Investigation of the Effect on Heat Transfer and Flow Characteristics of Circular Impingement Jets in a Rectangular Duct

Mehmet Gürdal

July 20, 2022

3rd International Conference on Applied Engineering and Natural Sciences

https://www.icaens.com/

July 20-23, 2022, Konya, Turkey

Investigation of the effect on heat transfer and flow characteristics of circular impingement jets in a rectangular duct

Mehmet Gürdal*

[1]Kastamonu Vocational High School, Kastamonu University, Kastamonu

*(mgurdal@kastamonu.edu.tr)

Abstract – In the study, it was investigated air in rectangular channel with jet impingement. It was examined numerically air flow under constant inlet temperature, and constant target surface wall boundary condition. All analyzes have been carried out by using Ansys Fluent software. It was used SST k-w turbulent model in the study. Basic parameters of the study is nozzle spacing, jet diameter, Reynolds number, Nusselt number, and pressure difference. It was examined turbulent air flow in the range of 15000Re≤35000. On the other hand, it was discussed flow behavior in the rectangular channel and local Nusselt number. As a result, it was observed that finding obtained compatible with well-known correlations. Average Nusselt number increased with increasing Reynolds number and decreasing target surface-to-nozzle spacing. In addition to, pressure difference increase logarithmically with Reynolds number.

Keywords – CFD, jet impingement, heat transfer, air flow, pressure drop

I. INTRODUCTION

Most of new designed energy storage devices have more efficiency according to traditional systems. Therefore, better heating or cooling performances are needed. For this aim, the jet impingement technique, which is one of the most substantial active heat transfer methods, is generally used in some industrial areas such as mini electronic card cooling [1], enhance of nanofluid's heating and cooling performance [2], gas turbine blade [3], cryosurgery [4]. Flow and heat transfer properties of impinging jets; It is known that it varies depending on many parameters from the jet exit geometry to the velocity profile at the jet exit, from the distance between the jet and the plate to the turbulence in the jet, from the striking plate geometry to the temperature difference between the jet and the plate [5].

II. SOLVER SETUP

In the this study, the numerical analyzes are implemented using Ansys Fluent program. A three-dimensional channel flow is setup with double precision, utilizing pressure-based solver neglecting body forces. The channel structure has been showed in figure 1 and figure 2. It was used turbulent model as SST k-w in the study.

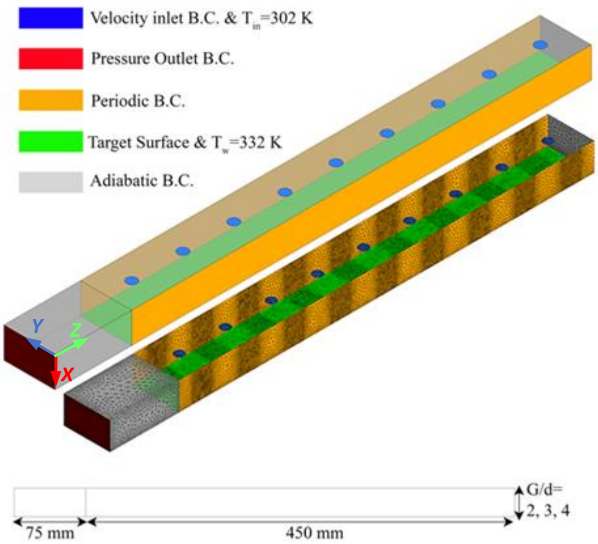

Fig. 1. Geometry, mesh, dimensions.

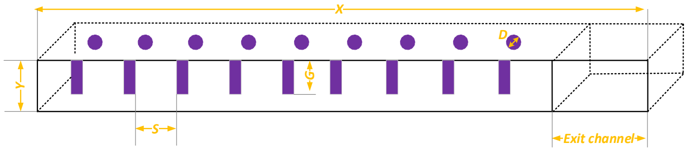

Fig. 2. Geometric parameters of the channel with jet.

The geometric dimensions of the impinging jet and the channel, properties of air, and mesh adaptation structure are presented in table 1, table 2, and table 3, respectively.

Table 1. Summary of geometric parameters

D	*X/D*	*G/D*	*Y/D*	*S/D*	*Exit*	*n*
10 mm	52.5	4	5	5	75 mm	9
10 mm	52.5	3	5	5	75 mm	9
10 mm	52.5	2	5	5	75 mm	9

Table 2. Thermophysical properties of Air

Material	*ρ(kg/m³)*	*k(W/m²)*	*μ(Pa.s)*	*cₚ(J/kgK)*
Air	996.24	0.6158	0.0008206	4179

On the other hand, it was solved base contiunity, momentum, and energy conservation equation by using Navier-Stokes balance with finite volume method. These equations have been presented Eq.(1), Eq.(2), and Eq.(3) as follows. Also, other important parameters was given Eq.(4), Eq.(5), and Eq.(6) with [6] reference.

Contiunity [7]:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u_i)}{dx}=0 \qquad (1)$$

Momentum [7]:

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_i u_j)}{\partial x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}[\mu(\frac{\partial u_i}{\partial u_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_j})]+i\frac{\partial}{\partial x_j}(-\rho\overline{u'_i u'_j})$$

(2)

Energy [8]:

$$\frac{\partial}{\partial t}(\rho E)+\frac{\partial}{\partial x_i}\left[u_i\left(\rho E+\Pr\right)\right]=\frac{\partial}{\partial t}\left[\left(k+\frac{C\mu_i}{\Pr_t}\right)\right]\frac{\partial T}{\partial x_j}+\mu_i(\tau_{ij})_{eff}$$

(3)

$$Re=\frac{\rho V D_j}{\mu} \qquad (4)$$

$$h=\frac{q''}{(T_w-T_b)_{avg}} \qquad (5)$$

$$Nu=\frac{hD_j}{k} \qquad (6)$$

Table 2. Mesh adaptation of the study

Mesh	*Grid1*	*Grid2*	*Grid3*	*Grid4*	*Grid5*
Cells	1435360	1522563	1697592	1871596	2155379
Nu	96.25	98.54	101.544	103.347	103.987
y+	1.37	1.01	0.98	0.86	0.83

III. RESULTS

Firstly, it has been validation with study which well-known correlations in literature as follows in figure 3. As result, it was observed that local Nusselt number suitable with literature. Then, variation of Nusselt number with Reynolds number have been examined in figure 4. When they were investigated in this graphics, it was observed that the Nusselt number linearly increase with increasing Reynolds number.

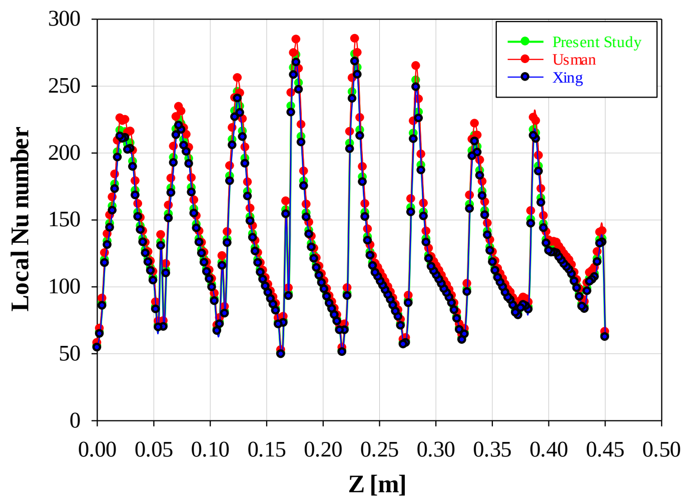

Fig. 3. Validation of local Nusselt number with the literature.

Also, pressure difference tends to increase logarithmically with increasing Reynolds number and decreasing G/D ratio as seen in figure 5. It was showed velocity distribution and surface Nusselt number at Re=35000 as seen in figure 6 and figure 7.

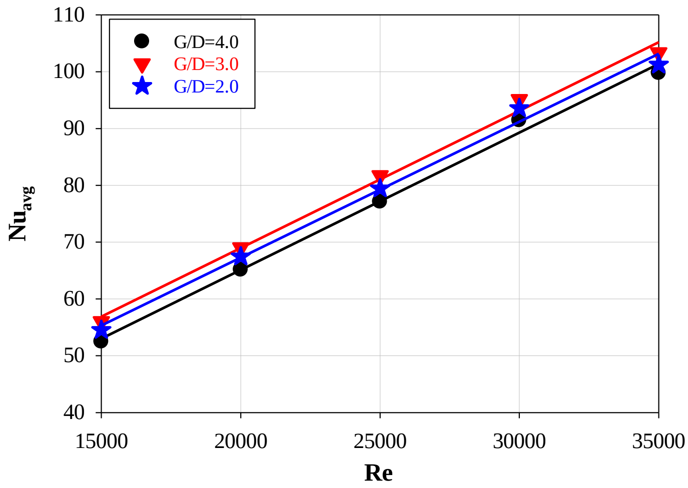

Fig. 4. Variation of average Nusselt number with Reynolds number.

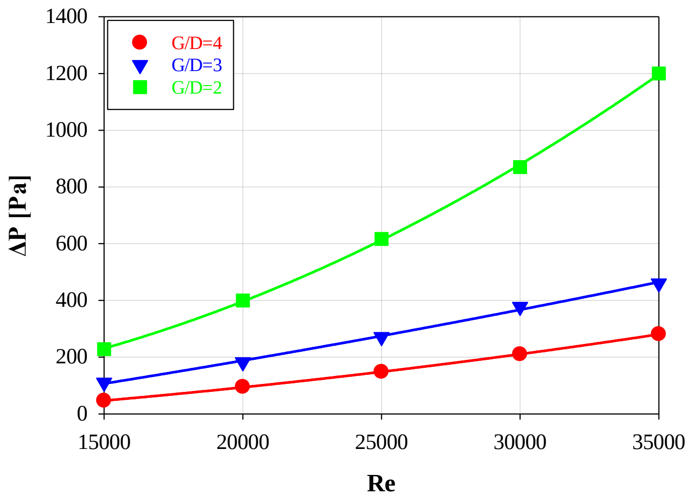

Fig. 5.Variation of pressure difference with Reynolds number.

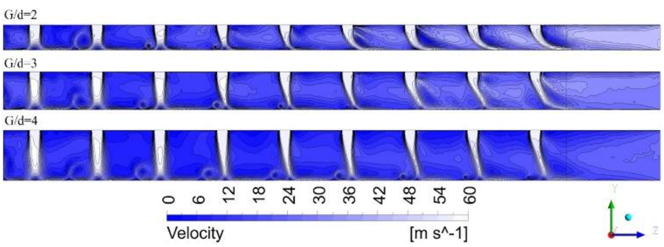

Fig. 6.Velocity distribution at Re=35000.

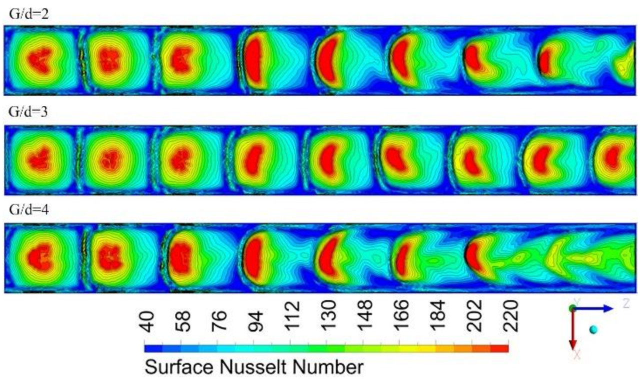

Fig. 7. Surface Nusselt number at Re=35000.

IV. CONCLUSIONS

The conclusions gained from the analyzes carried out by using the Ansys Fluent software are as follows:

- The average Nusselt number was obtained the highest value for G/D=3.0.
- It was observed that pressure difference decrease with increasing target surface-to-nozzle spacing. Therefore, pump power required for cooling or heating systems is to be less.
- It was observed that optimum target surface-to-nozzle spacing and maximum heat convection is G=3D.
- Maximum vortex and secondary flow observed in case G/D=3.

REFERENCE

[1] H. K. Pazarlıoğlu, A. Ü. Tepe, And K. Arslan, "Optimization Of Parameters Affecting Anti-Icing Performance On Wing Leading Edge Of Aircraft," *Eur. J. Sci. Technol.*, No. 34, Pp. 19–27, 2022, Doi: 10.31590/Ejosat.1062495.

[2] "(Pdf) Çarpan Jetle Isı Transferinde Isı Ve Akım Karakteristiklerinin Dikdörtgen Kanatçıklı Isı Alıcıda Belirlenmesi." https://Www.Researchgate.Net/Publication/320064659_Carpan_Jetle_Isi_Transferinde_Isi_Ve_Akim_Karakteristiklerinin_Dikdortgen_Kanatcikli_Isi_Alicida_Belirlenmesi (Accessed Jun. 17, 2022).

[3] M. Sözbir And Ü. Uysal, "Çok Sıralı Çarpmalı Jetlerin İki Geçişli Dikdörtgen Kanaldaki Isı Taşınımının Deneysel İncelenmesi," *J. Fac. Eng. Archit. Gazi Univ.*, Vol. 37, No. 2, Pp. 671–684, Feb. 2022, Doi: 10.17341/Gazımmfd.708122.

[4] F. Monnoyer And D. Lochegnies, "Heat Transfer And Flow Characteristics Of The Cooling System Of An İndustrial Glass Tempering Unit," *Appl. Therm. Eng.*, Vol. 28, No. 17–18, Pp. 2167–2177, Dec. 2008, Doi: 10.1016/J.Applthermaleng.2007.12.014.

[5] A. N. Özakin And F. Yeşildal, "Çarpan Jetle Isı Transferinde Geometrik Parametrelerin Sayısal Optimizasyonu: Yanıt Yüzey Yöntemi Numerical Optimization Of Geometric Parameters İn İmpinging Jet Heat Transfer: Response Surface Methodology," Vol. 12, No. 1, Pp. 357–369, 2022, Doi: 10.17714/Gumusfenbil.1026237.

[6] Yunus A. Çengel And Afshin J. Ghajar, *Heat And Mass Transfer Fundamentals And Applications*, Fifth. Mcgraw Hill, 2011.

[7] L. M. Jiji, "Heat Convection: Second Edition," *Heat Convect. Second Ed.*, Pp. 1–543, 2009, Doi: 10.1007/978-3-642-02971-4.

[8] A. Bejan, *Convection Heat Transfer*. John Wiley \& Sons, 2013.