

Heat transfer and fluid flow characteristics investigation using detached ribs in an axisymmetric impinging jet flow

Adnan Rasheed^{a*}, Usman Allauddin^b, H. M. Ali^c, Muhammad Uzair^b, Patrick G. Verdin^d, Yasir H. Siddiqui^a

^a *Department of Mechanical Engineering, Khwaja Fareed University of Engineering & Information Technology, 64200 Rahim Yar Khan, Pakistan*

^b *Department of Mechanical Engineering, NED University of Engineering & Technology, 75270 Karachi, Pakistan*

^c *Mechanical Engineering Department, King Fahd University of Petroleum and Minerals, Dhahran 31261, Saudi Arabia*

^d *Energy & Power, School of Water, Energy & Environment, Cranfield University, Cranfield MK43 0AL, UK*

Abstract

This research work is directly concerned with the heat transfer involved in a single jet flow covering a heated surface having detached ribs present above the surface. A computational study has been conducted to check the effect due to the presence of such detached ribs. The generalized $k - \omega$ (GEKO) turbulence model with Reynolds-averaged Navier Stokes (RANS) equations formulation has been adopted after validation with the relevant experimental data from the literature. A rectangular rib design was selected for which rib width, height, clearance, pitch and first rib radial distance were varied to study their effect on heat transfer. Local heat transfer distributions were investigated over the target surface, especially focusing on the ribs-based region for different nozzle positions. The thermal performance with and without ribs is compared. A maximum increase in heat transfer of 64% was observed for a normalized nozzle to plate spacing $z/d = 0.5$.

* Corresponding author

Email: adnan.rasheed@kfueit.edu.pk, Tel.: +92-307-7727269

Keywords: Jet impingement, heat transfer enhancement, detached ribs, GEKO turbulence model, Nusselt number.

1. INTRODUCTION

Thermal energy systems operate at high temperatures for increased performance, thus require efficient internal cooling to ensure longer life. Methodologies meant for heat transfer improvement include the utilization of a flowing fluid inside the system (the most common approach), applying design modification techniques for new surface geometries, or considering the insertion of novel roughness elements in the case of ducts and tubular geometries. In industry, many processes result in intense heating, and to ensure efficient processing, localized cooling is essential for preventing any deterioration of the physical structure, or for inducing definite physical properties to the product. This objective is achieved through impinging gas jets at particular locations, where an excessive and unwanted increase in temperature is present. Figure 1 shows a schematic diagram of jet impingement in typical applications of heat transfer through impinging jets; this includes the cooling of electronic components, cooling of machine elements facing friction, cooling of turbine components in jet engines, cooling of resulting batch products during metal forming processes, and many other industrial applications. Jet impingement techniques are also utilized in turbines, for cooling of different parts of the engine i.e., the turbine casing, inside the combustion chamber, and for the localized cooling of the turbine blade tips. Turbine blades are cooled using bleed airflow at high pressure. A heat flux of around 1 MWm^{-2} is required in order to cool the blades with the gas flowing at high temperature, around $1400\text{-}1500^\circ\text{C}$ [1]. This allows higher cyclic temperature ratios inside the turbojet engines, which ultimately leads to higher thermal efficiency. The turbine output is therefore increased, improving the fuel economy.

Many researchers have studied the effects of shape or geometry of the nozzle through which a jet is produced. Singh et al. [2] carried out an experimental and numerical study to determine how the nozzle shape affects the jet impingement heat transfer. They used a heated circular cylinder maintained at a constant heat flux for heat transfer through an unconfined jet impingement. Their work included a comparative study of circular, square and rectangular shaped nozzles having the same hydraulic diameters. They concluded that for fixed Reynolds number, a maximum heat transfer rate from the heated cylinder occurred using the circular nozzle. Another experimental and numerical study was conducted by Koseoglu et al. [3] in which they assessed how different orifice shapes affect the cooling of a flat surface through jet impingement. When comparing circular,

elliptical and rectangular orifices, a decrease in heat transfer in the wall jet region was observed, with an increase in cross-sectional area and aspect ratio of the jet. Siddiqui et al. [4] carried out numerical simulations using SST + γ - θ turbulence model on jet impingement using a circular nozzle for varying nozzle to impingement plate spacing and they were able to find out that the heat transfer rate increases with decrease in distance between the nozzle and plate beyond the stagnation region.

Various heat transfer enhancement techniques, active as well as passive, can be accompanied by the impinging jets with the aim to increase the heat transfer coefficients. These tend to improve the overall thermal efficiency of the thermal systems. Passive techniques turn out to be inexpensive as compared to the active methods. These do not require any additional power and rather involve various artificial attachments, surface roughness and flow blockages. These create turbulence, by interfering the boundary layer, which in turn enhances heat transfer. The selection of an appropriate geometry or surface roughness element should therefore take into account the pressure drop and the resulting pumping power and costs. There are many ways of creating surface roughness, which can be obtained from grooves on a plane surface, embedded foreign particles, and dimples over a flat surface. Sloped surfaces of different orientations can also be used to increase the flow turbulence. The factor of surface roughness has been studied for its impact on heat transfer by Zhang, et al. [5]. These authors used PIV to investigate the effects of protrusions developed on a surface and also obtained similar results numerically. Compared to a smooth target surface, they found out that the heat transfer increased relatively with protrusion depth. Similarly, combined heat and mass transfer was studied by Kim and Lee [6] for its variation due to surface roughness from small glass beads. Better cooling performance was achieved against a heated porous substrate.

Many researchers worked on roughened surfaces with and without the involvement of cross-flow. Beitelmal and Saad [7] observed up to 6% increase in the average Nusselt number using a single air jet impingement by the addition of a circular array of protrusions. Sharif and Ramirez [8] conducted a study on heat transfer from a convex hemispherical surface using impinging jet and observed an increase in heat transfer rates with the addition of surface roughness over the hemispherical surface. Xing and Weigand [9] used a nine-by-nine array of impinging jets on a dimpled surface and observed an increased value of heat transfer coefficient as compared to a flat surface. Gabour and Lienhard [10] obtained 50% more heat transfer rate when applying jet

impingement cooling techniques with surface roughness from protrusions. Celik [11] also studied the effects due to the presence of dimples on the target surface, utilizing a coaxial jet with concentric nozzle pipes. With the coaxial jets, he was able to get an increase in the heat transfer of up to 27%. Many other researchers, for instance Caliskan [12], Kanokjaruvijit et al. [13], Choi et al. [14] and El-Gabry [15] also proved through their investigative studies that surface roughness elements with different impinging jet configurations highly increase the heat transfer.

Rib roughened surface is one of the most popular passive methods applied to amplify the heat transfer rate in channel and cavity flows. Many engineering applications such as solar thermal energy systems, combustion, and especially the turbine blades cooling, involve a cavity flow behavior from the inner side where the flow can be envisioned as a separated wall jet flow. Ribs inside channels/cavities enhance heat transfer by providing an increased surface area. In addition, they disturb the wall sub-layer which cause flow separation and reattachment, yielding a higher heat transfer coefficient [16]. Changing the ribs shape, their positioning and geometry, also affect the heat transfer and, if an optimum design configuration is obtained, the heat transfer rate greatly improves and thus the thermal performance of a system can increase with lower pressure penalty. Tanda et al. [17] conducted an experimental study to determine the performance of a ribbed wall rectangular channel having ribs angled at 45° by varying ribs spacing and Reynolds Number. It was found that a two-ribbed wall channel gives a superior thermo-hydraulic performance at a pitch to height ratio which is almost half of a single ribbed wall channel. Tan et al. [18] conducted a study on convective heat transfer enhancement using impinging jets with different shapes of ribs attached to the heated surface. They used a single row of impinging jets striking the heated surface and cross flowing over rib structures. Their rib structures had 3 different constellations of Vee shaped ribs, inverted Vee shaped ribs and the normal/ orthogonal ribs; out of which the inverted V-shaped ribs with the inner angle of 120° were found to be the best for heat transfer up to a value of 30%. They also found that jet to plate spacing ratio should be double for highest heat transfer. However, with the installation of ribs, there is a significant drop in flow coefficient; minimum for jet to plate spacing ratio of 3 out of the 1, 2 & 3 configurations. Wongcharee et al. [19] performed a numerical investigation on the effect of various ribs shapes on the fluid flow characteristics and heat transfer through a channel with two opposite ribbed walls. They established that triangular ribs yield the highest Nusselt number while cylindrical ribs show the best thermal performance. Wang et al. [20] carried out a numerical and experimental study on the effect of various pin fins shapes having the same cross sectional area, on heat transfer and flow characteristics inside a

rectangular channel. They found out that the drop-shaped pin fins yield better thermo-hydraulic performance in comparison to circular pin fins. Chung et al. [21] performed an experimental and numerical study to examine the effect of semi-circular ribs on flow and heat transfer characteristics in a rectangular channel. They concluded that semi-circular ribs give better thermal performance than square ribs. Kumar et al. [22] investigated the effects of different shapes and ribs arrangements on heat transfer characteristics in a square duct having ribs attached on opposite walls. Simulation results showed that the ribs with rectangular perforations having rounded corners yield higher thermal performance than that of the other ribs arrangements. Nine et al. [23] studied experimentally the effect of semi-circular ribs attached to the bottom wall of a rectangular channel on turbulence, and the air free flow characteristics through varying the Reynolds number and rib pitch. They established that the heat transfer is positively influenced because of pressure drop and turbulent intensity. Arkan et al. [24] numerically studied a squared steel duct with square ribs through various constant air temperatures. They observed that the provision of ribs leads to an enhancement of the heat transfer rate. Heeyoon et al. [25] carried out an experimental study to examine the thermal performance of a rectangular duct having angled ribs using different aspect ratios with and without an intersecting rib. They observed that the addition of an intersecting rib to angled ribs arrangement enhances the thermal performance for all aspect ratios. Gogulothu et al. [26] and Maradia et al. [27] performed a literature review on the passive methods of heat transfer enhancement techniques for different thermal energy systems. Alenezi et al. [28] carried out a computational study on orthogonal air jet impinging on a smooth, horizontal surface and on a roughened uniformly heated flat plate to study the convective heat transfer in detail. They used circular ribs of square cross-section with different radial positioning centered at the stagnation point as surface roughness elements. The RNG-k epsilon turbulence model was considered for their numerical validation. They concluded that if the rib is placed at a $R/D = 2$ radial location, a maximum enhancement in average Nusselt number is achieved; R being the distance from the centerline in radial direction and D being the nozzle diameter. The authors also concluded from their results of variation in rib height that if the rib height is kept closer to the velocity boundary layer thickness corresponding to the smooth flow without ribs, the average Nusselt number becomes maximum. Zu et al. [29] analyzed the jet impingement heat transfer by carrying out a numerical study using the computational fluid dynamics (CFD) software ANSYS FLUENT. The impingement setup consisted of a round air jet impinging perpendicularly over a smooth plate

placed at a distance H . This distance H was varied from $1D$ to $6D$, D being the nozzle diameter. In order to find the best match with the benchmark experimental data, different turbulence models were tested. Out of the seven models applied, the Shear Stress Transport SST $k-\omega$ and Large Eddy Simulations (LES) models were found to be more accurate in evaluating the fluid flow as well as heat transfer. Since the LES model involves high computational costs, the SST $k-\omega$ model appeared a more economical turbulence model for comparison against experimental results.

There is a lot of published work on the use of surface roughness for the enhancement of heat transfer through impinging jets or radial wall jet flow, as discussed previously. The majority of the related literature focuses on the attached surface roughness elements such as dimples, perforations, grooves, and attached ribs with different orientations, etc. However, fewer studies are found on the use of detached ribs in the flow path of the impinging jets. Brackmann et al. [30] studied detached rib turbulators, but they strictly focused on the impinging jet cooling phenomenon with an array of jets targeting a flat surface. Their work was comprised of experiments; they concluded that the flow field is greatly influenced by the presence of rib elements above the impingement plate. The experimental work was backed by a numerical investigation, and produced an increase in the global Nusselt number by up to 4% compared to the heat transfer results of impinging jets with rib turbulators. Out of the very few publications in the literature, an experimental study involving the use of detached ribs to enhance heat transfer through radial wall jets was published by Katti and Prabhu [31], who developed a correlation for average Nusselt number against the jet Reynolds number. In order to develop this correlation, the authors considered a circular air jet impinging normally on a flat surface with axisymmetric detached circular rib-roughening elements having a rectangular cross-section. Different cases of jet impingement were considered with different Reynolds numbers. Furthermore, different ribs design parameters were tested and the best configuration was established through the results of the local Nusselt number. A continuous increase in the Nusselt number was observed in the stagnation region from the stagnation point, in all cases. A prominent increase in the average Nusselt number ratio for the ribbed and the smooth path was observed along with an increase of Reynolds number. The ratio of average Nusselt numbers of the ribbed and smooth surfaces was seen to increase with Reynolds number. Heat transfer augmentations decrease when larger jet to plate spacing was used.

In the present work, heat transfer and fluid flow characteristics are analyzed in a radial wall jet flow using ribs in a detached configuration. In this configuration, a rib of a rectangular cross-section is mounted over the heated target plate in a ring form with some clearance in between the

ribs and the plate. The ring can be mounted over the plate at two circumferential points with the help of a screw in practical. Here, a radial wall jet denotes a jet moving alongside the impingement plate after striking it, moving in all directions along the impingement plate. Rectangular shaped ribs are used for flow disturbance because these are simple rib shapes, easy to produce and have a high impact on heat transfer, as established from the literature survey. As the optimum nozzle shape is circular, the flow must be hindered with a rib concentric to the nozzle. In most literature dealing with the numerical evaluation of heat transfer and fluid flow characteristics of impinging jets, the k- ϵ and SST k- ω turbulence models are adopted[4], [32], [33]. New turbulence models have however been developed, which predict better turbulence effects and offer more tuning options. In the present work, the Generalized k- ω (GEKO) turbulence model has been selected, and a fine-tuning of the parameters related to fluid flow and heat transfer has been done to achieve optimum results.

2. NUMERICAL METHODOLOGY

2.1. Governing Equations

The present investigation involves the numerical analysis of heat transfer characteristics for a case of jet impingement. A nozzle is utilized to create an air jet with a high velocity, which strikes the orthogonal plate, creating a radial wall jet flow running over the plate. The plate is heated up initially and the objective is to cool it down efficiently. A constant heat flux is being initially applied as a boundary condition to heat up the plate.

The plate is designed with two concentric circular ribs of rectangular cross-section on top of it with some clearance provided amid, as shown in Figure 2. The dimensions of the given geometry are provided in Table 1 with reference to the two-dimensional view of the case. As the flow is axisymmetric, as shown in Figure 3, the fluid flow governing equations can be adopted as a two-dimensional approximation as follows:

$$\frac{\partial u}{\partial x} = 0 \quad (1)$$

$$\rho u \frac{\partial u}{\partial x} = -\frac{\partial P}{\partial x} + \frac{\partial}{\partial x} \left\{ \mu \left(\frac{\partial u}{\partial y} + \frac{\Delta v}{\Delta x} \right) - \rho \overline{u'v'} \right\} \quad (2)$$

$$\rho c_p u \frac{\partial T}{\partial x} = \frac{\partial}{\partial x} \left(\lambda \frac{\partial T}{\partial x} - \rho c_p \overline{u'T'} \right) \quad (3)$$

In the above equations, u and v are the velocity components in the longitudinal x and radial y directions, respectively; P and T are the average pressure and temperature, respectively; u' and v' are the fluctuating velocity components in the axial z and radial r directions, respectively; T' is the fluctuating temperature component; c_p is the specific heat capacity at constant pressure, and λ is the thermal conductivity of the fluid.

Different flow configurations were tested to compare the heat transfer results and establish the configuration achieving a high heat transfer rate from the target plate. Table 2 provides the details of the Jet to Plate Spacing (z/d) parameter being varied to obtain the different design configurations, and study its effect on heat transfer (z being the distance from the nozzle to the plate and d being the nozzle diameter). Numerical simulations were performed with the ANSYS FLUENT CFD software. The computational mesh was refined near both the heated surface and the nozzle using inflation layers to obtain accurate values of velocity and thermal boundary layer, and comply with the requirements of the generalized $k-\omega$ turbulence model, i.e. maintaining a non-dimensional distance at the walls $y^+ \leq 1$.

2.2. Computational Domain and Boundary Conditions

The computational domain is shown in Figure 4 for a basic design configuration with a simplified axisymmetric two-dimensional geometry. The nozzle outlet is located at $z = 2d$ with two ribs placed at a distance $c = 0.067d$ from the impingement plate. The two ribs have a width w and a height e equal to $0.23d$ while the pitch distance between the two ribs is kept at $p = 2e$. Air is introduced into the nozzle with a Reynolds number $Re = 20,000$. This nozzle is long enough to ensure that a fully developed flow is present. The jet strikes the heated plate and moves radially outwards, flowing past the ribs in the wall jet region. A mass flow inlet boundary condition is applied at the nozzle inlet. The supplied air temperature is kept as 300K. An adiabatic wall boundary condition is applied for the nozzle wall. The same condition is applied onto the ribs as no heat is transferred into or out of the ribs. A pressure outlet boundary condition is considered at the domain outlet. A constant heat flux boundary condition $q'' = 1000\text{W/m}^2$ is applied on the impingement plate.

For a compressible flow, it is recommended to apply a mass flow inlet boundary condition rather than the velocity inlet condition, valid merely for incompressible flows. At high Reynolds number,

the air jet flow tends to be compressible in nature so the use of a velocity inlet boundary condition in this situation will produce non-physical results.

For $Re = 20,000$, $d = 15\text{mm}$, and a dynamic viscosity $\mu = 0.00001789\text{ kg/m-s}$, the mass flow rate at the nozzle inlet considered in this work is $\dot{m} = 0.004215\text{ kg/s}$.

The Nusselt number Nu has been used to indicate the heat transfer enhancement:

$$Nu = hd/\lambda \quad (4)$$

with

$$h = q_{conv} / (T_r - T_j) \quad (5)$$

where h is the heat transfer coefficient, d is the exit diameter of the nozzle, λ is the thermal conductivity of air, q_{conv} is the net heat flux convected to the impinging jet, T_j is the jet air temperature, and T_r is the temperature of the target plate at a given radial location.

2.3. Mesh Independency Study

Several meshes have been created with and without rib turbulators, the objective being to obtain a suitable mesh generating accurate and computationally optimized numerical results. As mentioned previously, a fine mesh strategy based on inflation layers was adopted near the walls to obtain accurate results in the wall boundary layer and capture the effect of wall shearing in the wall jet region of the jet impingement system. As the SST $k-\omega$ turbulence model was used, meshes were generated such as the non-dimensional distance at the walls was $y^+ \leq 1$.

The mesh was refined near the rib surfaces in the flow domain, see Figure 5. A coarse mesh was developed first, with a maximum element size of 3mm. The mesh was then refined, decreasing the maximum element size to check improvement in heat transfer results. It was found that there is a considerable improvement for a mesh made of 103,443 elements and no further considerable improvement was obtained when increasing further the number of elements in the mesh, i.e. on going beyond the fine mesh size, the computation time and cost drastically increases with a very minimal increase in the average Nusselt number as can be seen from the sublime case. The mesh comprising 103,443 elements was therefore selected for further studies, as a stable numerical solution was achieved. The corresponding results are tabulated in Table 3.

The above mesh refinement technique was also applied on the smooth flow case for $z/d = 2.0$. A converged average Nu of 66.2 was obtained for a mesh of 97,493 elements for the smooth jet flow

impingement case without ribs after a complete study, see Figure 6. A summary of the grid independence study results is presented in Table 4.

2.4. Turbulence Model Validation

The simulated Nu variations along the jet impingement plate were compared with the experimental data of Katti and Prabhu [31]. Different turbulent models were tested to establish the best match from the computational study. As per the literature, jet impingement phenomena involve wall interactions. Numerical results therefore needed to be focused on the near wall region, and the $k-\omega$ turbulence model predicted results closer to the measured values. There are several variants of the $k-\omega$ model, in addition to transitional flow models. These models have therefore been tested here for a simple flow with and without ribs. As shown in Figure 7, the SST $k-\omega$ and Transitional $k-kl-\omega$ models under-predict the Nu values while the standard and baseline $k-\omega$ models over-predict the results. The generalized $k-\omega$ model shows the best agreement with the experimental values.

The corresponding equations of the generalized $k-\omega$ model for the turbulent kinetic energy k and the specific turbulence dissipation rate ω based on Menter's work [34] are provided below.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho v k)}{\partial r} = P_k - \rho C_\mu k \omega + \frac{\partial}{\partial r} \left\{ \left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial(k)}{\partial r} \right\} \quad (6)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho v \omega)}{\partial r} = C_{\omega 1} F_1 \frac{\omega}{k} P_k - C_{\omega 2} F_2 \rho \omega^2 + F_3 \frac{2}{\sigma_\omega} \frac{\rho}{\omega} \frac{\partial k}{\partial r} \frac{\partial \omega}{\partial r} + \frac{\partial}{\partial r} \left\{ \left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial(\omega)}{\partial r} \right\} \quad (7)$$

where

$$P_k = -\tau_{ij} \frac{\partial u}{\partial r} \quad (8)$$

$$\tau_{ij}^{EV} = -\rho \overline{u'v'} = \mu_t \cdot 2S_{ij} - \frac{2}{3} \rho k \delta_{ij} \quad (9)$$

$$\tau_{ij} = \tau_{ij}^{EV} - C_{CORNER} \frac{1.2\mu_t}{\max(0.3\omega, \sqrt{0.5(S_z + \Omega_z)})} (S_{ik}\Omega_{kj} - \Omega_{ik}S_{kj}) \quad (10)$$

$$v_t = \frac{\mu_t}{\rho} = \frac{k}{\max\left(\omega, \frac{S}{C_{Real}}\right)} \quad (11)$$

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u}{\partial r} + \frac{\partial v}{\partial r} \right) \quad (12)$$

$$S = \sqrt{2 \times S_{ij} \cdot S_{ij}} \quad (13)$$

$$\Omega_{ij} = \frac{1}{2} \left(\frac{\partial u}{\partial r} - \frac{\partial v}{\partial r} \right) \quad (14)$$

$$\Omega = \sqrt{2 \times \Omega_{ij} \cdot \Omega_{ij}} \quad (15)$$

$$\tilde{P}_k = \min(P_k, C_{pklim} \rho k) \quad (16)$$

All terms above are defined in the Nomenclature section, but S represents the variable of shear strain rate, and the different constants are such as $C_{Real} = \frac{1}{\sqrt{3}} = 0.577$, $C_{CORNER} = 0$, $C_\mu = 0.09$, $C_{\omega 1} = 2.0$, $C_{\omega 2} = 1.168$, $C_{pklim} = 10$. Note that F_1 , F_2 and F_3 are functions, which can vary depending on the value of coefficients contained in those functions, the free coefficients of the GEKO model.

The equations for k and ω containing the functions F_1 , F_2 and F_3 can be fine-tuned to obtain more accurate results for specific applications. To change the functions, the values of the associated GEKO coefficients should therefore be modified. As recommended in the ANSYS FLUENT users guide [35], three parameters are usually enough to control the flow scheme and fine-tune the model equations for jet flows with wall interactions. These parameters are the separation parameter C_{sep} , the near wall parameter C_{NW} , and the jet parameter C_{jet} . The separation parameter as the name suggests is meant for predicting the flow separation levels in a fluid flow problem. This parameter along with the near-wall parameter controls the behavior of fluid around the obstacles and walls. The utilization of higher values of C_{sep} sensitizes the flow to adverse pressure gradients for boundary layers, especially by a reduction in the values of eddy viscosity. The inside regions of wall boundary layers are greatly affected by the variation of the near-wall parameter. This constant has a moderate effect on vorticity generation and spread. Finally, increasing the jet parameter decreases the spreading rate of jets and hence the axial velocity of free jet is maintained and the shearing of the outer layer of the free jet is reduced.

Different parameter values have been tested to obtain the simulated heat transfer to agree with the experimental data, based on the recommendations from the ANSYS FLUENT user guide. The most suitable parameter values established for this work are summarized in Table 5. Note that in addition to C_{jet} , another GEKO model constant, C_{jet_aux} was also modified from 2.0 (default value) to 4.0; this improved the numerical maximum heat transfer results.

Similarly, flow simulations were performed with all turbulence models for the ribbed flow case to establish whether the newly adopted generalized k - ω model performs well in the presence of higher turbulence. Figure 8 shows the corresponding results. It appears clearly that GEKO model curve matches well the experimental results. As can also be seen, the SST k - ω model fails to work, this might be due to its large sensitivity at the wall and hence, might not be adequate in the region of free shear flow as it over-predicts separation layers. In comparison, the optimized generalized k - ω model provides more control on the separation allowance according to the prevailing condition. The results obtained here agree with the literature where it has been established that numerical results obtained with the GEKO model are usually more accurate than with the SST k - ω model.

3. RESULTS AND DISCUSSIONS

3.1. Heat Transfer Results

Numerical data obtained in this research work were compared with the experimental data of Katti and Prabhu [31]. The initial validation was performed for the turbulence model selection in Section 2.4. Different parameters also need to be investigated to assess their effect on the temperature distribution and overall heat transfer. These parameters include the roughness elements involved in the flow domain and the fluid flow properties. Geometric parameters of the roughening elements (ribs) include the number of ribs, their placement and accurate radial positioning and their size. The shape of the ribs remains the same throughout this work, they are of rectangular shape as described previously, but their width and height can vary; this might affect the flow and heat transfer results. The radial distance in between the ribs termed as ‘rib pitch’ is also an important parameter. All these parameters may play a significant role on the flow properties.

Another factor to be accounted for is the distance from the nozzle to the impingement plate, as a constant flux should be obtained to achieve the best heat transfer results. The above-mentioned factors were tested numerically one by one with different jet to plate spacing configurations. The overall average Nusselt numbers obtained for all configurations were compared for different Reynolds number values.

3.1.1. Rib Clearance Effect

To study how the provision of clearance between the ribs and the impingement wall impacts the radial heat transfer distribution, four different cases have been investigated with varied rib clearance ratios $c/d = 0.033, 0.067, 0.1$ and 0.133 , rib height $e/d = 0.23$, rib width $w/d = 0.23$, pitch distance between two ribs $p/e = 4$. The location of the inner rib $r_1/d = 0.5$ remained constant. Figure 9 shows the local distribution of the simulated Nu for the various clearance heights, which are compared with the experimental results of Katti and Prabhu [29].

As the clearance value decreases, the heat transfer increases near the first rib, as indicated by the increase in Nu . In addition, there is a drop in heat transfer in the region close to the back of the rib wall, this indicates that there is a disruption in fluid motion behind this rib i.e., a flow separation, followed by a recirculation due to pressure drop in the near regions ahead. The presence of a rib at a particular location inside the fluid flow channel enhances the heat transfer locally. Based on Figure 9, it can be concluded that heat transfer is best achieved in the case of ribs clearance equal to $0.067d$.

3.1.2. First Rib Radial Distance Effect

As seen in the cases of varied rib clearance, the $c/d = 0.067$ distance showed better performance. This value is thus set as a constant along with the other rib design constant parameters, rib pitch, height and width. The fluid intake is also set at $Re = 20,000$.

The placement of the first rib is now varied. For the first case, the rib is placed at a distance such as its inner surface is in line with the nozzle periphery. Then the distance is decreased and the ribs are moved inward along the axial direction. Hence, a larger part of the rib upper surface becomes exposed to the direct stream of fluid.

Numerical calculations proved that heat is better transferred if the radial distance of the first rib is kept at a distance equal to 0.33 times the nozzle diameter, as seen in Figure 10. When the radial distance is increased, the rate of heat transfer decreases. If the local heat transfer is to be increased, the rib should therefore be placed within the radius of the nozzle with $r_1/d = 0.33$. In addition, the comparison with the impingement over smooth surface shows that the ribs efficiently increase the heat transfer as the curve for Nu value (for rib cases) rises above the Nu -based curve obtained for cases without ribs.

3.1.3. Rib Width Effect

Based on the previous results, $0.067d$ was found the best clearance value, along with a $0.33d$ first rib radial distance, for optimum heat transfer in the region in-between $r/d = 0$ and $r/d = 2$. The

width of the ribs is now modified to evaluate the optimum rib design for high heat transfer rate in the selected region along the impingement plate.

Figure 11 shows that the heat transfer for $w/d = 0.23$ is best for different jet to plate spacing, compared to a width of $0.13d$. This can be explained by the fact that the rib with greater width blocks a higher quantity of the impinging fluid and disperses the fluid more before reaching the plate, as can clearly be observed in the velocity contour plots. If a larger width is used, more turbulence will appear in areas close to the stagnation region, this will result in an increase of the local heat transfer coefficient.

3.1.4. Rib Pitch Effect

The optimum parameters established so far are $c/d = 0.067$, $r_1/d = 0.33$ and $w/d = 0.23$, these values are used in the next cases with a variable jet to plate spacing, where the distance between two ribs in the radial direction is varied. In the first cases, $p/e = 4.0$, which was the value used in previous cases. This distance is decreased to $p/e = 2.0$ and the corresponding Nu values are plotted against the impingement plate. The second rib is then moved to a very large distance $p/e = 8.0$.

Looking closely at Figure 12, it appears that the heat transfer increases significantly when the second rib is placed at a high radial distance, so at a location where the air flow is not perturbed much. The presence of the second rib greatly reduces the flow of air stream, reducing the temperature gradients so the value of Nu reduces in the region near the first rib. Based on the results, it appears that having a single rib is a better option than having multiple ribs in order to enhance the heat transfer rate.

3.1.5. Rib Height Effect

Another parameter that can greatly influence heat transfer is the rib height. Results obtained when varying this height are shown in Figure 13. As the rib height is increased, the primary peak value increases but a downtrend in Nu is present just ahead of the rib. This indicates that there is an increased flow separation due to the flow blockage created by the rib. In these conditions, the Nu values fall below the values obtained for the smooth flow case.

As the rib height is reduced from $e/d = 0.23$ to $e/d = 0.13$, the flow interruption is reduced, hence the Nu peak value is also decreased. A rib height of $e/d = 0.23$ appears suitable for this work as an increased Nu peak value is present and the Nu curve does not reduce below the Nu curve obtained for the smooth flow case, when moving away from the rib.

3.2. Nusselt Number Variation Over The Impingement Plate

Nu is now plotted over the impingement plate to understand how the heat transfer occurs in the radial wall jet flow direction, downstream of the stagnation region. Simulations based on previously established optimum parameters, are shown in Figure 14, where the right hand side describes the highest heat transfer inside the stagnation region where a continuous impingement of cool air jet is generated over the heated plate (simple jet impingement, no roughening elements over the impingement plate).

A secondary peak occurs at $r/d = 1.25$, adjacent to the wall in the wall jet region close to the stagnation region, where the turbulent kinetic energy is the highest. The Nu trends established through the numerical investigation agree with the conclusions from Narayanan et al. [36] for a case of slot jet impingement heat transfer. Narayanan et al. found that the local Nu has high values wherever turbulence in the wall jet region on the outward side is high, and this was not the case for high turbulence in the near-wall segment of the radial wall jet. They specifically pointed out that on the outside region of the wall jet stagnation point, an unsteadiness was present in the thermal boundary layer owing to the presence of the outer region of turbulence.

The rate of heat transfer decreases as the nozzle is moved away from the target plate i.e. when z/d is higher. This can be clearly seen from the Nu contours plots of the smooth flow. When a rib is present, the situation is different and the maximum heat transfer position moves radially outwards. As the fluid momentum is maximum beneath the ribs, a very high heat transfer occurs at the same position because of momentum diffusivity. Ahead of the ribs, flow separation occurs and therefore heat transfer decreases momentarily. It then rises again when the flow is reattached and momentum is regained. A secondary peak is then present in the local Nu values when the fluid moves radially outwards.

If the Nu contours plot for the flow involving rib turbulators is considered, it can be seen that the maximum heat transfer is obtained when the nozzle-plate spacing is minimum. The heat transfer rate then gradually decreases as the nozzle is placed at distances greater than the nozzle diameter. This is due to the fluid striking with a maximum axial velocity and with the least momentum lost to the surroundings. This also means that less shearing of the fluid layers occurs with minimum jet axial velocity decay in the free jet region, and the jet strikes the plate and imparts maximum momentum which is then turned radially outwards.

After turning orthogonally, the wall jet accelerates to conserve its momentum. The jet spreads smoothly over the entire target plate with the minimum boundary layer thickness. High velocity layers are then present at a minimum distance from the heated plate. As a result, maximum thermal diffusivity is observed.

3.3. Flow Visualization

The resulting trends of the heat transfer parameter can be better explained if the flow behavior of fluid inside the system is known. The air velocity contours have therefore been plotted for all the jet to plate spacing configurations up to a dimensionless radial distance of 3.0, see Figure 15. Velocity contours for smooth flow cases without ribs have also been included to compare the fluid behavior in the system with and without ribs.

For all results, a clear stagnation zone is present at $r/d=0$, which is indicated by the blue region (zero velocity). Such stagnation region appears because of the sudden decrease in axial air velocity due to the momentum transfer and building up of a high static pressure region from the continuous striking of fluid particles.

When looking at the smooth-based case of jet impingement, there is a clear increase of the jet layer total height above the wall, in the wall jet region, as the nozzle is moved away from the impingement plate. However, there is no or a very slight change in the size of the stagnation region. This stagnation region is larger when the nozzle is closely placed ahead of the heated target plate. It can be also observed that the fluid particles, i.e. air molecules, have their radial velocity component increased in case of a larger spacing in between the plate and the nozzle due to a more prominent decay of the axial velocity in the free jet region. Hence, a greater spread rate of fluid particles is visible, which explains that the wall jet thickness increases.

In all jet to plate spacing cases, the fluid velocity is highly affected by the presence of the ribs. As can be seen in the contours plots, a higher fluid velocity is present, as indicated by the red color, just beneath the rib above the wall. This is due to the narrowing of the path and the velocity continuously remaining enhanced, even beyond the rib. The velocity boundary layer is very thin in this region, but because of the presence of rib as a hindrance, the flow is disrupted and the fluid layers become detached. There is low pressure region which forms just behind the rib because the fluid gets disturbed and a wake region builds-up, as can be seen in the blue color just ahead of the rib.

The fluid particles then reattach in downstream regions but because of the lower pressure, vortices are generated and the turbulence increases in this region and the in the region ahead, which is the wall jet region. The turbulence intensity increases in these regions because of the wall shearing and because of the rib.

3.4. Reynolds Number Effect

The fluid kinetic energy will have a positive effect on the flow distribution inside the fluid domain so it is expected that the Reynolds number Re will also have an impact on the heat transfer through the momentum diffusivity. To establish the effect of the jet Re on the heat transfer rate, numerical simulations were performed with the final case scenario for a single rib having a width of $w/d = 0.23$, a height $e/d = 0.23$, a clearance $c/d = 0.067$, located at a radial position $r_1/d = 0.33$.

The heat transfer coefficient is now defined as the average Nu along the impingement plate over a radial distance $r/d = 2.0$. The average Nu determined for all the z/d cases with smooth jet impingement are then compared with the average Nu values of the ribbed plate jet impingement case, see Figure 16. The first graph in Figure 16 shows the average Nu results for all cases of jet to plate spacing i.e. 0.5, 1.0, 2.0, 3.0, 4.0 and 6.0 for $Re = 20,000$ for the ribbed flow and the smooth flow. The second graph shows the same results but for $Re = 15,000$.

To establish how the heat transfer rate increases through the use of rib turbulators, the ratio of Nu averaged up to a radial distance of twice the nozzle diameter for the ribbed and smooth flows, are considered for different jet to plate spacings. The corresponding results are plotted in Figure 17 for $Re = 20,000$ and $Re = 15,000$. It is evident from these graphs, that there is an increase in heat transfer rate for varying values of the jet Reynolds number. The increase in heat transfer rate is highest for the cases in which nozzle is more closer to the target plate and if the gap z is increased, the gain is reduced. The reason behind this very obvious increase is that increase in Reynolds number augments the momentum and as the momentum increases so there will more dispersion of the fluid layers at the stagnation region resulting in more turbulence and heat transfer.

Figure 18 shows the numerical heat transfer results, which are compared to experimental data. For all cases of normalized jet to plate spacing z/d , the increase in Reynolds number increases the ratio of average Nu for ribbed and smooth flows which indicated that the rate of heat transfer enhancement increases significantly. The simulated heat transfer agrees well with the experimental results, and only a slight over-prediction of the numerical heat transfer is present. However, the

resulting trends are identical, which shows that irrespective of the Re value, the heat transfer rate decreases when the nozzle is moved away from the impingement plate. Note that there are some fluctuations in the experimental results which might depict measurement errors, as highlighted in Katti and Prabhu [31]. Moreover, there is uncertainty in the provision of turbulence intensity locally in the experimental setup. All these reasons can explain the slight differences between the numerical and the experimental results.

Overall, the numerical results show that a maximum of 64% heat transfer enhancement is achieved with a jet to plate spacing half of the nozzle diameter, with a minimum length of the free jet region, and hence a minimum decay in kinetic energy.

4. CONCLUSION

An air jet impingement cooling technique was used to cool down a plate at constant heat flux and for detached rib turbulators present above its surface. Presence of such detached ribs increases the turbulence and hence the heat transfer augments by 40% on average for the different jet to plate spacing investigated. Different rib configurations were considered and conclusions can be drawn from the numerical study. A $c/d = 0.067$ rib clearance has been shown to be optimum for the development of the thermal boundary layer. To obtain maximum turbulence using ribs, the first rib should be placed at a radial distance around $0.33d$. The width of a rib should be $0.23d$ so as to obtain a maximum flow separation and a narrowing of the velocity boundary layer which ultimately results in a good heat transfer. It has also been identified that one rib is enough to provide maximum turbulence in a region where the fluid has maximum momentum in the radial direction, along the heated plate. The presence of a second rib blocks the flow and causes a decrease in momentum while the heat transfer locally drops in between the ribs. It has also been shown that a $0.23d$ rib height is optimum, suggesting that a square rib should be used.

Different nozzle configurations were adopted. It has been established that the optimum configuration is obtained when the nozzle is close to the impingement plate, thus imparting maximum momentum to the flow. The average Nusselt number has been found the highest when the nozzle was placed at a distance half of its diameter from the target plate. As the nozzle was moved away from the impingement surface, the heat transfer rate gradually decreased.

Finally, the impact of the Reynolds number was studied for its effect on the heat transfer enhancement through varying the flow conditions at the nozzle inlet. As the Reynolds number increased, heat transfer enhancement was also shown to increase.

ACKNOWLEDGEMENT

The authors want to express their gratitude to the MED, Khwaja Fareed University of Engineering & Information, Rahim Yar Khan, for providing the software package, and to NED University of Engineering & Technology, Karachi, for providing the opportunity to work in this particular field of research.

NOMENCLATURE

<i>A</i>	Hydraulic area of flow in mm ²
<i>c</i>	Clearance between the ribs and the impingement wall in millimeters
<i>c_p</i>	Specific heat capacity in J/kg.K
<i>d</i>	Nozzle diameter (hydraulic diameter) in millimeters
<i>e</i>	Height of rib in millimeters
<i>h</i>	Convective heat transfer coefficient in W/m ² .K
<i>I</i>	Turbulence intensity (%)
<i>k</i>	Turbulent kinetic energy in J/kg
<i>l</i>	Nozzle length in millimeters
<i>L</i>	Length of the impingement plate in millimeters
<i>m</i>	Mass flow rate of fluid in kg/s
<i>Nu</i>	Local Nusselt number
<i>Nu_{avg}</i>	Average Nusselt number
<i>Nu_{avg)o}</i>	Average Nusselt number for smooth flow
<i>P</i>	Production of energy
<i>p</i>	Pitch distance between ribs in millimeters
<i>q</i>	Heat flux in W/m ²
<i>Q</i>	Flow rate in m ³ /s
<i>r</i>	Radial distance from stagnation point in millimeters
<i>r₁</i>	First rib radial distance from the nozzle axis in millimeters

Re	Reynolds number
S	Strain rate tensor
T_j	Air jet temperature in Kelvins
T_r	Temperature of the target plate at given radial location in Kelvins
u	Axial velocity component in m/s
u'	Fluctuating velocity component in axial direction in m/s
v	Fluid velocity component in radial direction in m/s
v'	Fluctuating velocity component in radial direction in m/s
w	Width of rib in millimeters
x	Distance from the stagnation point in the axial direction in millimeters
y	Perpendicular Distance from the stagnation point along the length of the Impingement plate in millimeters
z	Nozzle-plate spacing in millimeters

Greek Symbols

ω	Specific turbulent energy dissipation rate in s^{-1}
λ	Thermal conductivity in W/m.K
μ	Dynamic viscosity in kg/m.s
ν	Kinematic viscosity of fluid in m^2/s
ρ	Fluid density as defined by ideal gas law in kg/m^3
τ	Shear stress in N/m^2
Ω	Vorticity in s^{-1}

SUBSCRIPTS

avg	Average
i	Component in x-direction
j	Jet
k	Kinetic energy
NW	Near wall
o	For smooth flow

<i>p</i>	Constant pressure
<i>sep</i>	Separation

SUPERSCRIPTS

'	Fluctuation of quantity
---	-------------------------

ABBREVIATIONS

<i>CFD</i>	Computational fluid dynamics
<i>GEKO</i>	Generalized k-omega
<i>LES</i>	Large eddy simulation
<i>PIV</i>	Particle image velocimetry
<i>RANS</i>	Reynolds averaged Navier Stokes
<i>SST</i>	Shear stress transport

REFERENCES

- [1] N. Zuckerman and N. Lior, “Jet Impingement Heat Transfer : Physics , Correlations , and Numerical Modeling,” vol. 39, no. 06, pp. 565–631, 2006, doi: 10.1016/S0065-2717(06)39006-5.
- [2] D. Singh, B. Premachandran, and S. Kohli, “Effect of nozzle shape on jet impingement heat transfer from a circular cylinder,” *Int. J. Therm. Sci.*, vol. 96, pp. 45–69, 2015, doi: 10.1016/j.ijthermalsci.2015.04.011.
- [3] M. F. Koseoglu and S. Baskaya, “The role of jet inlet geometry in impinging jet heat transfer, modeling and experiments,” *Int. J. Therm. Sci.*, vol. 49, no. 8, pp. 1417–1426, 2010, doi: 10.1016/j.ijthermalsci.2010.02.009.
- [4] M. U. Siddique, A. Syed, S. A. Khan, and J. P. Meyer, “On numerical investigation of heat transfer augmentation of flat target surface under impingement of steady air jet for varying heat flux boundary condition,” *J. Therm. Anal. Calorim.*, vol. 147, no. 6, pp. 4325–4337, 2022, doi: 10.1007/s10973-021-10785-4.
- [5] D. Zhang, H. Qu, J. Lan, J. Chen, and Y. Xie, “International Journal of Heat and Mass Transfer Flow and heat transfer characteristics of single jet impinging on protrusioned surface,” *HEAT MASS Transf.*, vol. 58, no. 1–2, pp. 18–28, 2013, doi:

- 10.1016/j.ijheatmasstransfer.2012.11.019.
- [6] W. S. Kim and S. Y. Lee, "Behavior of a water drop impinging on heated porous surfaces," *Exp. Therm. FLUID Sci.*, vol. 55, pp. 62–70, 2014, doi: 10.1016/j.expthermflusci.2014.02.023.
 - [7] A. H. Beitelmal, M. A. Saad, and C. D. Patel, "Effects of surface roughness on the average heat transfer of an impinging air jet," *Int. Commun. Heat Mass Transf.*, vol. 27, no. 1, pp. 1–12, 2000, doi: 10.1016/S0735-1933(00)00079-8.
 - [8] M. A. R. Sharif and N. M. Ramirez, "Surface roughness effects on the heat transfer due to turbulent round jet impingement on convex hemispherical surfaces," *Appl. Therm. Eng.*, vol. 51, no. 1–2, pp. 1026–1037, 2013, doi: 10.1016/j.applthermaleng.2012.10.015.
 - [9] Y. Xing and B. Weigand, "Experimental investigation of impingement heat transfer on a flat and dimpled plate with different crossflow schemes," *Int. J. Heat Mass Transf.*, vol. 53, no. 19–20, pp. 3874–3886, 2010, doi: 10.1016/j.ijheatmasstransfer.2010.05.006.
 - [10] L. A. Gabour, "Wall Roughness Effects on Stagnation-Point Heat Transfer Beneath an Impinging Liquid Jet," vol. 116, no. February 1994, 2016.
 - [11] N. Celik, "Effects of the surface roughness on heat transfer of perpendicularly impinging co-axial jet," pp. 1209–1217, 2011, doi: 10.1007/s00231-011-0785-9.
 - [12] S. Caliskan, "Flow and heat transfer characteristics of transverse perforated ribs under impingement jets," *Int. J. Heat Mass Transf.*, vol. 66, pp. 244–260, 2013, doi: 10.1016/j.ijheatmasstransfer.2013.07.027.
 - [13] K. Kanokjaruvijit and R. F. Martinez-Botas, "Jet impingement on a dimpled surface with different crossflow schemes," *Int. J. Heat Mass Transf.*, vol. 48, no. 1, pp. 161–170, 2005, doi: 10.1016/j.ijheatmasstransfer.2004.08.005.
 - [14] E. Y. Choi, Y. D. Choi, W. S. Lee, J. T. Chung, and J. S. Kwak, "Heat transfer augmentation using a rib-dimple compound cooling technique," *Appl. Therm. Eng.*, vol. 51, no. 1–2, pp. 435–441, 2013, doi: 10.1016/j.applthermaleng.2012.09.041.
 - [15] L. A. El-Gabry and D. A. Kaminski, "Experimental investigation of local heat transfer distribution on smooth and roughened surfaces under an array of angled impinging jets," *J. Turbomach.*, vol. 127, no. 3, pp. 532–544, 2005, doi: 10.1115/1.1861918.
 - [16] J.-J. Hwang, "Heat Transfer-Friction Characteristic Comparison in Rectangular Channel Arrays of Attached, Detached, and Alternate Attached-Detached Ribs on Two Opposite Walls." Jun. 02, 1997, doi: 10.1115/97-GT-488.
 - [17] G. Tanda, "Effect of rib spacing on heat transfer and friction in a rectangular channel with 45° angled rib turbulators on one/two walls," *Int. J. Heat Mass Transf.*, vol. 54, no. 5–6, pp. 1081–1090, 2011, doi: 10.1016/j.ijheatmasstransfer.2010.11.015.
 - [18] L. Tan, J. Z. Zhang, and H. S. Xu, "Jet impingement on a rib-roughened wall inside semi-

- confined channel,” *Int. J. Therm. Sci.*, vol. 86, pp. 210–218, 2014, doi: 10.1016/j.ijthermalsci.2014.06.037.
- [19] K. Wongcharee, W. Changcharoen, and S. Eiamsa-Ard, “Numerical investigation of flow friction and heat transfer in a channel with various shaped ribs mounted on two opposite ribbed walls,” *Int. J. Chem. React. Eng.*, vol. 9, 2011, doi: 10.1515/1542-6580.2560.
- [20] F. Wang, J. Zhang, and S. Wang, “Investigation on flow and heat transfer characteristics in rectangular channel with drop-shaped pin fins,” *Propuls. Power Res.*, vol. 1, no. 1, pp. 64–70, 2012, doi: 10.1016/j.jprr.2012.10.003.
- [21] H. S. Chung, G. H. Lee, M. J. Nine, K. Bae, and H. M. Jeong, “Study on the thermal and flow characteristics on the periodically arranged semi-circular ribs in a rectangular channel,” *Exp. Heat Transf.*, vol. 27, no. 1, pp. 56–71, 2014, doi: 10.1080/08916152.2012.719067.
- [22] A. Kumar and R. Saravanasathiyaprabhahar, “Numerical Investigation of Heat Transfer Characteristics in A Square Duct with Internal,” vol. 3, no. 1, pp. 1179–1185, 2014.
- [23] J. Nine, G. H. Lee, H. S. Chung, M. Ji, and H. Jeong, “Turbulence and pressure drop behaviors around semicircular ribs in a rectangular channel,” *Therm. Sci.*, vol. 18, no. 2, pp. 419–430, 2014, doi: 10.2298/TSCI111022142N.
- [24] A. K. AL-Taie and M. J. Kadham, “Numerical Heat Transfer Enhancement in Square Duct With Internal Rib,” *Numer. Heat Transf.*, no. 3, pp. 87–100, 2015.
- [25] H. Chung, J. S. Park, S. Park, S. M. Choi, D. H. Rhee, and H. H. Cho, “Augmented heat transfer with intersecting rib in rectangular channels having different aspect ratios,” *Int. J. Heat Mass Transf.*, vol. 88, pp. 357–367, 2015, doi: 10.1016/j.ijheatmasstransfer.2015.04.033.
- [26] R. Gugulothu, K. V. K. Reddy, N. S. Somanchi, and E. L. Adithya, “A Review on Enhancement of Heat Transfer Techniques,” *Mater. Today Proc.*, vol. 4, no. 2, pp. 1051–1056, 2017, doi: 10.1016/j.matpr.2017.01.119.
- [27] C. Maradiya, J. Vadher, and R. Agarwal, “The heat transfer enhancement techniques and their Thermal Performance Factor,” *Beni-Suef Univ. J. Basic Appl. Sci.*, vol. 7, no. 1, pp. 1–21, 2018, doi: 10.1016/j.bjbas.2017.10.001.
- [28] A. H. Alenezi, A. Almutairi, H. M. Alhajeri, A. Addali, and A. A. A. Gamil, “Flow structure and heat transfer of jet impingement on a rib-roughened flat plate,” *Energies*, vol. 11, no. 6, 2018, doi: 10.3390/en11061550.
- [29] Y. Q. Zu, Y. Y. Yan, and J. Maltson, “Numerical study on stagnation point heat transfer by jet impingement in a confined narrow gap,” *J. Heat Transfer*, vol. 131, no. 9, pp. 1–4, 2009, doi: 10.1115/1.3139183.
- [30] R. Brakmann, L. Chen, R. Poser, J. Rodriguez, M. Crawford, and B. Weigand, “Heat transfer investigation of an array of jets impinging on a target plate with detached ribs,” *Int.*

- J. Heat Fluid Flow*, vol. 78, no. May, p. 108420, 2019, doi: 10.1016/j.ijheatfluidflow.2019.05.009.
- [31] V. Katti and S. V. Prabhu, “Heat transfer enhancement on a flat surface with axisymmetric detached ribs by normal impingement of circular air jet,” *Int. J. Heat Fluid Flow*, vol. 29, no. 5, pp. 1279–1294, 2008, doi: 10.1016/j.ijheatfluidflow.2008.05.003.
- [32] A. Hadipour, M. Rajabi, and Z. Maziar, “Effect of micro - pin characteristics on flow and heat transfer by a circular jet impinging to the flat surface,” *J. Therm. Anal. Calorim.*, 2020, doi: 10.1007/s10973-019-09232-2.
- [33] A. Hadipour, M. R. Zargarabadi, and J. Mohammadpour, “Effects of a triangular guide rib on flow and heat transfer in a turbulent jet impingement on an asymmetric concave surface Effects of a triangular guide rib on flow and heat transfer in a turbulent jet impingement on an asymmetric concave surface,” vol. 075112, 2020, doi: 10.1063/5.0010480.
- [34] a Hellsten, “k-omega SST Turbulence Model 29th AIAA Fluid Dynamics Conference,” *AIAA Fluid Dyn. Conf.*, pp. 2–12, 1998.
- [35] F. R. Menter, R. Lechner, and A. Matyushenko, “Best Practice : Generalized k- w Two-Equation Turbulence Model in ANSYS CFD (GEKO),” pp. 1–38, 2019.
- [36] V. Narayanan, J. Seyed-Yagoobi, and R. H. Page, “An experimental study of fluid mechanics and heat transfer in an impinging slot jet flow,” *Int. J. Heat Mass Transf.*, vol. 47, no. 8–9, pp. 1827–1845, 2004, doi: 10.1016/j.ijheatmasstransfer.2003.10.029.

Table Captions List

Table 1: Design parameter values considered for the nozzle and ribs

Table 2: Jet-to-plate spacing values considered as a parameter for its effect on heat transfer

Table 3: Summary of mesh independence study for the smooth flow case without ribs

Table 4: Summary of the mesh independence study for the ribbed channel case-1

Table 5: Controlling parameters of GEKO turbulence model

Table 1: Design parameter values considered for the nozzle and ribs

Dimension Property	Notation	Values			
Nozzle Diameter	D	15 mm			
Impingement Plate Length	L	160 mm (in total)			
Clearance Ratio	c/d	0.03	0.067	0.1	0.133
Width of Rib Ratio	w/d	0.13		0.23	
Rib Height Ratio	e/d	0.13	0.23	0.33	0.4
Rib Pitch Ratio	p/e	2.0	4.0	∞	
First Rib Location	r_1/d	0.5	0.4	0.33	
Reynold's Number	Re	20,000		15,000	

Table 2: Jet-to-plate spacing values considered as a parameter for its effect on heat transfer

Dimension Property	Notation	Values
Jet to Plate Spacing	z/d	0.5
		1.0
		2.0
		3.0
		4.0
		6.0

Table 3: Summary of the mesh independence study for the ribbed channel case-1

Mesh	Mesh number	y^+	$y^+_{\max}$	Nu_{avg}
coarse	42,384	0.27462	0.742948	66.7806
medium	68,779	0.275273	0.7504	66.8227
fine	103,443	0.236607	0.487106	70.2393
sublime	139,284	0.239142	0.449145	70.5542

Table 4: Summary of the mesh independence study for the smooth flow case without ribs

Mesh	Mesh number	y^+	$y^+_{\max}$	Nu_{avg}
coarse	9422	0.3942637	0.89687014	63.09267
medium	35,422	0.497154	0.911695	65.7767
fine	97,493	0.500114	0.916204	66.2412

Table 5: Controlling parameters of GEKO turbulence model

Controlling Parameter	Values
Separation Parameter, C_{sep}	2.0
Near wall Parameter, C_{NW}	0.5
Jet Parameter, C_{jet}	0.9

Figure Captions List

- Figure 1: Jet impingement schematic diagram
- Figure 2: Jet impingement flow case
- Figure 3: Geometry of axisymmetric detached rib with jet impingement
- Figure 4: Schematic flow diagram with inlet & boundary conditions
- Figure 5: Mesh refinement details (a)- complete axisymmetric geometry mesh (b)- mesh refinement at nozzle wall (c)- mesh refinement near the heated plate (d)- mesh refinement at rib wall
- Figure 6: Mesh independence study for surface Nusselt number
- Figure 7: Turbulence model validation for smooth flow impingement (without ribs)
- Figure 8: Turbulence model validation for jet impingement case involving rib turbulators
- Figure 9: Nusselt number plots for heat transfer study with different clearance ratios at $p/e = 4.0$, $e/d = w/d = 0.23$, $r_1/d = 0.5$ & $Re = 20,000$
- Figure 10: Nusselt number plot for heat transfer study with different radial distances of first rib, $p/e = 4.0$, $e/d = w/d = 0.23$, $c/d = 0.067$ & $Re = 20,000$
- Figure 11: Nusselt number plot for heat transfer study with different rib widths, $r_1/d = 0.33$, $p/e = 4.0$, $e/d = 0.23$, $c/d = 0.067$ & $Re = 20,000$
- Figure 12: Nusselt number plot for heat transfer study with different pitch distances of ribs, $r_1/d = 0.33$, $e/d = 0.23$, $w/d = 0.23$, $c/d = 0.067$ & $Re = 20,000$
- Figure 13: Nusselt number plot for heat transfer study with different rib heights, $r_1/d = 0.33$, $p/e = 4$, $w/d = 0.23$, $c/d = 0.067$ & $Re = 20,000$
- Figure 14: Nusselt number contours over the impingement plate up to the radial location of $r/d = 2.0$; Rib cases on left side and Smooth Surface without ribs on right side
- Figure 15: Velocity contours in the axisymmetric region (free & wall jet) up to the radial location of $r/d = 3.0$; Rib cases on left side & respective cases of Smooth Surface without ribs on right side
- Figure 16: Comparison of average Nusselt number (ribbed vs smooth surface) up to $r/d = 2.0$ a) $Re = 20,000$; b) $Re = 15,000$
- Figure 17: Average gain in heat transfer as a ratio of average Nusselt numbers for ribbed & smooth surfaces up to $r/d = 2.0$ a) $Re = 20,000$; b) $Re = 15,000$
- Figure 18: Effect of Reynolds number variance on heat transfer enhancement

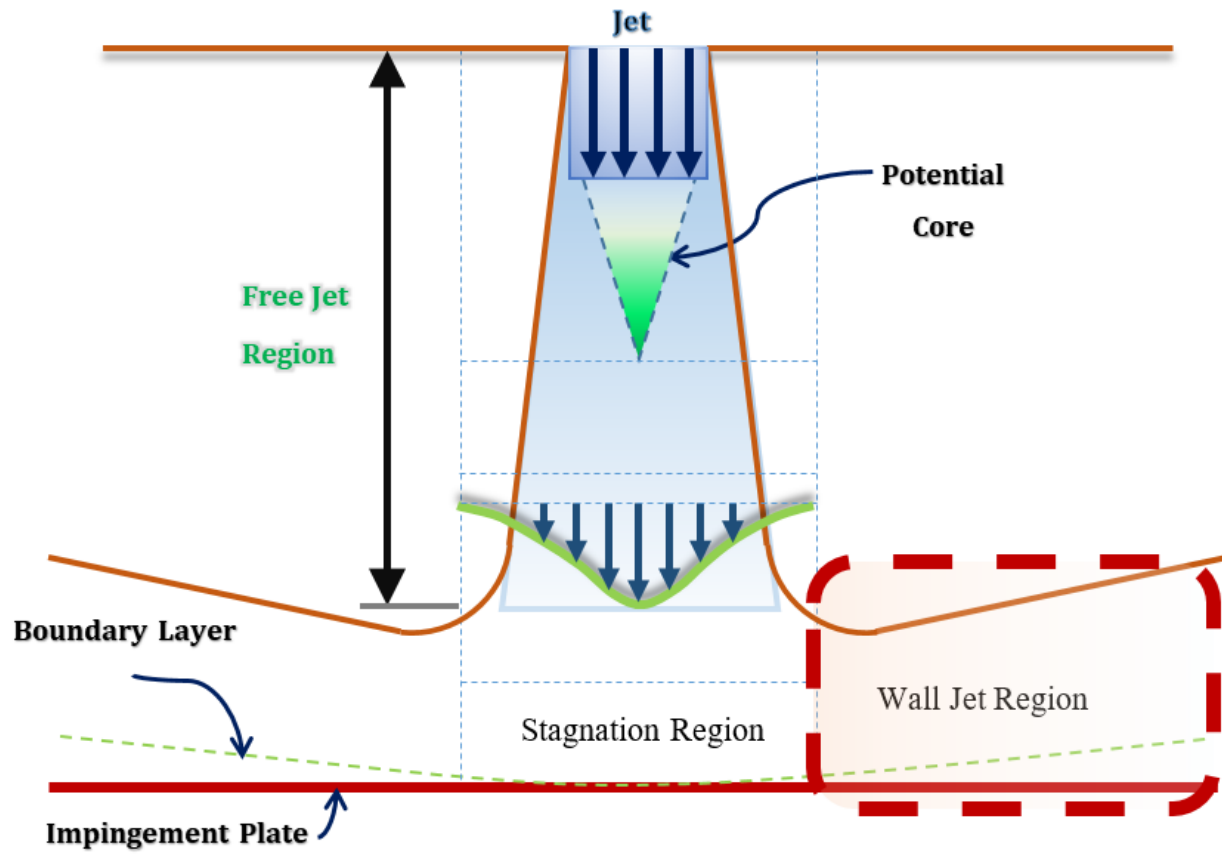


Figure 1: Jet impingement schematic diagram

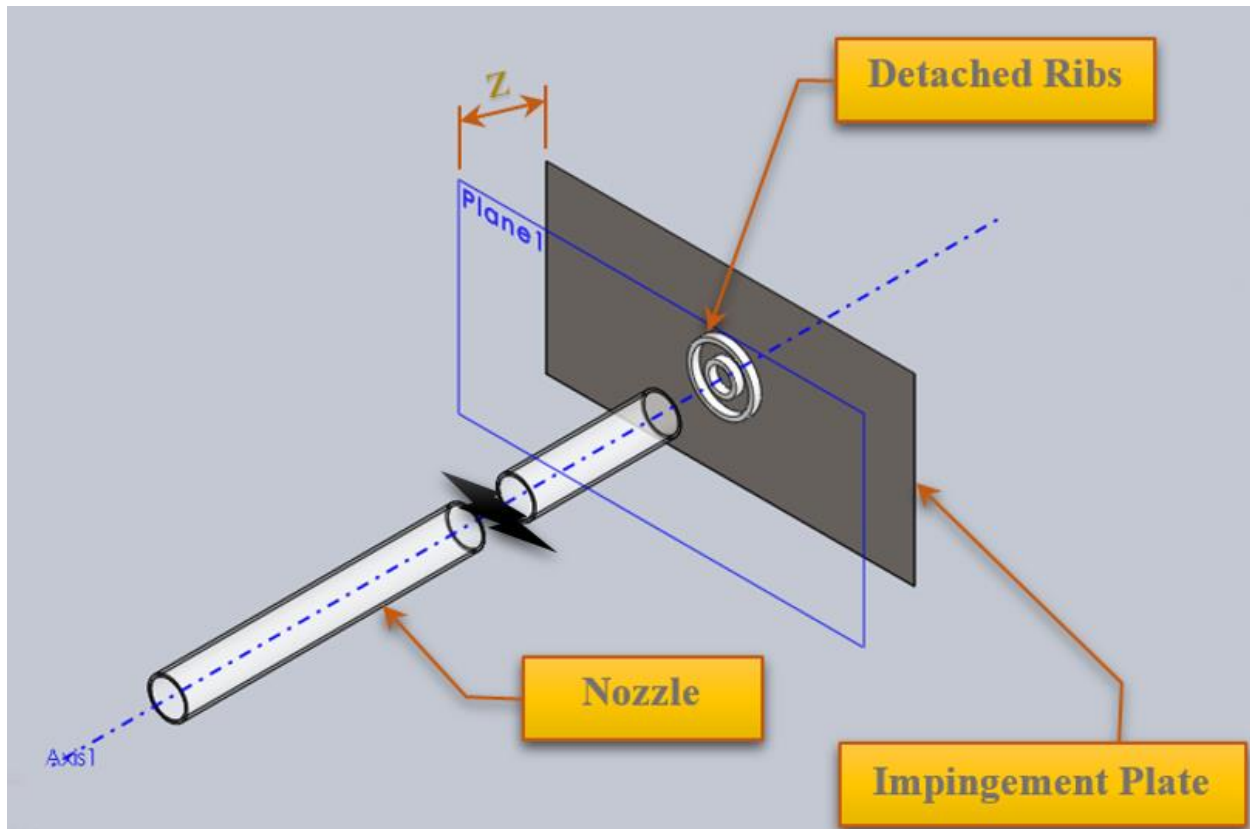


Figure 2: Jet impingement flow case

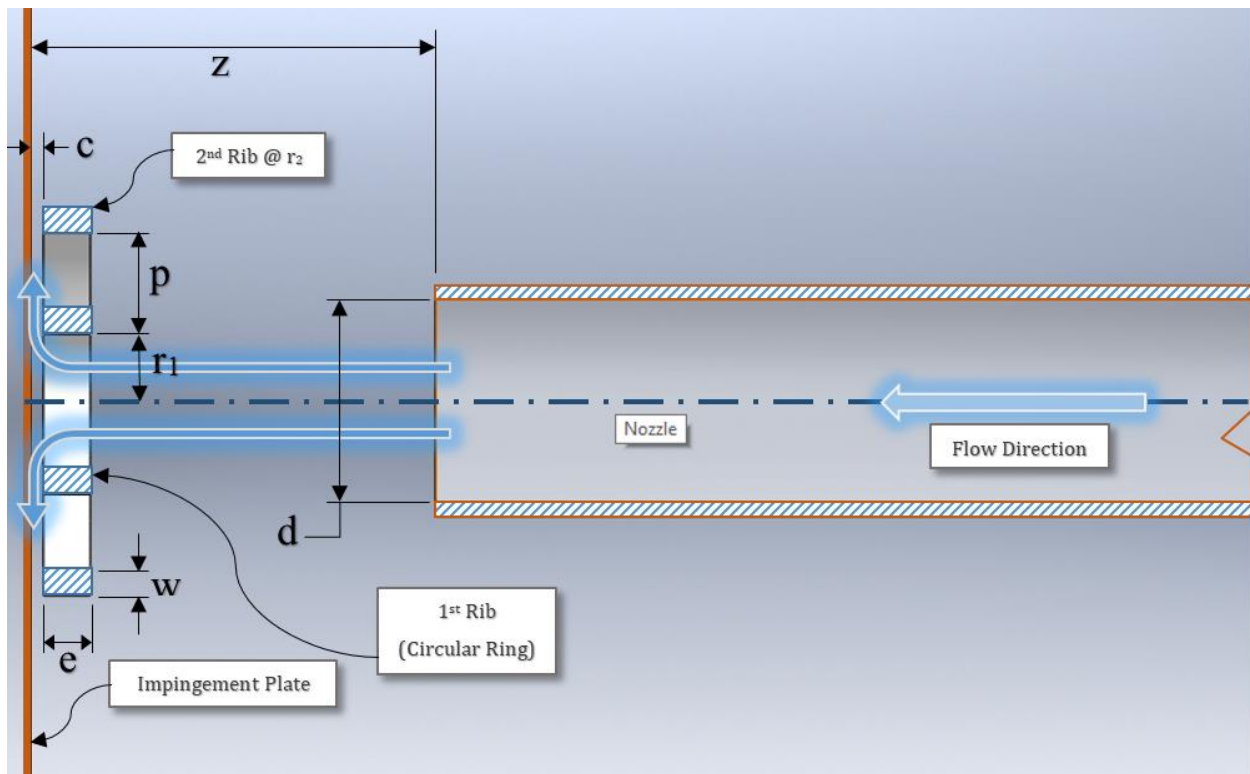


Figure 3: Geometry of axisymmetric detached rib with jet impingement

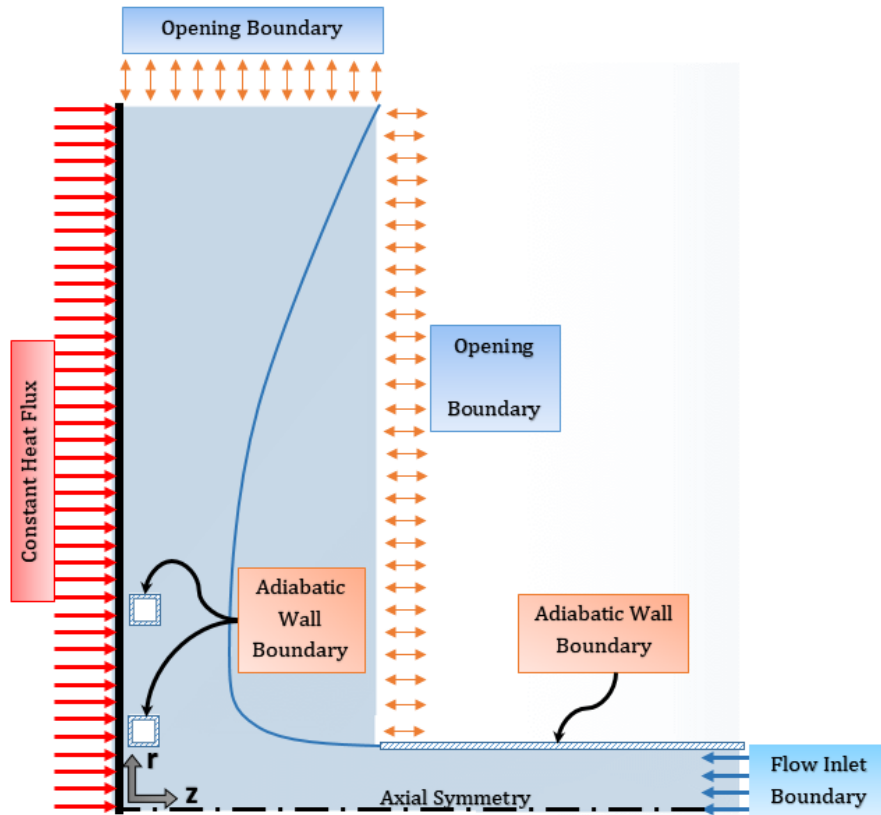


Figure 4: Schematic flow diagram with inlet & boundary conditions

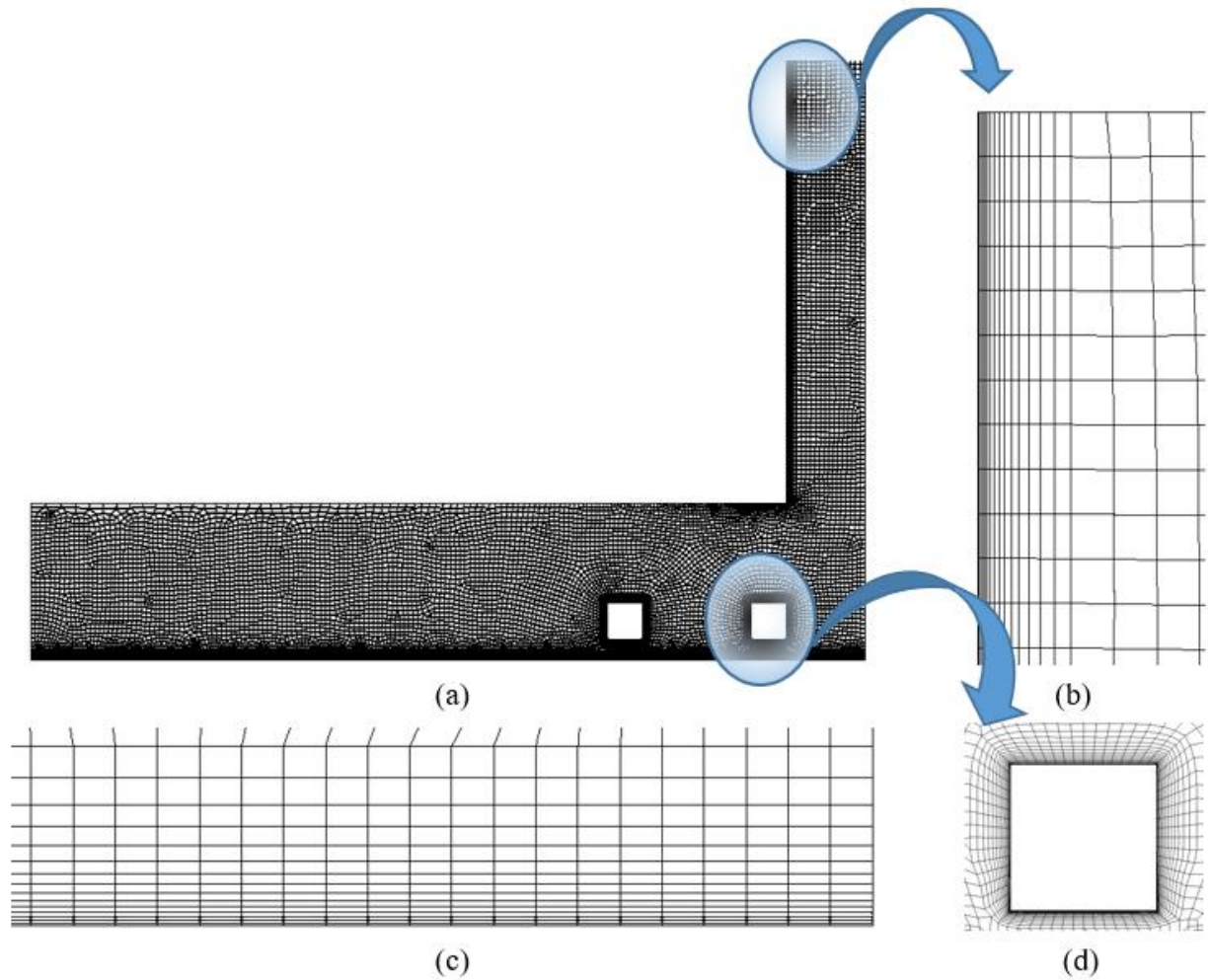


Figure 5: Mesh refinement details (a)- complete axisymmetric geometry mesh (b)- mesh refinement at nozzle wall (c)- mesh refinement near the heated plate (d)- mesh refinement at rib wall

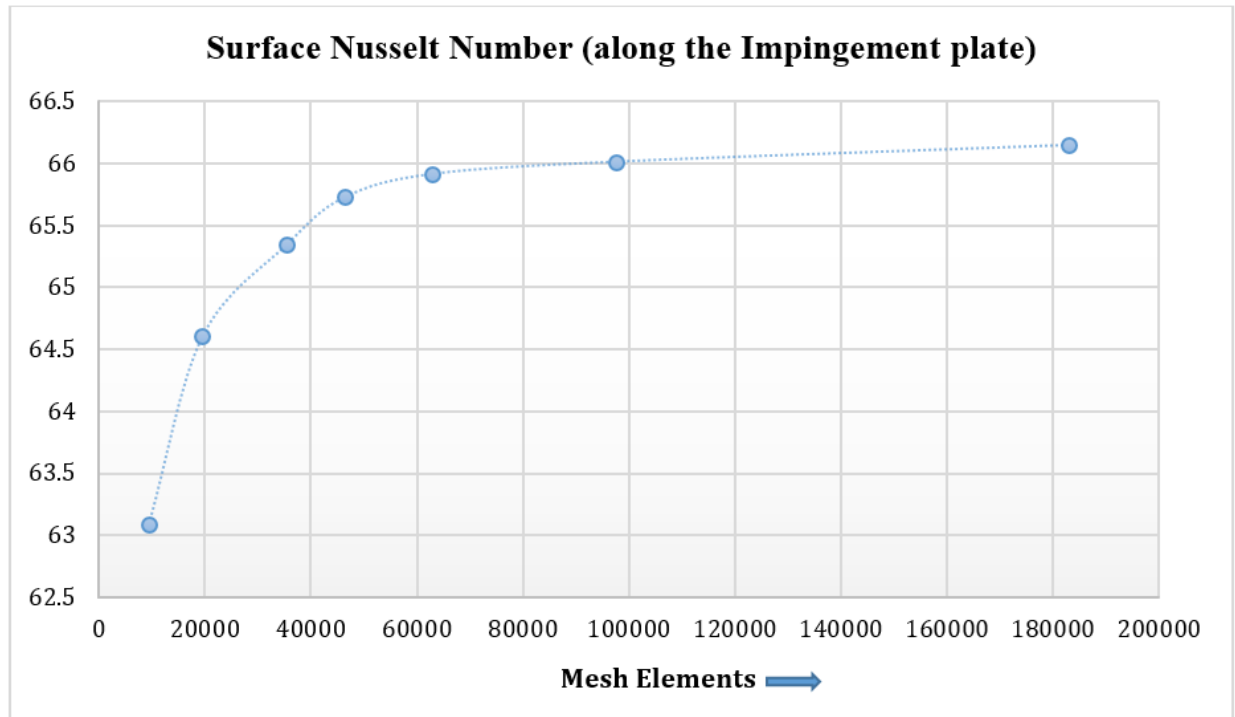


Figure 6: Mesh independence study for surface Nusselt number

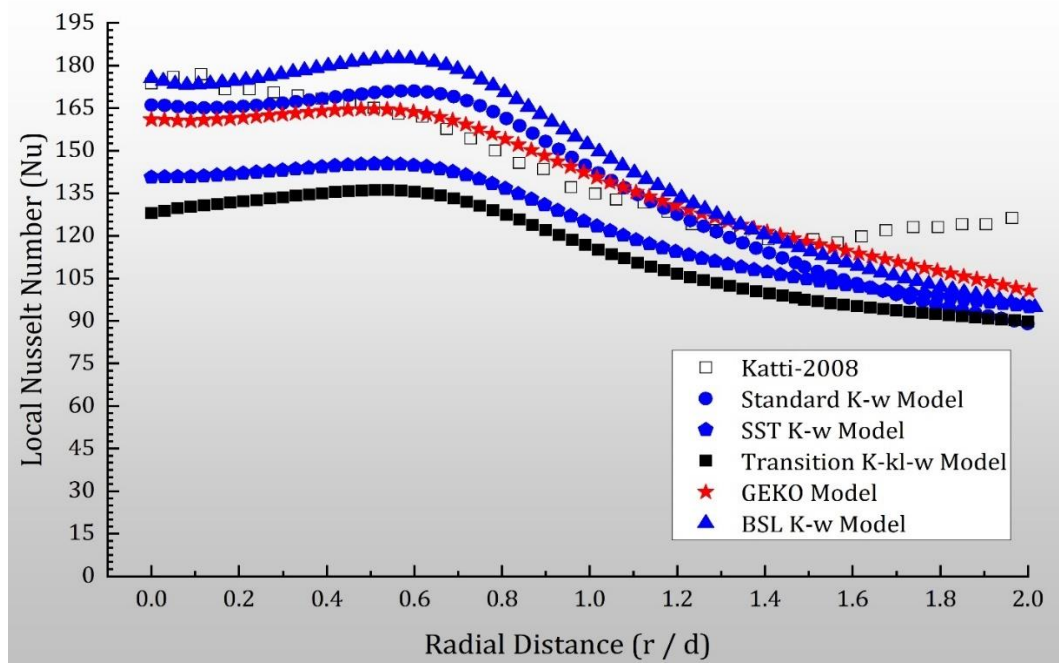


Figure 7: Turbulence model validation for smooth flow impingement (without ribs)

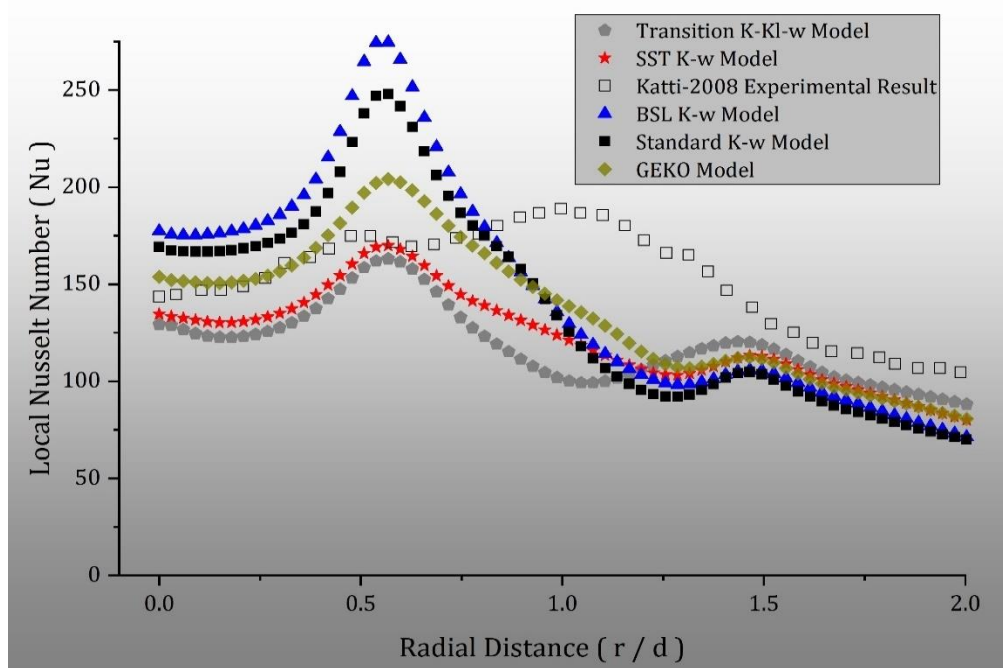


Figure 8: Turbulence model validation for jet impingement case involving rib turbulators

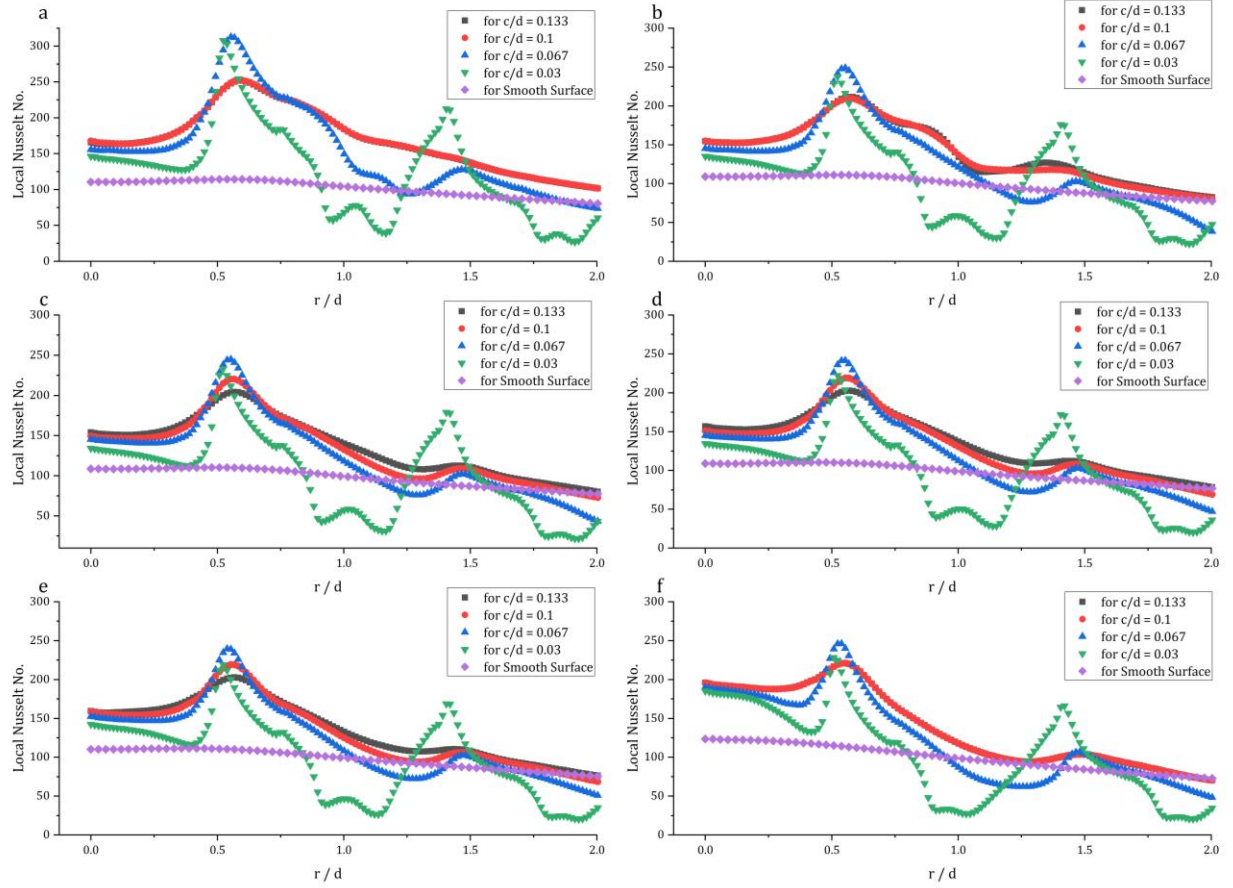


Figure 9: Nusselt number plots for heat transfer study with different clearance ratios at $p/e = 4.0$, $e/d = w/d = 0.23$, $r_1/d = 0.5$ & $Re = 20,000$; a) $z/d = 0.5$; b) $z/d = 1.0$; c) $z/d = 2.0$; d) $z/d = 3.0$; e) $z/d = 4.0$; f) $z/d = 6.0$

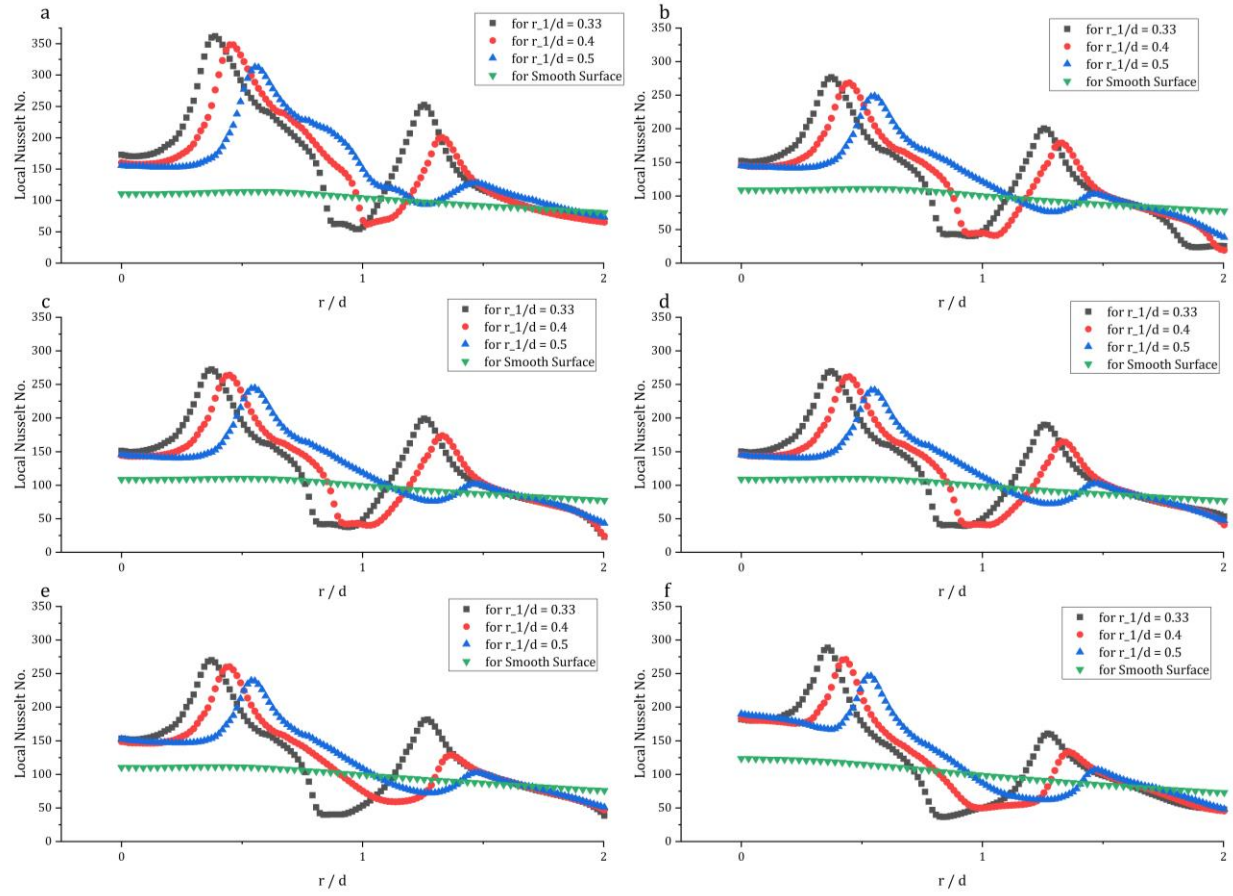


Figure 10: Nusselt number plot for heat transfer study with different radial distances of first rib, $p/e = 4.0$, $e/d = w/d = 0.23$, $c/d = 0.067$ & $Re = 20,000$; a) $z/d = 0.5$; b) $z/d = 1.0$; c) $z/d = 2.0$; d) $z/d = 3.0$; e) $z/d = 4.0$; f) $z/d = 6.0$

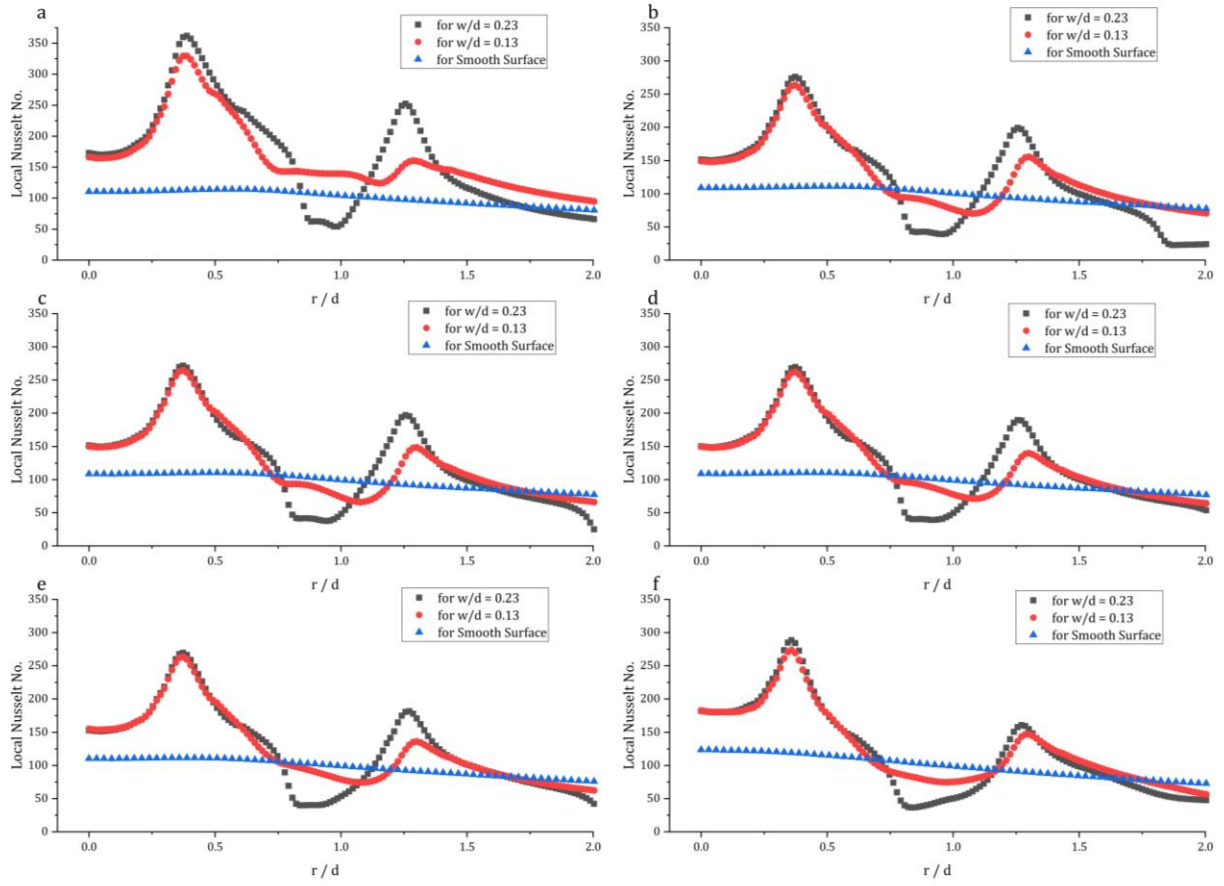


Figure 11: Nusselt number plot for heat transfer study with different rib widths, $r_1/d = 0.33$, $p/e = 4.0$, $e/d = 0.23$, $c/d = 0.067$ & $Re = 20,000$; a) $z/d = 0.5$; b) $z/d = 1.0$; c) $z/d = 2.0$; d) $z/d = 3.0$; e) $z/d = 4.0$; f) $z/d = 6.0$

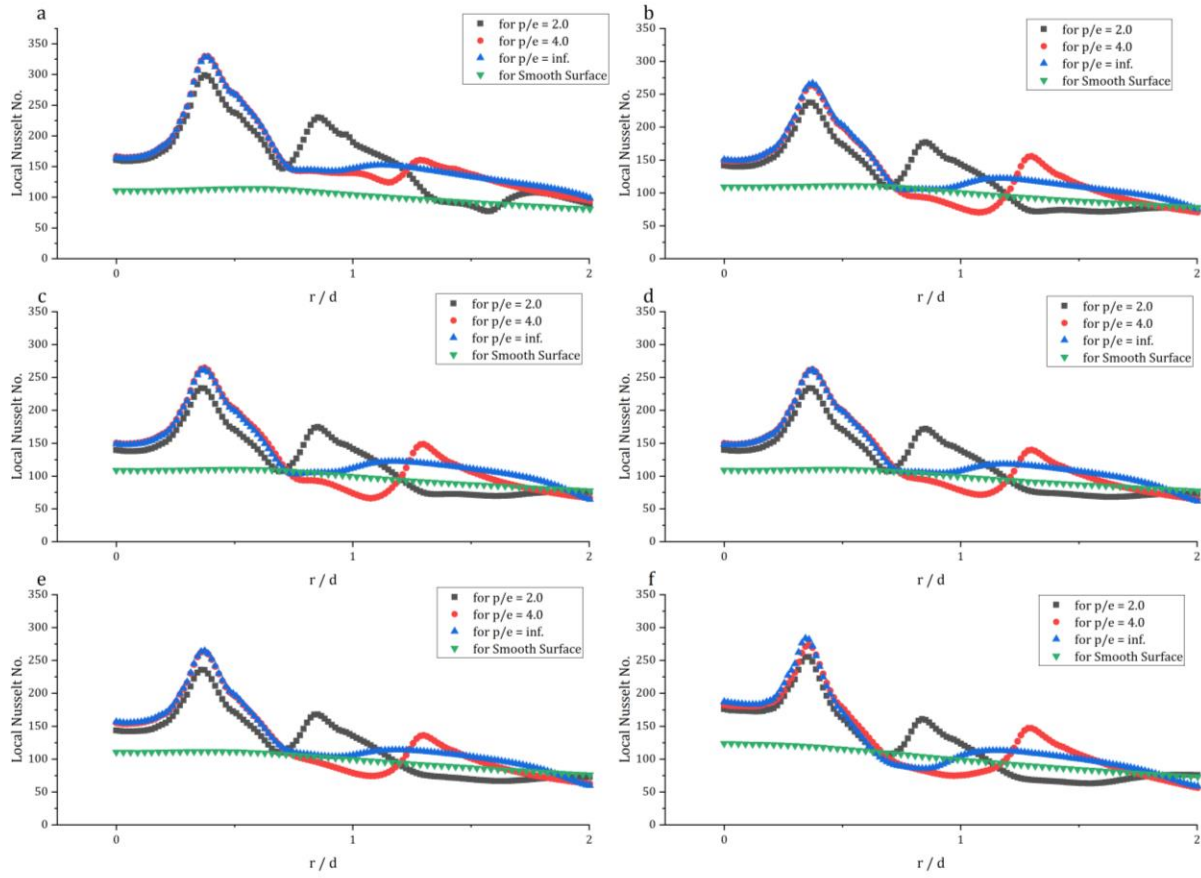


Figure 12: Nusselt number plot for heat transfer study with different pitch distances of ribs, $r_1/d = 0.33$, $e/d = 0.23$, $w/d = 0.23$, $c/d = 0.067$ & $Re = 20,000$; a) $z/d = 0.5$; b) $z/d = 1.0$; c) $z/d = 2.0$; d) $z/d = 3.0$; e) $z/d = 4.0$; f) $z/d = 6.0$

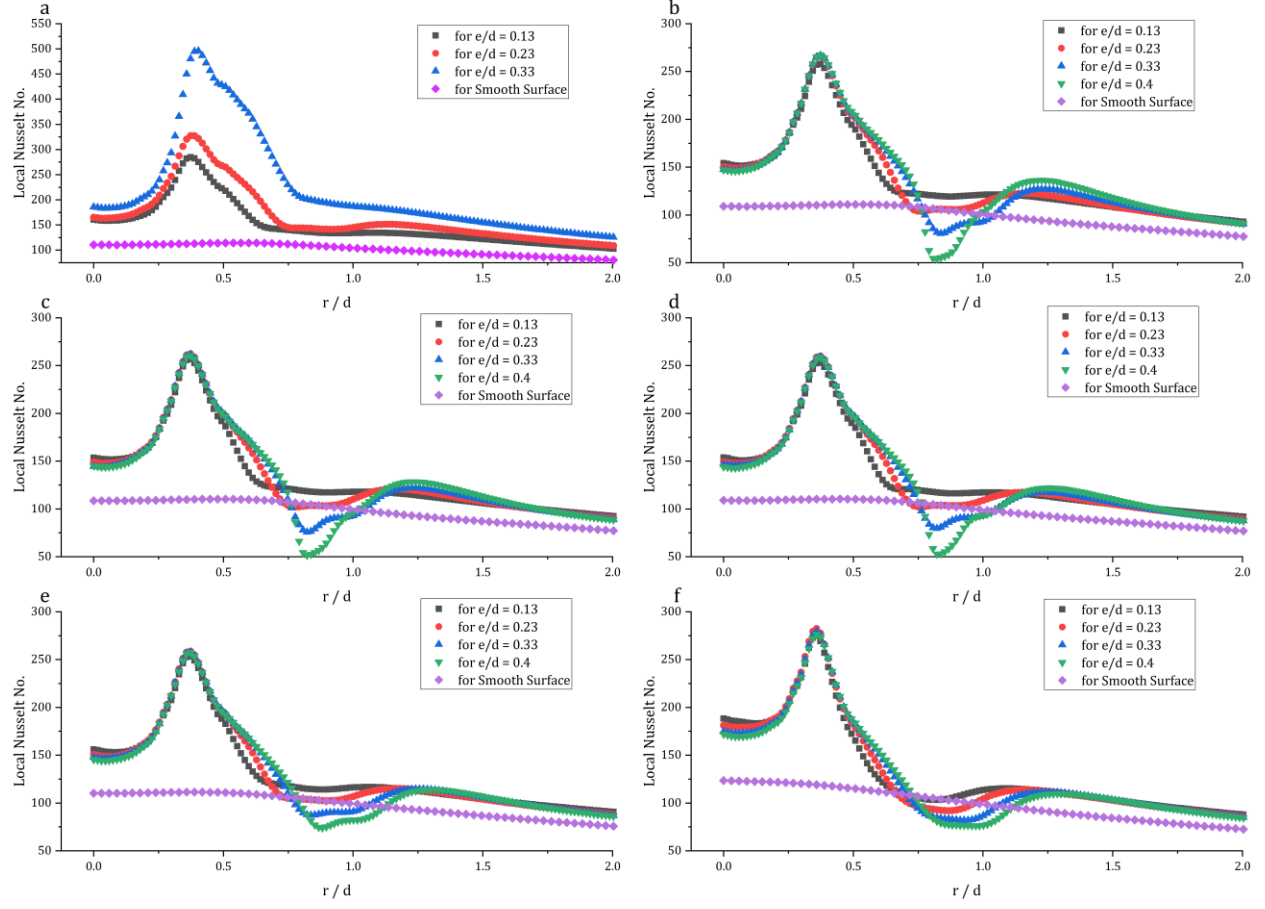


Figure 13: Nusselt number plot for heat transfer study with different rib heights, $r_1/d = 0.33$, $p/e = 4$, $w/d = 0.23$, $c/d = 0.067$ & $Re = 20,000$; a) $z/d = 0.5$; b) $z/d = 1.0$; c) $z/d = 2.0$; d) $z/d = 3.0$; e) $z/d = 4.0$; f) $z/d = 6.0$

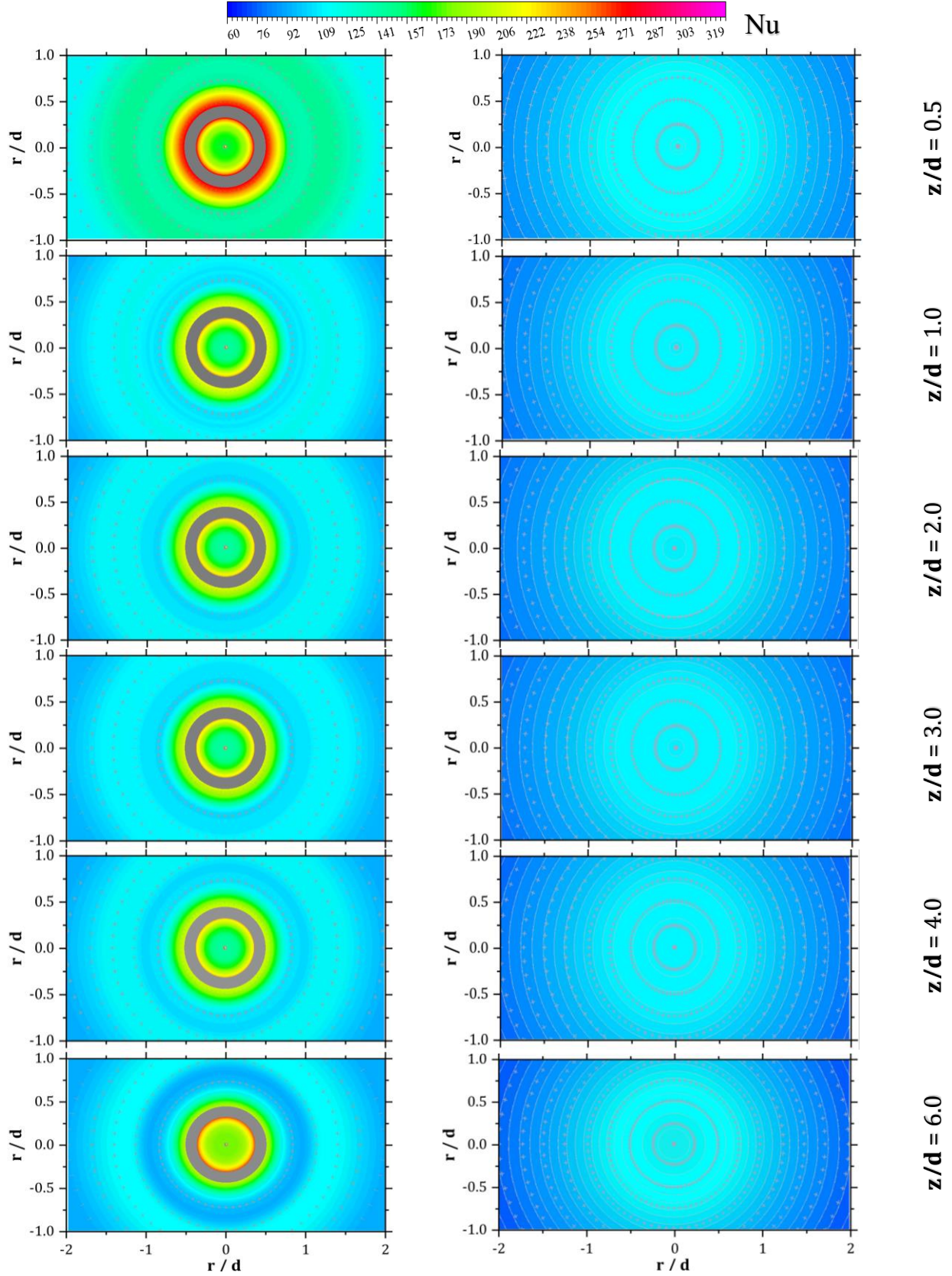


Figure 14: Nusselt number contours over the impingement plate up to the radial location of $r/d = 2.0$; Rib cases on left side and smooth surface without ribs on right side

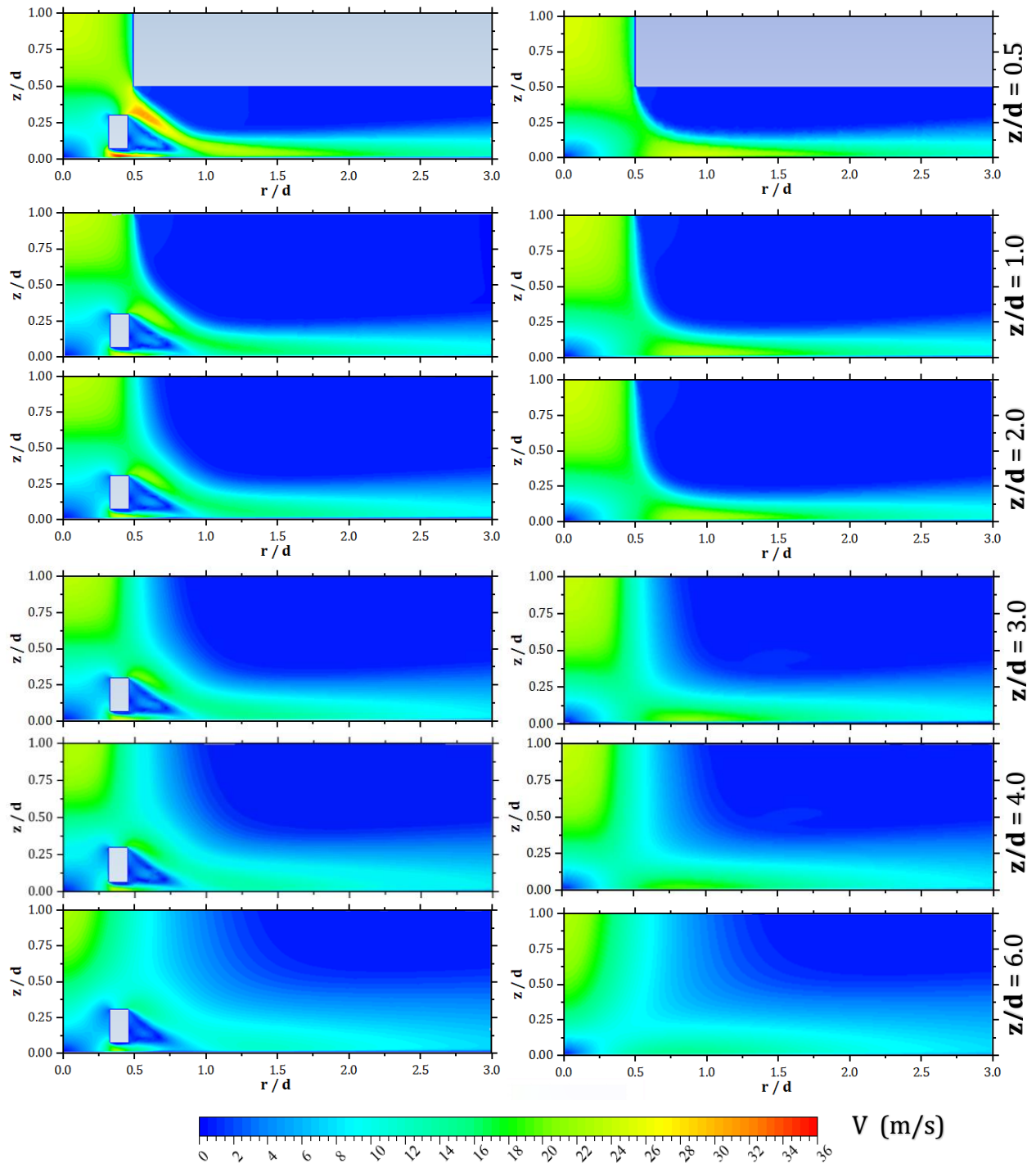


Figure 15: Velocity contours in the axisymmetric region (free & wall jet) up to the radial location of $r/d = 3.0$; Rib cases on left side & respective cases of Smooth Surface without ribs on right side

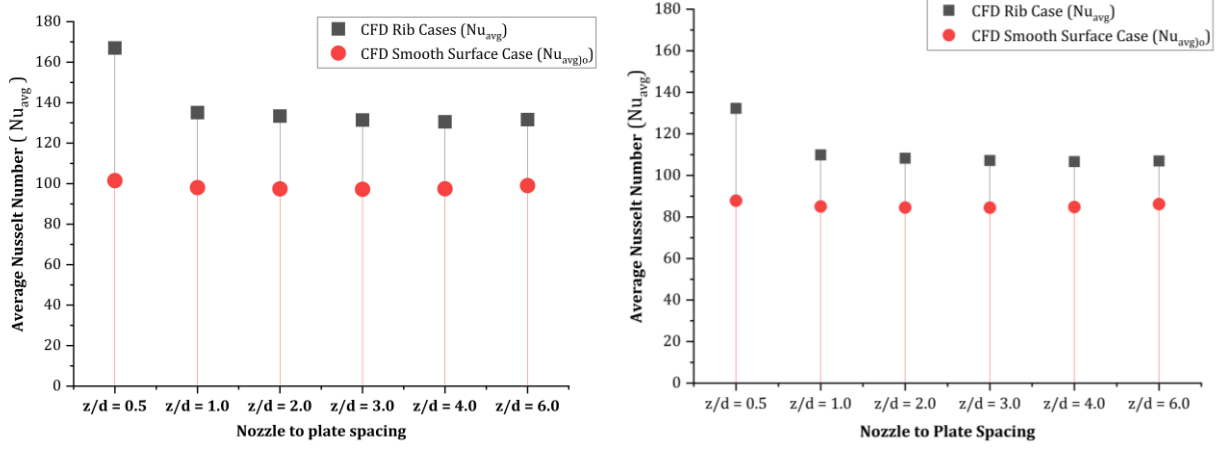


Figure 16: Comparison of average Nusselt number (ribbed vs smooth Surface) up to $r/d = 2.0$ a) $Re = 20,000$; b) $Re = 15,000$

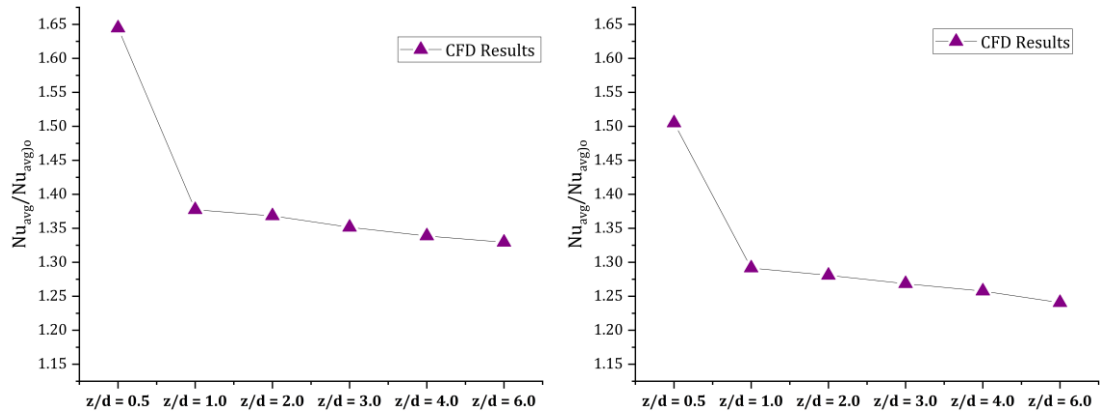


Figure 17: Average gain in heat transfer as a ratio of average Nusselt numbers for ribbed & smooth surfaces up to $r/d = 2.0$ a) $Re = 20,000$; b) $Re = 15,000$

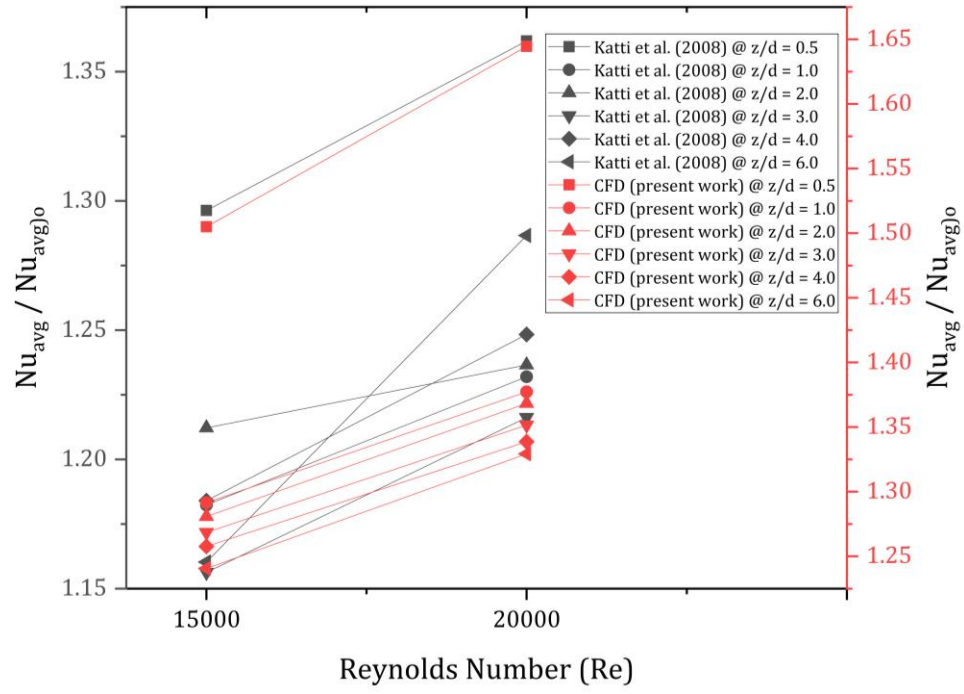


Figure 18: Effect of Reynolds number variance on heat transfer enhancement (left axis: experimental results, right axis: numerical results)



DSpace

<https://dspace.org/>

Heat transfer and fluid flow characteristics investigation using detached ribs in an axisymmetric impinging jet flow

Rasheed, Adnan; Allauddin, Usman; Ali, Hafiz Muhammad; Uzair, Muhammad; Verdin, Patrick G. ...

2022-10-15

Springer

<https://doi.org/10.1007/s10973-022-11640-w>

Rasheed A, Allauddin U, Ali HM, et al., (2022) Heat transfer and fluid flow characteristics investigation using detached ribs in an axisymmetric impinging jet flow. Journal of Thermal Analysis and Calorimetry, Volume 147, Issue 24, December 2022, pp. 14517-14537