

# Numerical investigation of a heat transfer characteristics of an impingement cooling system with non-uniform temperature on a cooled surface

K Marzec<sup>1</sup> and A Kucaba-Pietal<sup>2</sup>

<sup>1</sup>Rzeszow University of Technology, MTU Aero Engines, Rzeszow, Poland

<sup>2</sup>Rzeszow University of Technology, Rzeszow, Poland

E-mail: <sup>1</sup>k\_marzec@prz.edu.pl, <sup>2</sup>anpietal@prz.edu.pl

**Abstract.** A series of numerical analysis have been performed to investigate heat transfer characteristics of an impingement cooling array of ten jets directed to the flat surface with different heat flux  $q_w(x)$ . A three-dimensional finite element model was used to solve equations of heat and mass transfer. The study focused on thermal stresses reduction on a cooled surface and aims at answering the question how the Nusselt number distribution on the cooled surface is affected by various inlet flow parameters for different heat flux distributions. The setup consists of a cylindrical plenum with an inline array of ten impingement jets. Simulation has been performed using the Computational Fluid Dynamics (CFD) code Ansys CFX. The  $k - \omega$  shear stress transport (SST) turbulence model is used in calculations. The numerical analysis of the different mesh density results in good convergence of the GCI index, what excluded mesh size dependency. The physical model is simplified by using the steady state analysis and the incompressible and viscous flow of the fluid.

## 1. Introduction

### 1.1 Motivation

An impingement cooling system is an array of jets with high velocity fluid which is made to strike a target surface. Such systems are installed in electronic devices, aero engines, heavy industry equipment, automotive industry equipment and many others. Technological processes are supported by impingement cooling systems. This is due to the fact that modern machines operate in very high temperatures to achieve high efficiency. Therefore usage of impingement cooling systems is mandatory to provide high rate of heat and mass transfer. Such applications as drying of paper, de-icing of aircraft wings, heat exchangers in automotive applications, cooling of gas turbine blades and turbine casing in aero engines are a few examples of jet impingement cooling. Impingement cooling of the gas turbine casings besides high heat and mass transfer reduces fuel consumption of the engine. Those cooling systems are called Active Clearance Control (ACC) [1-3]. Taking into consideration turbo fan engines, cooling systems are used to reduce temperature of the low and high pressure turbine casings. Many experimental and numerical investigations of arrays of impingement cooling jets directed to the flat surface with constant temperature [4], [6] have been performed. Many of them are

<sup>1</sup> Corresponding author: k\_marzec@prz.edu.pl , phone +(48)-731-706-731



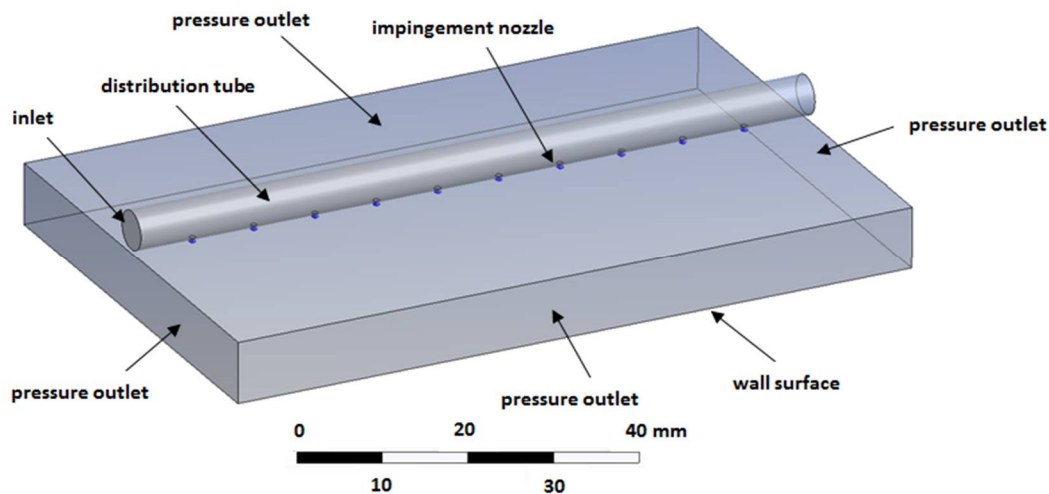
focused on high Nusselt number delivery [7]. The majority of the studies are experimental ones [8-9]. However many simulations of the impingement cooling systems are numerical [10]. Only a few are devoted to the problem of cooling a surface with an inhomogeneous distribution of temperature. Such situation occurs in many technical applications such as photovoltaic cells.

### 1.2 Objective

The main goal of the numerical investigation was to examine an inline array of ten impingement jets which is heated with various heat fluxes  $q_w(x)$ . Additionally, various inlet flow parameters were analysed.

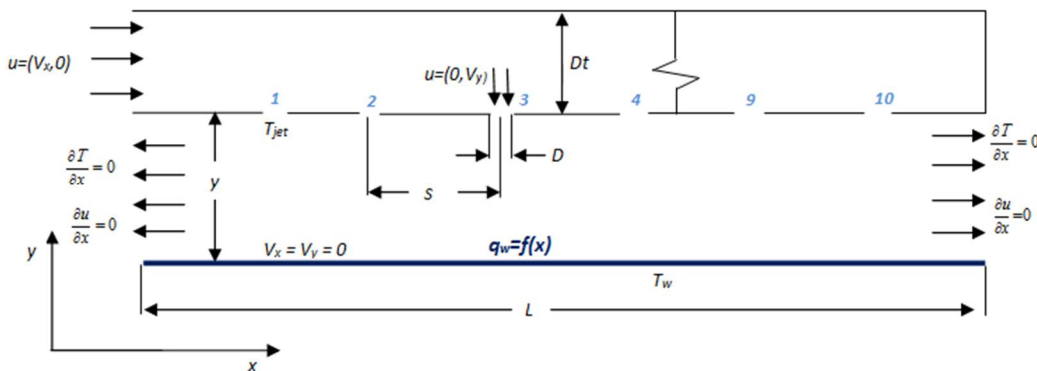
## 2. Problem formulation

The geometry of the system under consideration is shown in Figure 1. Fluid data, thermal and flow parameters used for the calculations are extension of the research work presented in [11]. The impingement cooling system consists of an array of ten cylindrical impingement nozzles directed normally to the flat surface. In this study diameter  $D=0.8$  mm of the nozzle was taken into consideration. A Schematic 2D diagram of the problem is presented in Figure 2. The nozzle exit-plate distance  $X/D=8$  and the nozzle pitch-diameter ratio  $S/D=8$  is constant for all simulations. The distribution tube inlet diameter is  $D_T=5$ mm. The total length of the target plate is  $L=88$ mm. The width of the target plate is  $W=56$ mm. This ensured that the outlet boundaries were away from the section of interest where the jet impinged onto the target plate. Air is supplied to the system by the supply-tube only from the left side. The right side of the tube is closed.



**Figure 1.** Investigated computational 3D domain.

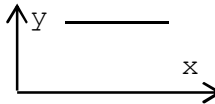
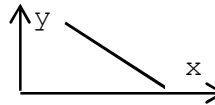
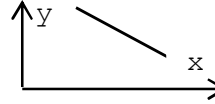
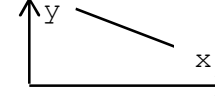
Both the left hand side and the right hand side of the system are open to allow the fluid to flow. Fluid is free to expand after imping the target surface. Cooling air is supplied by the distribution tube from the left side. The right side of the distribution tube is closed.



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

The wall, the jet impinged onto is heated with the heat flux represented by four different linear functions presented in Table 1.

**Table 1.** Cooled surface heat flux  $q_w(x)$  representation by different linear functions.

| No. | Heat flux function $q_w(x)$ | $q_w(0)$<br>[W/m <sup>2</sup> ] | $q_w(88)$<br>[W/m <sup>2</sup> ] | Graf of function<br>$q_w(x)$  |
|-----|-----------------------------|---------------------------------|----------------------------------|---|
| 1.  | $q_w(x)=5000$               | 5000                            | 5000                             |   |
| 2.  | $q_w(x)=-56.81x+5000$       | 5000                            | 0                                |  |
| 3.  | $q_w(x)=-45.45x+5000$       | 5000                            | 1000                             |  |
| 4.  | $q_w(x)=-28.41x+5000$       | 5000                            | 2500                             |  |

### 3. Governing equations

The governing equations for the problem under consideration are based on the conservation laws of the mass, the linear momentum, and energy in RANS form in the steady state and for incompressible and viscous fluid flows [5]. The heat transfer rates have been described by the local and the line averaged Nusselt number. Thereby, the Nusselt number is defined as:

$$Nu = \frac{hD}{k} \quad (1)$$

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:

$$h = \frac{q_w}{T_w - T_{jet}} \quad h = -k \frac{\frac{\partial T}{\partial n}}{T_{jet} - T_w} \quad (2)$$

where  $q_w$  is the wall heat flux,  $T_w$  is the wall adiabatic temperature,  $T_{jet}$  is the jet temperature,

$\frac{\partial T}{\partial n}$  gives the temperature gradient component normal to the wall

In this study, the line averaged Nusselt Number is defined as below:

$$\overline{Nu} = \frac{1}{L} \int_L Nu(x) dx \quad (3)$$

where  $L$  is the line of the plate parallel to the distribution tube axis.

#### 4. Solution method

The simulations were performed using Computational Fluid Dynamics (CFD) code Ansys CFX, that solves equations of continuity, momentum and energy using the Reynolds-Averaged Navier-Stokes approach. In the present investigation the  $k - \omega$  shear stress transport (SST) turbulence model was used. It combines the  $k - \omega$  model near the wall and the  $k - \epsilon$  model further from the wall to utilize the strengths of each model. SST model is recommended as the best compromise between the solution speed and accuracy [5]. The investigation is limited to the steady state assumption and dynamic features of the impinging jets are ignored. However, the steady state assumption is able to provide average flow and temperature fields.

#### 5. Numerical Setup

##### 5.1. Initial and Boundary Conditions

The velocity of the flow  $u=(V_x, V_y)$  at the inlet of the supply tube  $V_x=15\text{m/s}$  is prescribed to obtain the Reynolds number  $Re=4166$  at the impingement nozzle. Thereby, the Reynolds number is based on the mean velocity at the nozzle and the jet diameter  $D$ . The air density is  $\rho=1.17 \text{ kg/m}^3$ , the thermal conductivity is  $k=0.025 \text{ W/mK}$  and the dynamic viscosity is  $\mu=1.8 \cdot 10^{-5} \text{ Pas}$ . The fluid temperature at the inlet is  $T_{jet}=20^\circ\text{C}$ . The outlet boundaries of the calculation domain are represented by opening pressure boundary conditions which permit fluid to both enter or leave the domain. To simplify the analysis, the steady-state, incompressible, viscous fluid flow is considered and it is 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  $30 \mu\text{m}$ .

##### 5.2. Numerical grids and numerical accuracy

The numerical calculations were carried out using unstructured tetrahedral grids with 1.79 mln elements and 323119 nodes generated by the Ansys CFX mesher. 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 the surface) were taken into consideration. As to investigate the sensitivity of the results of the numerical analysis, the Grid Convergence Index GCI (equation 4) was calculated [12]. The safety factor  $F_s = 3$ , was set for two grids comparison. Temperature was the chosen parameter. It was measured in the cooled surface for each grid. Order of convergence was  $p=2$ ,  $r^p$  was the density factor. For the third analysis  $GCI=1.3\%$

was obtained. Therefore, it might be concluded that numerical results on the fine grid are grid independent.

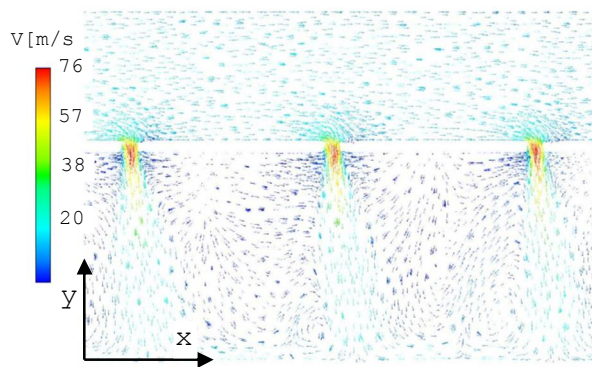
$$GCI = F_s \frac{\left| \frac{T_{h2} - T_{h1}}{T_{h1}} \right|}{r^p - 1} 100\% \quad (4)$$

## 6. Results and discussion

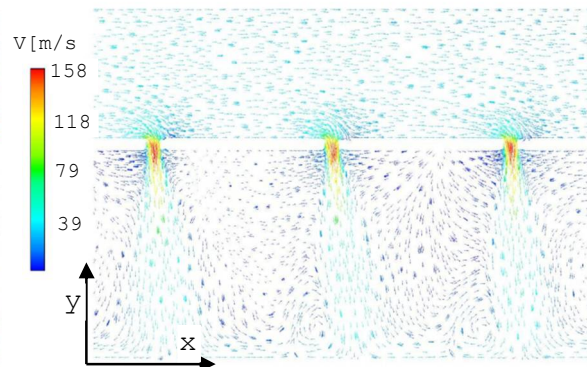
This section provides the analysis of the flow field behaviour and heat transfer characteristic of an array of ten impingement jets directed normally to the flat surface with four various ways of the heat flux  $q_w(x)$  distribution.

### 6.1. Flow field characteristics

Results of simulations with two different distribution tube inlet velocity values  $u=15\text{m/s}$  and  $u=30\text{m/s}$  are presented. Flow of an array of ten impinging nozzles have a very complex behaviour for each type of the jets and inlet flow parameters. It is a result of the deflection at the stagnation area (especially at the first jets) and because of the jet to jet interaction, which is called “fountain effect”. It is worth to note that this phenomena depends on the neighbouring nozzle distance [5]. If this distance is large enough, the fountain effect is negligible. However if the jet axis are located close to each other the heat transfer characteristic is affected due to the jet to jet interaction. In Figures 4 and 5 the velocity in the plane crossing axis in the distribution tube is sketched to visualise the fountain effect for heat distribution on the cooled plane described by the  $q_w(x)=5000\text{W/m}^2$ .



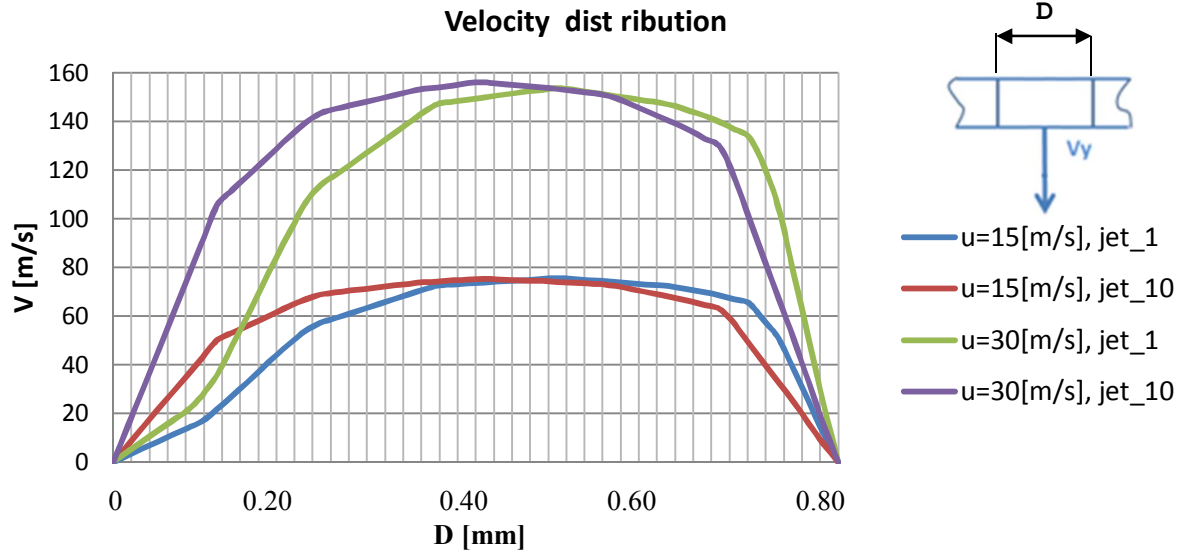
**Figure 4.** Velocity vector plot for first three adjacent cylindrical nozzles, distribution tube inlet velocity  $u=15\text{m/s}$ .



**Figure 5.** Velocity vector plot for first three adjacent cylindrical nozzles, distribution tube inlet velocity  $u=30\text{m/s}$ .

The axis direction in the distribution tube is perpendicular to that in the nozzles. The diameter of the impingement holes  $D$  is small compared to the diameter of the distribution tube diameter  $D_T$ . It is observed that the mass flow entering the distribution tube is uniformly distributed among the ten nozzles and maintains almost constant for both inlet velocities. Discrepancy in mass flow  $Q$  between the first and the last nozzle is 6.28% for  $u=15\text{m/s}$  and 9.46% for  $u=30\text{m/s}$ . The sudden change in the flow direction in the nozzles before striking onto the target plate results in the impinging jet deflection angle in the direction of the flow inside the distribution tube. The highest flow deflection angle is observed at the first nozzle. On the last one, the flow direction is perpendicular to the cooled surface. Therefore an eccentricity of the velocity distribution at the nozzle area can be noticed (Figure 6). This discrepancy can be found for both distribution tube inlet velocities  $u=15\text{m/s}$  and  $u=30\text{m/s}$ . It seems

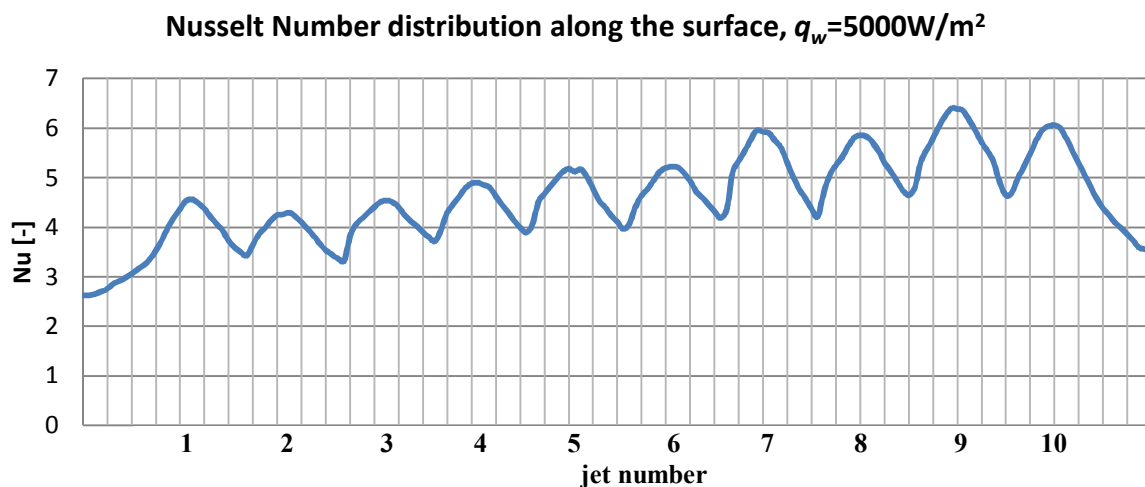
that it is a result of the closed right side of the distribution tube (air flow has no longer possibility to flow parallel to the distribution tube and finally, it is redirected to the nozzle).



**Figure 6.** Velocity distribution  $V_y$  in the cross-section area of the first and last cylindrical jet for inlet velocity  $u=15\text{m/s}$  and  $u=30\text{m/s}$ .

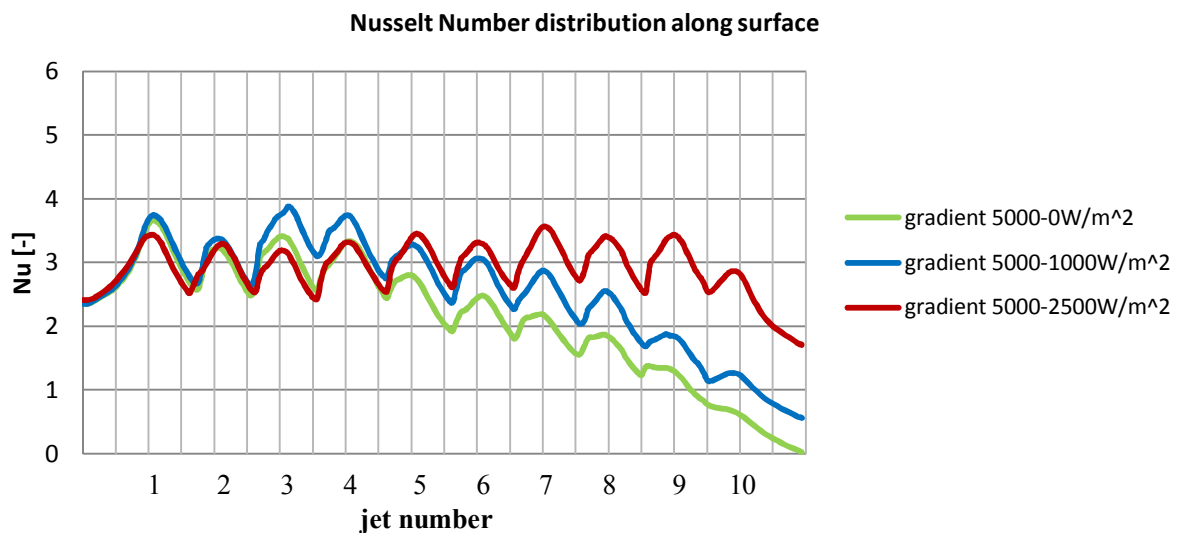
### 6.2. Heat transfer characteristic

The optimal system configuration for a given heat flux gradient  $q_w(x)$  will be determined by the constant mass flow rate and the uniform Nusselt number distribution on a cooled surface. The uniform Nusselt number  $\overline{Nu}$  distribution plays a significant role as it might reduce thermal stresses on a cooled surface. In Figure 7 the Nusselt number  $\overline{Nu}$  corresponding to an usage of ten cylindrical nozzles with the constant heat flux  $q_w(x)=5000\text{W/m}^2$  is presented. In comparison to all other heat flux configurations (Table 1), the usage of a constant heat flux  $q_w=5000\text{W/m}^2$  resulted in the highest averaged Nusselt number  $\overline{Nu} = 4.59$ . In addition, it can be seen that the line of an averaged Nusselt number increased slightly in the  $x$  direction (along the flow of the distribution tube). Between nozzles, the maximum difference in the Nusselt Number  $Nu_o$  is 31.5% for the distribution tube inlet velocity  $u=15\text{m/s}$ .



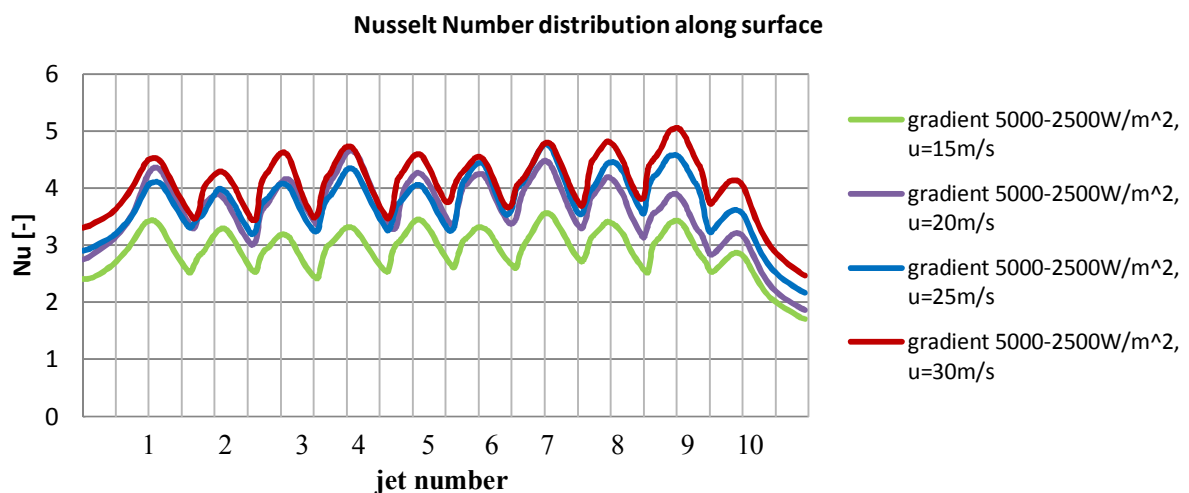
**Figure 7.** Area of the averaged Nusselt number  $\overline{Nu}$  for constant heat flux  $q_w=5000\text{W/m}^2$ , inlet velocity  $u=15\text{m/s}$ .

In Figure 8 the Nusselt number corresponding to an usage of ten cylindrical nozzles with variable heat flux  $q_w(x)$  decreasing in the direction of the flow is presented. The wall, the jet impinged onto, is heated with the heat flux represented by three different linear functions. The first one represents the heat flux  $q_w=5000\text{W/m}^2$  at the beginning of the cooled surface and  $q_w=0\text{W/m}^2$  at the end of the cooled surface. The second one represents the heat flux  $q_w=5000-1000\text{W/m}^2$ . The third one represents the heat flux  $q_w=5000-25000\text{W/m}^2$ . The distribution tube inlet velocity is  $u=15\text{m/s}$ . It can be seen that the line of the averaged Nusselt number decreased in the direction of the flow in the distribution tube for the first and the second heat flux functions. The heat flux  $q_w=5000-25000\text{W/m}^2$  represents similar values of the Nusselt number  $Nu_o$  in the stagnation region for all of the jets. The average difference is 8.05%.



**Figure 8.** Area of the averaged Nusselt number  $\overline{Nu}$  for various heat flux  $q_w(x)$ , inlet velocity  $u=15\text{m/s}$ .

Figure 9 presents same experiment's results with the increased inlet velocity up to  $u=20\text{m/s}$ ,  $u=25\text{m/s}$  and  $u=30\text{m/s}$  which was performed to obtain the Reynolds numbers:  $Re=5555$ ,  $Re=6944$  and  $Re=8333$  at the impingement nozzle. Results of this experiment show, that the average difference in the Nusselt number  $Nu_o$  between nozzles is 9.38% for  $u=20\text{m/s}$  12.4% for  $u=25\text{m/s}$  and 10.4% for  $u=30\text{m/s}$ .



**Figure 9.** Area of the averaged Nusselt number  $\overline{Nu}$  for heat flux  $q_w=5000-2500\text{W/m}^2$  and various distribution tube inlet velocity.



## 7. Conclusions

The aim of the work was the numerical analysis of the flow and heat transfer characteristics of the Nusselt number distribution for an array of ten impingement jets directed to the flat surface with different heat flux distribution  $q_w(x)$  (see Table 1). The usage of the array of cooling jets resulted in the uniform distribution of the mass flow and among the nozzles. The usage of the constant heat flux  $q_w=5000\text{W/m}^2$  resulted in the highest averaged Nusselt number  $\overline{Nu} = 4.59$ . The line of the averaged Nusselt number decreased in the direction of the flow in the distribution tube for the heat flux  $q_w(x)=5000-0\text{W/m}^2$  and  $q_w(x)=5000-1000\text{W/m}^2$ . For the heat flux  $q_w(x)=5000-2500\text{W/m}^2$  the Nusselt number distribution was very similar for all of the nozzles. The average difference in the Nusselt number  $Nu_o$  was 8.05%. The increased Reynolds number  $Re$  resulted in higher average values of the Nusselt number  $\overline{Nu}$  across the cooled surface. Presented conclusions are significant for the designers to handle with the temperature gradient on the cooled surface. Especially during the design of the photovoltaic cells or turbine cooling systems. It is worth to take into consideration usage of a various nozzle geometry for the cooling systems operating with non-uniform heat flux on the cooled surface. This might result in homogenisation of the average Nusselt number  $\overline{Nu}$  and finally reduce thermal stresses of the cooled surface.

## 8. References

- [1] Andreini A, Da Soghe R, Facchini B, Maiuolo F, Tarchi L, Coutandin D 2013 *Journal of Turbomachinery* **135** 031016
- [2] Ahmed F B, Weigand B and Meier K 2010 Heat transfer and pressure drop characteristic for a turbine casing impingement cooling system *Proceedings of 14th International Heat Transfer Conference* Washington
- [3] Ruiz R, Alberts B, Sak W, Seitzer K, Steinetz B 2006 Benefits of improved HP Turbine Active Clearance Control NASA/CP—2007-214995 Air System Workshop Cleveland OH
- [4] Hee H, Kyung Ch, Kim M, Song J 2011 Applications of impingement jet cooling systems, Department of Mechanical *In: Cooling Systems: Energy, Engineering and applications* Editor: Aaron I Shanley Nova Science Publishers 37-68
- [5] Zuckerman N and Lior N 2006 *Advanced in Heat Transfer* **39** 565–631
- [6] El-Maghlany M, Hanafy A, Abdou M, Teamah A 2012 *European Journal of Scientific Research* **76** 553-566
- [7] San J and Shiao W 2006 *International Journal of Heat and Mass transfer* **49** 3477-86
- [8] Nirmalkumar M, Katti V, Prabhu S V 2011 *International Journal of Heat and Mass Transfer* **54** 727-738
- [9] Goordo M, Jongmyung P, Ligrani P, Fox M and Hee-Koo M 2007 *International Journal of Heat and Mass Transfer* **50** 367-380
- [10] Zukowski M 2013 *International Journal of Heat and Mass Transfer* **57** 484-490
- [11] Marzec K and Kucaba-Piętal A 2014 Heat transfer characteristic of an impingement cooling system with different nozzle geometry *Journal of Physics Conference Series Ser. 530* 012038
- [12] Błoński S 2009 *Polish Academy of Science, Institute of Fund. Techn. Research* PhD Thesis: Laminar-turbulent flow analysis in micro-channels

## 9. Acknowledgements

Acknowledgements: The calculations were performed in ICM UW, Grant G64-0