹Rzeszow University of Technology, MTU Aero Engines, Rzeszow, Poland
²Rzeszow University of Technology, Rzeszow, Poland

E-mail: ¹k.marzec@prz.edu.pl, ²anpietal@prz.edu.pl

Abstract. The influence of different geometries of the nozzles of an impingement cooling array of six jets directed to the flat surface on the flow mechanism and the heat transfer were investigated numerically. Basically the setup consisted of a cylindrical plenum with an inline array of impingement jets. Simulation were performed using Computational Fluid Dynamics (CFD) code Ansys CFX. The $k-\omega$ shear stress transport (SST) turbulence model was used in calculations. The physical model was simplified by using the steady state three-dimensional analysis and incompressible and viscous flow of the fluid. The study focused on an usage of different nozzles shapes in the cooling system for constant inlet flow parameters and boundary conditions. The numerical analysis of the different mesh density resulted in good convergence of the GCI index, what excluded mesh size dependency. The obtained results indicate, that the usage of various types of nozzles results in different values of the heat transfer coefficient and the Nusselt number in the affected area.

1. Introduction
Literature includes many investigations of arrays of impingement cooling jets. A variety of geometrical or physical parameters of cooling systems has been studied. The majority of the studies were experimental ones [1-7]. Many numerical simulations of the impingement cooling affecting different types of the jets [8-11] or different CFD turbulence models [12] are presented in the literature. Many studies have been made to analyze the different dimensionless y/D parameter or different Reynolds number values. Many correlations were introduced to calculate the heat transfer coefficient or the Nusselt number in the stagnation region, applicable for the adequate Reynolds number and nozzle height range [12].
An impingement cooling system is an array of jets of high velocity fluid which is made to strike a target surface. An impinging jets can be classified as a submerged jet or a free jet. If the fluid issuing from the jet is of the same density and nature as that of the surrounding fluid then the jet is called the submerged. If the fluid issuing from the nozzle is of a different density and nature than that of the surrounding fluid then the jet is called free. Further classification between jets can be made for confined and un-confined jets. In the case of confined, the jet remains bounded between two surfaces during its flow. There is less entrainment of air from the atmosphere. Un-confined jets are free to expand after they imping on the target surface.

¹ Corresponding author: k.marzec@poczta.onet.pl, phone +(48)-731-706-731
A jet impingement heat transfer cooling systems plays a significant role in many industrial applications. This is due to the fact, that the impinged jet provides higher rate of heat and mass transfer. Such applications as drying of paper, cooling of photovoltaic cells, de-icing of aircraft wings, annealing of metals, heat exchangers in automotive and aeronautical applications, cooling in grinding process or cooling of gas turbine blades and other moving parts of small distances between them, are a few examples of jet impingement cooling.

A flow field is characterized by a jet zone, a stagnation zone and a wall jet zone, as indicated in Figure 1.

2. Numerical Setup

In this section, an overview on the numerical setup investigations is presented. The numerical grid density is taken into consideration.

2.1. Boundary and Initial Conditions

The schematic geometry of the system under consideration is shown in Figure 2. Both: the left hand side and the right hand side of the system is open to allow the fluid to flow. Fluid is free to expand after imping target surface. The velocity of the flow $V=(u,w)$ at the inlet of the supply tube $u=14\text{m/s}$ was prescribed to obtain the Reynolds number $Re=4800$ at the impingement nozzle. Thereby, the Reynolds number was based on the mean velocity at the nozzle and the jet diameter $D$. The air density was $\rho=1.17\text{ kg/m}^3$, thermal conductivity was $k=0.025\text{ W/mK}$ and the dynamic viscosity was $\mu=1.8 \times 10^{-5}\text{ Pas}$. The fluid temperature at the inlet was $T_{jet}=20\text{ C}$. The wall, the jet impinged onto was heated with the heat flux $q_w=5000\text{W/m}^2$. The outlet boundaries of the calculation domain were represented by opening pressure boundary conditions which permitted the fluid to both enter or leave the domain. To simplify the analysis, the steady-state fluid flow is considered and it was assumed that the fluid physical properties are constant and the effect of the gravity and radiation is neglected. The flow field is three-dimensional. The roughness of the tube which contains the flowing fluid is $30 \mu\text{m}$.
2.2. Computational domain

The chosen computational domain corresponds to an array of six cooling impingement jets directed to a flat surface. In this study four different nozzle shapes were taken into consideration (see Figure 3). The nozzle exit-plate distance $y/D=8$ and nozzle pitch-diameter ratio $S/D=8$ were constant for all simulations. For the chamfer and countersunk nozzle type, the smaller diameter $d=0.8\text{mm}$ and the convergence ratio $D/d=2$ were set. The tube inlet diameter was $D_T=5\text{mm}$. The total length and width of the target plate was $L=56\text{mm}$. This ensured that the outlet boundaries were away from the section of interest where the jet impinged onto the target plate. Air was supplied to the system by the supply-tube only from the left side. The right side of the tube was closed.

Figure 2. A schematic 2D diagram of geometry of the considered problem.

Figure 3. Investigated computational domain with different nozzles geometry.
The governing equations for the problem under consideration are based on the balance laws of the mass, the linear momentum, and the thermal energy in the steady state and incompressible and viscous flows.

2.3. Numerical grids and numerical accuracy
The numerical calculations were carried out using unstructured grids with 1.9mln elements generated by the Ansys CFX. The influence of the numerical grid density on the results of the heat transfer coefficient and the Nusselt number in the stagnation region was taken into consideration. Therefore, four analysis with different cell size of the area of a single impingement jet in the area of the interface (between air flow and flat surface) were taken into consideration.

As to investigate the sensitivity of the results of the numerical analysis, the Grid Convergence Index GCI (equation 1) was calculated [13]. The safety factor \( F_s = 3 \), was set for two grids comparison. The chosen parameter was temperature \( T_b \) measured in the distance of 0.025mm from the surface in the \( y \) direction, appropriately for each grids. Order of convergence was \( p=2, r_p \) was the density factor. For the analysis, number 3 GCI index equal of 0.6% was obtained. Therefore, it might be concluded that numerical results on the fine grid are grid independent.

\[
GCI = F_s \left( \frac{T_{h2} - T_{h1}}{T_{h1}} \right) \cdot 100\%
\]

3. Numerical solver
The simulation were performed using Computational Fluid Dynamics (CFD) code Ansys CFX, that solves equations of continuity, momentum end energy using the Reynolds-Averaged Navier-Stokes approach.

In the present investigation the k – \( \omega \) shear stress transport (SST) turbulence model was used. SST model is one of the most successful Hybrid models. It combines the k – \( \omega \) model near the wall and the k – \( \varepsilon \) model further from the wall to utilize the strengths of each model. SST model can produce better predictions of fluid properties in impinging jet flows and is recommended as the best compromise between the solution speed and accuracy [12]. This is interesting to note that results from presented numerical solver were compared with physical experiment and gives satisfying results [14].

4. Results and discussion
This section provides the analysis of the flow field behavior of the array of six impingement jets. Definition of pressure drop in the form of the discharge coefficient is given. Finally, the heat transfer characteristics in terms of the Nusselt number is presented.

4.1. Flow field characteristics
The proper prediction of the heat transfer values needs an analysis of the flow field mechanism. An array of presented six nozzles has a very complex behavior. Firstly, it is a result of the deflection at the stagnation areas. Secondly, the jet to jet interaction appears during the experiment. The flow direction in the tube plenum is perpendicular to that in nozzles. Figure 4 and Figure 5 illustrate numerical results of the velocity of the fluid, which appears during the investigation of the cylindrical and chamfer nozzle impingement jets, respectively. Presented figures correspond to the first and the second jet of the array of six impingement nozzles. The diameter of the impingement jets was small \((D=0.8mm)\) compared to the diameter of the plenum \((D_T=5mm)\).
It was found out that the air velocity at the area of each type of the nozzles maintained almost constant and the mass flow entering the plenum was uniformly distributed among nozzles. Figure 6 presents the velocity distribution $w$ related to the jet inlet velocity $w_o$ for the jet axis for the first and the last nozzle of the cylindrical and countersunk geometry.

The sudden change in the flow direction in the nozzles before discharging onto the target plate led to flow deflection in the direction of the flow inside the plenum. The biggest deflection was observed for the chamfer nozzle geometry when the flow was not directed perpendicularly to the target plate (see Figure 5). For all cases the deflection angle decreased downstream in the plenum towards to the last jet. The nozzle pitch-diameter ratio $S/D=8$ resulted in adjacent jet flows interactions and a fountain effect was observed leading to re-circulations.
4.2. Pressure drop
Pressure drop characteristics includes both the pressure drop in the distribution tube and in impingement nozzles. For different nozzle shapes pressure drop characteristics were performed using Computational Fluid Dynamics (CFD), the Ansys CFX code. To verify results of the pressure drop additional investigation according to Bernoulli’s formulas [4] were performed. Input values of velocities were taken from the Ansys CFX code. Results of both investigations are convergent. Figure 7 presents characteristics of the pressure drop $P_d$ between the different nozzle geometry in relation to the supply tube inlet dynamic pressure $P_i$.

![Figure 7](image)

**Figure 7.** Normalized Pressure drop $P_d$ to supply tube inlet dynamic pressure $P_i$ through different nozzle geometry.

4.3. Heat transfer characteristics
The heat transfer measurements along the impingement surface is presented in terms of the Nusselt number

$$ Nu = \frac{hD}{k} \quad (2) $$

where $h$ is the heat transfer coefficient, $D$ is the nozzle diameter, and $k$ is the thermal conductivity of the fluid.

The heat transfer coefficient is defined as

$$ h = \frac{q_w}{T_w - T_{jet}} \quad h = -k \frac{\partial T}{T_{jet} - T_w} \quad (3) $$

where $q_w$ is the wall heat flux, $T_w$ is the wall adiabatic temperature, $T_{jet}$ is a jet temperature,
The optimal nozzle configuration for a given system will be determined by the factor of pressure drop and the flow rate to achieve the demanded heat transfer coefficient and the average Nusselt number. In the Figure 8 the Nusselt number \( Nu_o \) distribution corresponding to usage of different geometrical nozzle configurations is presented. In comparison to other geometrical configurations, the usage of chamfer nozzle results in low values of the heat transfer coefficient and the Nusselt number \( Nu_o \) in the stagnation region. In this system, except the first jet where the deflection angle is big, the heat transfer coefficient is uniform. The big deflection angle resulted in non-perpendicular direction of the flow to the target surface. For countersunk and cylinder long nozzle configuration, the Nusselt number \( Nu_o \) resulted in very similar values. The highest values of the Nusselt number \( Nu_o \) were achieved for the cylinder short configuration of nozzles.

\[
\frac{\partial T}{\partial n} \quad \text{gives the temperature gradient component normal to the wall}
\]

The optimal nozzle configuration for a given system will be determined by the factor of pressure drop and the flow rate to achieve the demanded heat transfer coefficient and the average Nusselt number. In the Figure 8 the Nusselt number \( Nu_o \) distribution corresponding to usage of different geometrical nozzle configurations is presented. In comparison to other geometrical configurations, the usage of chamfer nozzle results in low values of the heat transfer coefficient and the Nusselt number \( Nu_o \) in the stagnation region. In this system, except the first jet where the deflection angle is big, the heat transfer coefficient is uniform. The big deflection angle resulted in non-perpendicular direction of the flow to the target surface. For countersunk and cylinder long nozzle configuration, the Nusselt number \( Nu_o \) resulted in very similar values. The highest values of the Nusselt number \( Nu_o \) were achieved for the cylinder short configuration of nozzles.

Figure 8. Nusselt number \( Nu_o \) distribution along target surface for different nozzle geometry.

Figure 8 shows that \( Nu_o \) increases slightly along the supply tube axis. At the entrance to the first nozzle flow is directed perpendicularly to the jet axis. Because of the closed end of the supply-tube air approaching last nozzles is directed with angle lower than 90deg to the nozzles entrance and since the flow rate maintains almost constant along all jets \( Nu_o \) increase slightly along the jets located on the end of the supply-tube.

5. Conclusions
The aim of the work was the numerical analysis of a flow characteristic, pressure drop, heat transfer coefficient and the Nusselt number distribution for an array of six impingement jets. The usage of an array of cooling jets resulted in the uniform distribution of the mass flow and pressure drop among each type of nozzles. For the chamfer nozzle configuration the biggest deflection angle was observed especially at the first jet. The lowest pressure drop occurs during the analysis with the countersunk type of nozzles. The high pressure drop for cylindrical nozzles are caused by rapid increase in the flow rate while entering the nozzle. This analysis might approach to idea of the usage of different nozzle geometry for one configuration of the cooling system if the target plate has no constant temperature.
This might result in the uniform heat transfer distribution and therefore the lower thermal stresses on a target plate. The presented analysis shows, that the heat transfer coefficient depends on the direction of the flow while the air is entering the nozzle. Therefore, the further analysis of installing features between adjacent jets to redirect the flow and to make it more convergent with the jet axis are justified.

6. References

[1] Goordo M, Jongmyung P, Ligrani P, Fox M and Hee-Koo M 2007 Effects of Mach number and Reynolds number on jet array impingement heat transfer Inter. J. of Heat and Mass Transfer 50 pp 367-380

[2] Nimalkumar M, Katti V and Prabhu S V 2011 Local heat transfer distribution on a smooth flat plate impinged by a slot jet Inter. J. of Heat and Mass Transfer 54 pp 727-738

[3] Jung-Yang S and Wen-Zheng S 2006 Effects of jet plate size and plate spacing on the stagnation Nusselt number for a confined circular air jet impinging on a flat surface Inter. J. of Heat and Mass Transfer 49 pp 3477-86

[4] Royne A and Dey Ch 2006 Effect of nozzle geometry on pressure drop and heat transfer in submerged jet arrays Inter. J. of Heat and Mass Transfer 49 pp 800-804

[5] Koseoglu M F and Baskaya S 2010 The role of jet inlet geometry in impinging jet heat transfer, modeling and experiments Inter. J. of Therm. Scien. 49 pp 1417-26

[6] Whelan B and Robinson A 2009 Nozzle geometry effects in liquid jet array impingement Appl. Therm. Engineering. 29 pp 2211-21

[7] Attalla M and Salem M 2013 Effect of nozzle geometry on heat transfer characteristics from a single circular air jet Appl. Therm. Engineering 51 723-733

[8] Zukowski M 2013 Heat transfer performance of a confined single slot jet if air impinging on a flat surface Inter. J. of Heat and Mass Transfer 57 pp 484-490

[9] Shu J J 2010 Heat transfer on a hot surface impinged by a cold circular liquid jet V Europ. Conf. on CFD, ECCOMAS CFD

[10] Ahmed F B, Weigard B and Meier K 2010 Heat transfer and pressure drop characteristics for a turbine casing impingement cooling system Proceed. of Intern. Heat Transfer Conf., Washington, USA

[11] Ee-Mahghany W M, Hanafy A A, Khaled M A and Mohamed A T 2012 Numerical simulation for confined rectangular slot jets impingement on isothermal horizontal plate, Europ. J. of Scien. Research 76 pp 553-566

[12] Zuckerman N and Lior N 2006 Jet impingement heat Transfer: Physics, Correlations and Numerical Modeling, Adv. in Heat Transfer 39

[13] Błoński S 2009 PhD Thesis: Laminar-turbulent flow analysis in micro-channels Polish Academy of Science, Institute of Fund. Techn. Research http://www.ippt.pan.pl/_download/doktoraty/blonski_doktorat.pdf

[14] Marzec K and Kucaba-Pietal A 2013 Applications of computer science in impingement cooling system design Pre-Proceed. of 9th Inter. Conf. on Appl. Mathem., Baia Mare, Romania