A numerical investigation on turbulent convective flow characteristics over periodic grooves of different curvatures

Auronil Mukherjee\textsuperscript{1} and Arnab Chakraborty\textsuperscript{2, *}

\textsuperscript{1}Department of Applied Mechanics, IIT Madras, Chennai-600036, Tamil Nadu, India

\textsuperscript{2}Department of Mechanical Engineering, IIT Madras, Chennai-600036, Tamil Nadu, India

ABSTRACT

Heat transfer augmentation is an essential requirement in all heat transfer devices, such as heat exchangers used in biomedical applications, electronic cooling, solar air heaters, nuclear reactor cores, and gas turbine blade cooling etc. Extended surfaces, protrusions, dimples, internal ribs or, grooves are among the few which are incorporated in the last three decades with the goal of heat transfer augmentation for the better performance of the thermal device. However, considering the usefulness, its relative scarcity in literature, and other advantages of grooved surfaces, we have considered this as a method of heat transfer augmentation here. This makes its necessary to investigate the physics due to turbulence and thermal behavior over periodic grooves (or, ribs) is the prime importance of the present paper. In this work, numerical simulations of turbulent forced convection are carried out by introducing curvatures at sharp corners of a periodic grooved channel at the lower wall. A heat source of constant magnitude is supplied in the bottom wall while the upper wall is insulated. Computations were performed using $k-\varepsilon$ (RNG) model in RANS formulation implemented in finite volume-based solver in the commercial package ANSYS Fluent 19.2\textsuperscript{\textregistered}. The simulations are performed over varying Reynolds numbers (Re) of 6000-36000 where ratio of the height of the channel and breadth of the groove, groove pitch ratio, and depth ratio kept constant as 1, 2, and 0.5 respectively. Assessment of coefficient of heat transfer, frictional losses, and magnitude of heat enhancement are systematically carried out over varying radius of curvatures on grooves. The insertion of curvatures improves the overall heat transfer by a reasonable magnitude of 12\% with an overall 5\% increment in the magnitude of heat transfer enhancement. Further, the magnitude of heat transfer increases with increase in curvature radius.

The present study proposes several optimal parameters for enhancement of heat transfer in a periodic grooved channel that can be used in many practical thermal devices

\textbf{I. NOMENCLATURE}

\begin{align*}
D_h & = \text{Hydraulic diameter of channel}=2H \\
H & = \text{Distance between grooves} \\
f & = \text{Friction Factor} \\
C_f & = \text{Skin Friction Coefficient}
\end{align*}

\textsuperscript{1} Department of Applied Mechanics, IIT Madras, Chennai-600036, Tamil Nadu, India.
\textsuperscript{2, *} Corresponding author, Department of Mechanical Engineering, IIT Madras, Chennai-600036, Tamil Nadu, India.
\( h = \) Coefficient of heat transfer
\( k = \) Thermal conductivity
\( Nu = \) Nusselt Number
\( p = \) Static Pressure
\( Re = \) Reynolds Number
\( T = \) Température
\( V = \) Mean Velocity
\( P = \) Production of TKE
\( \varepsilon = \) Destruction of TKE
\( V_i' = \) Fluctuation Velocity components
\( \mu = \) Dynamic viscosity
\( \mu_{eff} = \) Effective viscosity
\( \omega = \) Turbulent specific dissipation rate
\( \varepsilon = \) Turbulent dissipation rate
\( \rho = \) Density

II. INTRODUCTION

Heat transfer augmentation is an important requirement in almost all heat transfer devices ranging from heat exchanger, biomedical applications, electronic cooling, solar air heater, nuclear reactor cores, and also in gas turbine blade cooling [1]. The performance of major engineering applications relay on improved convective heat transfer in order to reduce heat exchanger size, weight, and cost. There are many techniques to enhance the transfer of thermal energy to improve the performance of the device. Extended surfaces such as ribs, groove, and rib with helical geometry have been used since a long time to influence the fluid motion leading to formation transverse and longitudinal vortices that improve performances of heat exchangers. Channels where ribs are fitted orthogonal to flow direction induce complex flow scenarios to investigate the influence of inserts on heated turbulent convective flows. The phenomenon of heat transfer and associated frictional losses in flow domains with ribs in perpendicular direction, have been the subject of numerous experimental research since long time. These techniques include arrays of different shaped protrusions, swirl and tumbling chambers, pin fins, dimple surfaces, introducing surface roughness, and rib turbulators and many more [1, 2]. All of these techniques eventually produce
stream-wise coherent vortices that cause the increase in advection within the boundary layer adjacent to the solid surface. These stream-wise vortices and secondary flows increase the turbulence level on the boundary layer and cause flow detachment. Further, detached flows past over the next rib and again produce a boundary layer and the process goes on. This is known as reattached boundary layers (hydrodynamic as well as thermal) that cause a better mixing which improves the overall heat transfer coefficient and skin friction coefficient [3]. This concept was first inspired by observing a natural presence of dermal denticles on shark skins and their movement underneath the sea surface [4]. However, the proper heat removal is vital to reduce heat load from the turbine aero foil blades in hot gas side using high speed air circulated for internal cooling purpose as shown in Figure 1 [5]. So, many researchers in the past investigated the impact of these riblets on heat transmission augmentation. Donne and Meyer [6] has reported an extensive measurement of heat transmission and frictional losses for different rib designs. Comparison of heat transmission and pressure losses were done for four varieties of roughened surfaces by them. Investigations were conducted to efficiently select a particular type of rough surface keeping in mind of practical application. Several studies are conducted on thermal transport and flow dynamics with very less geometric parameter modifications to keep the design simple while the heat transfer is optimum.

Fig 1: Schematic design of an internally cooled gas turbine blade [5].

Liou and Hwang [7] used holographic interferometry technique in order to assess heat transfer coefficients locally along with the capturing frictional loss in an asymmetrically channel with square shaped ribs at varying ratios of height to pitch. Lorenz et al. [8] studied patterns of coefficient of heat transmission and frictional losses on the walls within an asymmetric grooved channel with ribs for turbulent flow and heat transfer. It was observed that onset of development of boundary layer due to the thermal effect started growing at stagnation point proximal to edge of a groove. They further compared the coefficient of heat transmission to a smooth channel. Yang and Hwang [9] investigated thermal transport in ducts with the shape of a rectangle where ribs were installed on one side of the wall and looked into the thermal properties of a duct flow with periodic ribs. They found pair of vortices rotating in the anticlockwise direction. Mukherjee and
Saha [10] studied the effect of fillets in contracted and expanded rectangular grooves for a laminar incompressible flow. They demonstrated an improvement in heat transfer using fillets of an arbitrary radius. The investigation, however, was limited to a low Reynolds number flow and effect of the fillet radius on flow physics was not reported. Eiamsard et al. [11] studied effect of different RANS turbulent models and groove width to channel ratio in a rectangular grooved channel and compared the increment in heat transfer with respect to a smooth channel of the equivalent hydraulic diameter. The RANS model computes statistically time averaged flow quantities while modelling all turbulence effects, resulting in fairly accurate predictions and success in a wide range of industrial applications. However, a detailed investigations of effect of curvature of varying radius in one sided expanded grooved channel in a high-speed turbulent convective flow is not present in existing literature. Hence there is a need to investigate the occurring flow dynamics and thermal transport due to the insertion of these curvatures at the sharp corners in a grooved channel. Further it is also necessary to observe the variation in the heat transfer rate and frictional loss with the curvature radius for a given magnitude of Re.

The objective of the present study is many folds: Firstly, this study is the first one which is going to report the effect of curvatures on the rectangular grooves on the effectiveness of the channel's total heat transfer. The associated pressure loss due to the introduction of the curvatures of different radius are also studied along the channel over the range of Reynolds numbers. Further, the heat transfer enhancement factor over a range of Re varying from 6000 to 36000 is also investigated. The turbulent characteristics (TKE, Production of TKE, destruction or, dissipation etc) of the forced convective flow are also evaluated over as these parameters are important to analyse flow mixing and hereby, the heat transfer coefficient and pressure loss via skin friction coefficient. Finally, all the parameters are compared with similar geometry consisting rectangular groove without curvature.

So, the present paper is organized as follows: section II reports the introduction on the topic while section III describes the problem formulation, computational domain, grid convergence and a bit more detail on the RANS modelling approaches. Section IV is about the in-depth explanations of the results and finally the key findings are discussed in conclusion.

III. METHODOLOGY

A. Description of the 2D Computational Domain

The computational domain of the present study consists of a rectangular grooved channel separately consisting both with and without curvature in the horizontal plane as depicted in Fig 2. The red arrow shows the flow direction of the fluid.
Fig 2: Diagram of the computational domain: (a) Without curvature, (b) With Curvature on the grooves.

An illustration of a single groove with the geometric dimensions is shown in Fig 3. The geometric dimensions are kept same as mentioned in Eiamsa-ard and Promvonge [11], with expanded region height, $H=40$ mm. Curvatures of various radius ranging from 3 mm to 9.5 mm, are incorporated in the sharp corners of the initial rectangular geometry.

Fig 3: Geometric details of a single groove: (a) without groove, (b) with curvature at corners.

A constant heat source is supplied to the lower wall while the upper wall is insulated, as stated by Lorenz et al. [6] for the similar reasons. Air is used as the working fluid and it travels unidirectionally along the specified direction. The flow is presumptively independent of time and incompressible. [11]. The $Re$ at the inlet ranges from 6000 to 36000. At the exit, there is no pressure gradient and there are no slip boundary conditions at the wall [11]. Chaube et al. [15] reported that the results obtained using a 2D flow model are in reasonable agreement with a 3D model. This statement is further verified by in the present study for a single test case and we found similar results. Hence, 2D geometry is used in the present work to save more computer memory, processing time, and the computational costs. The boundary conditions considered here are listed as given below in Table 1.

| Location | Type of Boundary | In mathematical form | Magnitude | Unit |
|----------|------------------|---------------------|-----------|------|
| Inlet    | Constant Uniform Velocity | $U = \text{constant}$ | 1.19-7 | m/s  |
### Outlet

| Outlet          | Constant Pressure | \( P = 0 \) | 0 | \( Pa \) |
|-----------------|-------------------|--------------|---|----------|
| Upper Wall      | Insulated         | \( q = 0 \) | 0 | \( W/m^2 \) |
| Lower Wall      | Constant Heat Flux | \( q = \) constant | 2000 | \( W/m^2 \) |

**B. Governing Equations of Fluid Flow**

The dynamics of fluid transport and thermal transmission through the flow domain are controlled by the following 2D steady equations of mass balance, momentum balance, and equation for energy transport:

**Equation of continuity:**

\[
\frac{d}{dx_i}(\rho u_i) = 0
\]  

**Equation of momentum:**

\[
\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \frac{\partial}{\partial x_j} \left( -\rho u_i' u_j' \right)
\]  

**Energy Equation:**

\[
\frac{\partial}{\partial x_i}(\rho u_i T) = \frac{\partial}{\partial x_j} \left( (\Gamma + \Gamma_t) \frac{\partial T}{\partial x_j} \right)
\]  

In expression (3), \( \Gamma \) and \( \Gamma_t \) represents the two types of thermal diffusivity: molecular and turbulent respectively.

**Reynolds-averaged Navier-Stokes (RANS) modelling**

In most industrial scale turbulent flow situations, where a reasonable match between average accuracy and computing cost is utter most important, RANS is the most cost-effective and extensively utilised modelling approach. The dependent variables in RANS are decomposed into a mean and fluctuating component and inserted into the mass, momentum, and energy equations. Further, these quantities are averaged over time to generate equations for mean variables. In order to provide a closure of mean flow equations, the effects of variable quantities on the mean flow must be modelled. RANS is discussed in further depth elsewhere [16-19]. In this work, however, the RNG K-\( \varepsilon \) turbulence model is applied.

The governing equations for the RNG k-\( \varepsilon \) model are:

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b + \rho \varepsilon - Y_M + S_k
\]  

\( \alpha_k \) = 1.0

\( \mu_{eff} \) = 0.25

\( Y_M \) = 0.01

\( S_k \) = 0.2

\( \varepsilon \) = 3.0

\( G_b \) = 0.1

\( G_k \) = 0.1
\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \sum_{i} \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \quad (5)
\]

Mean velocity gradients that cause turbulence by creating kinetic energy is denoted by \( G_k \) in the equations 4&5 while the production of kinetic energy during turbulence because of buoyancy is given by \( G_b \). Variable dilatation in compressible turbulence and its impact on a measure of overall dissipation is represented by \( Y_M \). For \( k \) and \( \varepsilon \) the reverse \( Pr \), are \( \alpha_k \) and \( \alpha_\varepsilon \) respectively. \( S_k \) and \( S_\varepsilon \) represent the individual source terms.

The RANS approach to create a turbulence model requires the Reynolds stress term denoted by \(-\rho u'_i u'_j\) in equation (2) to be modelled. We choose the \( k - \varepsilon \) turbulence framework for closure of the equations. The Reynolds stress term and the mean velocity gradients term are related using the widely used Boussinesq technique. It is given by:

\[
-\rho u'_i u'_j = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (6)
\]

In the above expression (6), \( \mu_t \) denotes the term which computes the turbulent viscosity. It is given by:

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (7)
\]

The expression for the TKE production, \( k \) is written as:

\[
\frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon \quad (8)
\]

The corresponding equation for the turbulence destruction of the TKE, \( \varepsilon \) is shown below:

\[
\frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (9)
\]

In the above expression (9), \( G_k \) denotes the rate of energy production due to turbulence and \( \rho \varepsilon \) denotes the rate of destruction. \( G_k \) is further denoted by:

\[
G_k = -\rho u'_i u'_j \frac{\partial u_j}{\partial x_i} \quad (10)
\]

We define and specify the boundary value of the turbulent quantities proximal to the wall using the enhanced wall treatment method in ANSYS Fluent 19.2\textsuperscript{®}. The empirical constants are taken as \( C_\mu = 0.087, C_{1\varepsilon} = 1.43, C_{2\varepsilon} = 1.9, \sigma_\varepsilon = 1.32 \) and \( \sigma_k = 0.98 \) respectively in the governing transport equation of turbulent flow.

**C. Meshing and grid independence study**
Figure 4 depicts the variation of coefficient of heat transfer at the bottom wall over five different mesh sizes. The laminar sub-layer is resolved rectangular grids using face meshing techniques, with adopting a grid for \( y^+ \approx 2 \) at zone close to wall [11]. This helps to retain the advantages of a structured mesh. To establish the optimal mesh size for numerical simulations, a grid sensitivity analysis is further performed. The number of cells used to attain grid independence is varied between 50,900 and 155,992 rectangular types of meshes and the coefficient of heat transfer is calculated for each case at \( Re \) of 12000. With reference to the finest grid, the error is calculated. The optimum grid is obtained, keeping residual less than 10e-03.

![Graph 1]

**Fig 4:** Mesh independent study at different grid sizes.

![Graph 2]

**Fig 5:** Local Nusselt number (\( Nu \)) distribution over different \( l/H \).

E. CFD Model Validation

The results obtained through numerical simulations were validated for a steady incompressible flow at a \( Re \) of 12000 using experimentally obtained results of Lorenz et al. and numerical outputs of Eiamsa-ard et al. [8, 11] respectively. As seen in Fig. 5, the fluctuation of \( Nu_m \) is examined along the wall of a single groove. The results of the current simulation are seen to reasonably and satisfactorily match those found in previous literature.

F. Numerical Methods

The numerical method based on the principle of finite volume solver was employed to discretize both the steady incompressible Navier–Stokes equations and the RANS turbulence model. QUICK and central differencing schemes were used respectively to discretize convective and diffusion terms. Discrete nonlinear equations were implicitly implemented using the coupling of pressure and velocity terms. Second order upwind approach is employed to model energy while the SIMPLE framework is used to analyse the pressure field. A constant velocity profile has been established at the inlet. The channel wall has an impermeable boundary condition, while the
bottom wall of the computational domain has a supply of fixed heat flux. The magnitude of intensity of turbulence is kept 10% at inlet \[11\]. The present study has six parameters of interests, namely: (1) Nusselt Number, (2) Frictional losses, (3) Skin Friction Coefficient, (4) Turbulence Kinetic Energy Production, (5) Turbulent Dissipation Rate and (6) Thermal Enhancement Factor respectively. We compute the friction factor value \(f\) using the pressure losses \(\Delta p\) across the computational domain having the hydraulic diameter, \(D_h = 2H\). It is given as:

\[
f = \frac{\left(\frac{\Delta p}{L}\right)D_h}{\frac{1}{2} \rho u^2}\tag{6}
\]

The coefficient of skin friction drag denoted by \(C_f\), is defined as:

\[
C_f = \frac{\tau_{wall}}{\frac{1}{2} \rho u^2}\tag{7}
\]

The magnitude of thermal energy transferred is calculated by \(N_u_m\) which is computed by the following expression:

\[
N u(x) = \frac{h(x)D_h}{k}\tag{8}
\]

The mean Nusselt number is calculated by:

\[
N u = \frac{1}{L} \int N u(x)Ax\tag{9}
\]

We calculate the thermal enhancement factor, \(\eta\), using the following expression:

\[
\eta = \left(\frac{N u}{N u_s}\right) / \left(\frac{j}{j_s}\right)^{1/3}\tag{10}
\]

IV. RESULTS AND DISCUSSIONS

A. Heat Transfer Behavior

The variation of mean Nusselt number with the curvature radius ranging from 3 to 9.5 mm is shown in figure 6 over varying Reynolds numbers of 6000-36000. From the figure, it could be observed that, at a particular \(Re\), the corner losses are reducing with the insertion of a curvature on the corner of the grooves which improves the magnitude of heat transmission. This parameter is directly related to the \(Nu\), so \(Nu\) is also increasing in the similar manner. Similar observations are reported for all \(Re\) used in the present study. A bar chart for comparison in increment of \(Nu\) is provided in Figure 7 for with groove and without groove to show the advantage of incorporating the corner curved grooves over the normal ones.
B. Velocity Streamline and Fluid Flow

Figure 8(a-b) illustrates a comparative image of the vorticity contours and velocity streamlines both with and without curvature inside the 7th and 8th groove of the channel modelled using the RANS framework for a $Re$ of 12000. It gives us a visual feature of the fluid flow. A large recirculation region is formed in the grooved region of the channel after flow separation from the main span of the channel. The curvature at the sharp corners results in relocation of the flow structures which increases the size of the eye of the vortices. The curvatures make the cooler fluid move towards the walls of the grooved region. As the radius of curvature increases for a given $Re$, greater number of streamline structures gets relocated on the wall region which further increase the volume of relatively cooler fluid in the near grooved wall regimes. It is evident that these curvatures further enhance the heat transport in lower wall region owing to shifting in streamline patterns. Addition of a curvature at the sharp corners results in increased formation of longitudinal vortices close to the wall proximal to the groove. Longitudinal vortices enhances the heat transmission more effectively than transverse vortices [13]. Previous researchers found a similar observation of high association between the vorticity flow and the Nusselt number at the vortex production zones [11-12]. However, the curvature effects further improve the effective mixing area of the flow which is indicated by the shifting of the high velocity streamlines towards the lower wall region of the groove thus, improves the heat transfer.

Figure 8(c) illustrates the temperature variation inside the channel. It is observed that addition of curvatures increases the mean temperature distribution proximal to the wall region which enhances the heat transfer rate. Figure 7 depicts a histogram of the amount of heat transfer enhancement has been observed due to the insertion of these curvatures on the simple rectangle groove. The highest magnitude of $Nu_m$ is obtained for a curvature radius of 9.5 mm. The comparative analysis of thermal transport, without and with an optimum curvature is obtained using this configuration. The average percentage of heat transfer enhancement due to the introduction of this optimum curvature is found to be around 12%.
Fig 8: The contours and streamlines for without (Left column) and with (Right) curvature at $Re=12000$ for $7^{th}$ and $8^{th}$ groove: (a) Vorticity, (b) Velocity Streamline, (c) Temperature.

Fig 9 (a-b) illustrates the vorticity contours and streamlines with and without curvature for a $Re$ of 24000 for the 7th and 8th groove (indicated in figure 2) of the channel. It is observed that the effective area of the zone of recirculation in the latter case is reasonably smaller compared to the former one where the mean inlet velocity is considerably higher. From the figure 9a, one can observed that the local vorticity distribution inside the groove with curvature is more uniform than that of the without curvature case. Again, from figure 9b, it is visualized that more uniform size vortices and mixing zone is created in the groove with curvature that further strengthen our claim of its capability to enhance heat transfer. Similar kind of association between the vorticity magnitude and $Nu_m$ is observed here. Although the addition of the curvatures enhances the mixing of the fluid particles thereby improving the transmission of thermal energy, the temperature of the cooler fluid in the proximal region of the groove is higher in this case. This is evident from the fact that average rate of heat transmission rises with increment in the $Re$ as observed from Figure 6.
Fig 9: The contours and streamlines for without (Left column) and with (Right) curvature at \( Re=24000 \) for 7th and 8th groove: (a) Vorticity, (b) Velocity Streamline, (c) Temperature.

Further, the amount of the shifting of the streamlines towards the wall region of the groove is considerably lower in this case. From figure 9c, it is observed, that the uniformity in temperature distribution is better in the curvature corrected groove channel than that of the groove without curvature for the similar reasons discussed above.

We visualize the distribution of vorticity, streamlines and temperature in Fig 10 (a-b) with and without curvature for a \( Re \) of 36000 for the 7th and 8th groove (indicated in figure 2) of the channel. Although there is no significant change in the size of the vortices, a gradual mixing is observed in the temperature profiles. As the mean velocity at the inlet increases, relatively cooler fluid descends towards the grooved wall which enhances the net thermal transmission. Song et al. [20] suggested that significant improvement in heat transmission is obtained through structures originating from the secondary flow. The flux of the magnitude of vorticity in the direction of the flow describes the intensity of the secondary flow and influences the associated thermal enhancement along the main flow. The relation of \( Nu_m \) and magnitude of vorticity depicts its significant effects on the degree of intensity of thermal transport achieved by the secondary flow structures.
Variation of Frictional Losses

Fig. 11 illustrates a trend of approximate values of the friction factor with curvature radius of the fillet using the k-ε turbulence model for unit groove width to height ratio. The fluid on entering the flow domain suffers by a finite magnitude of pressure drop caused due to a loss in energy in corresponding flow fields. However, it could be observed that the losses incurred due to friction reduces with increase in $Re$, as the sub layer zone of the viscous medium is suppressed. Also, the magnitude of frictional losses rises moderately with increment in curvature radius. The net pressure drop across the computational domain is found out using numerical computations. Further, the friction factor $f$, is calculated using the expression (6).
Literature studies suggest that the incurred frictional losses in the grooved channel is higher compared to a smooth channel of an equivalent hydraulic diameter. We visualize the increment in the magnitude of friction factor due to the insertion of these curvatures through a comparative histogram for a curvature radius of 9.5 mm, as illustrated in Fig 12. We observe that the frictional losses has increased by a moderate magnitude of 5%.

D. Analysis Turbulence Kinetic Energy Production and Destruction

The fluid particles inside the channel have a finite magnitude of kinetic energy associated with them. The velocity of a fluid particle consists of the mean and fluctuating quantity respectively. The kinetic energy associated with these fluctuations is termed as the turbulence kinetic energy (TKE). A simple time averaging as shown in expression (11) is performed in terms of a distribution of the flow rate and the magnitude of the mean quantity.

\[(V')^2 = \frac{1}{T} \int_0^T (V(t) - \bar{V})^2 dt\]  

(11)

The mean component of the velocity is represented by overbar quantity. The fluctuating quantities in the above expression (11) are subjected to change in space. In the present study, the flow is incompressible, therefore the governing equation of the flux conservation due to the associated turbulence kinetic energy is given by:

\[\frac{dk}{dt} = -\nabla \cdot T + (P - \varepsilon)\]  

(12)

where, \(k\) is TKE, \(T\) is temperature, \(P\) is production of TKE, and \(\varepsilon\) is the destruction of the energy. We compute the turbulent kinetic energy's material derivative using equation (12) which models the the generation of kinetic energy by an external force and its destruction originating from viscous effects. In the expanded expression of the material derivative, we have the generation
of TKE, diffusion, and destruction terms due to the effects originating from the viscosity of the fluid. The destruction terms hold responsible for generation of eddies with the fluid motion which shears off from the main flow and gets dissipated.

![Image](image1.png)

![Image](image2.png)

![Image](image3.png)

**Fig 13:** The contours of TKE for without (Left column) and with (Right) curvature for 7th and 8th groove for: (a) $Re=12000$, (b) $Re=24000$, and (b) $Re=36000$.

Fig 13(a-c) depicts the turbulence kinetic energy contour plots with and without curvature for three different $Re$ respectively. The maximum magnitude of TKE is observed proximal to the top region of the ribs while its magnitude is minimum at the bottom regions of the grooves for both the configurations. The maximum intensity zone is observed to occur proximal to the main flow inside the channel. The net effective area of the peak turbulent intensity is observed to be higher in the curvature inserted configuration compared to the former one. This indicated a strong magnitude of energy transport of the associated fluid particles which further explains the influence of turbulent intensity on the achieved thermal enhancement. The magnitude of the TKE increases with increase in the mean inlet velocity. This consequently increases the kinetic energy associated with the fluid particles in motion thereby improving the thermal transmission.
The variation of the turbulence energy production and dissipation is illustrated in Fig 14(a). The energy associated to turbulence in the grooved channel is produced by the acting shear force and mixing. From Fig 14(a), the phenomenon of energy cascading seems prominent as the TKE is transmitted from larger eddies to the smaller eddies which is dissipated by the existing viscous forces at the Kolmogorov length scale. The magnitude of energy production and destruction is relatively higher with a curvature configuration in the present study. This higher magnitude of dissipation further plays an influential role in mixing layer and in the turbulence energy budget by acting as a sink term which dissipates energy in the form of heat due to viscous effects. This further explains the higher magnitude of heat transfer for a channel with curvature.

Fig 14(b) depicts a comparative plot of the variation of turbulence kinetic energy (TKE) with $Re$ for both the configuration. The TKE associated with a curvature configuration is observed to be higher compared to a without curvature arrangement due to the aforementioned reasons.

**E. Investigation of Skin Friction Coefficient**

The fluid in motion inside the grooved channel, suffers by frictional losses. This generates a tangential force exerted by the fluid which results in increase in wall shear stress and drag force. Its magnitude is a function of the fluid velocity and the wall skin friction coefficient. The skin friction coefficient, $C_f$ is defined as stated in expression (7). Fig 15(a) illustrates a variation of the area averaged skin friction coefficient on top and lower wall of associated flow domain with $Re$, with and without curvature respectively. The magnitude of $C_f$, is reasonably higher in the upper wall of the channel compared with the grooved lower wall for both the configurations. This is due to the higher magnitude of the associated tangential stress on upper wall of grooved channel compared to lower boundary.

---

**Fig 14:** (a) Trend of kinetic energy production and dissipation with $Re$, (b) Variation of TKE with $Re$ for different configurations (with and without curvature on the corners of the groove).

---

**Fig 15:** (a) Variation of skin friction coefficient with $Re$ for different configurations (with and without curvature on the corners of the groove).
It is also observed that the magnitude of the skin friction drag has increased due to the insertion of these curvatures as evident from Fig. 15. Addition of these curvatures has increased the net shear stress acting on the walls of the grooves for a given \( Re \). The magnitude of \( C_f \), further decrease with increment in \( Re \) due to increase in the mean inlet velocity of the flow.

Fig 15(b) illustrates the trend in variation of the regional value of \( C_f \) on the walls of the 8th groove for both with and without curvature configuration for a \( Re \) of 12000. The trend in the variation of the magnitude of the local \( C_f \), is fairly similar for both the configuration across the entire length scale. The magnitude of \( C_f \), is higher for a channel with curvature due to the greater magnitude of incurred shear stresses along the walls of the rib or, the groove. The higher magnitude of the \( C_f \), is specifically observed in the areas of higher turbulent intensity as evident from the TKE contour plots in Fig 13. A contrasting trend in the variation of local \( C_f \), is observed in the trailing region of the groove (1.5 to 2) as seen in Fig 15(b). The magnitude is higher in this region for a without curvature configuration. This might be due to the presence of sharp corners which causes the more pressure loss (more shear stress on the fluid layers adjacent to the sharp corners) in the fluid whereas, the curvature nature of the geometry reduces the resulting shear stress acting on the grooved wall region which reduces the local skin friction drag.

F. Evaluation of thermal enhancement factor

The addition of the curvatures at the sharp corners enhances the heat transfer with a rise in the frictional losses. Both these parameters are counteracting and there has to be a reasonable tradeoff between the thermal transport enhancement achieved and the incurred losses due to friction. We investigate the performance to compute the effectiveness of the proposed configuration. Figure 16 presents a comparative analysis of the trend in magnitude of heat transmission improvement with \( Re \) for both the configurations without and with a curvature. The thermal enhancement factor is computed for a radius of curvature of 9.5 mm which gives the maximum magnitude of thermal
transmission. The thermal augmentation is calculated with respect to a smooth channel in agreement with reported literatures of Eiamsard et al. [11]. Here, $Nu_s$ and $f_s$ is the corresponding Nusselt number and frictional losses for a channel without grooves. The thermal augmentation factor is observed to get improved for a grooved channel with curvature and it further reduces with increment in $Re$. The average percentage of heat transfer enhancement due to the introduction these curvatures is found to be around 10%.

![Graph of heat transmission improvement vs Re for both configurations.](image)

**Fig 16: Comparison of trend of heat transmission improvement with $Re$ for both configurations.**

**CONCLUSIONS**

Turbulent convective thermal characteristics and flow dynamics behaviours across periodic grooves channel are numerically studied in this work. The primary goal of this research is to investigate the role of curvatures of different radius in heat transfer enhancement and the corresponding frictional loss, skin friction drags and the effect of turbulent kinetic energy production and dissipation in a reasonably high-speed flow regime. The numerical solver is validated by comparing it to prior results reported in literature obtained experimentally, conducted under similar set of conditions. We observe that the introduction of curvature enhances the thermal transmission by a reasonable magnitude of 12\% with minimal rise in friction factor by a margin of 5\%. Further, the magnitude of heat transfer enhancement shows an increasing trend with increment in the curvature radius. Overall, the average percentage of thermal enhancement due to the introduction these curvatures is found to be around 10\%.

**ACKNOWLEDGEMENTS**

The computation resources of IIT Madras are sincerely acknowledged.
DECLARATION OF COMPETING INTEREST

The authors affirm that they are free of any known financial conflicts of interest or close personal ties that might have looked to have affected the research described in the current study.

REFERENCES

[1] Ligrani, Phil M., Mauro M. Oliveira, and Tim Blaskovich. "Comparison of heat transfer augmentation techniques." AIAA journal 41, no. 3 (2003): 337-362.
[2] Liu, Yao-Hsien, Yuan-Hsiang Lo, Xin-Xian Li, and Michael Huh. "Heat transfer and friction in a square channel with ribs and grooves." Journal of Thermophysics and Heat Transfer 30, no. 1 (2016): 144-151.
[3] Duan, Lian, and Meelan Choudhari. "Effects of riblets on skin friction and heat transfer in high-speed turbulent boundary layers." In 50th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, p. 1108. 2012.
[4] Bixler, Gregory D., and Bharat Bhushan. "Fluid drag reduction with shark-skin riblet inspired microstructured surfaces." Advanced Functional Materials 23, no. 36 (2013): 4507-4528.
[5] Han, J. C., Park, J. S., and Lie, C. K., “Heat Transfer and Pressure Drop in Blade Cooling Channels with Turbulence Promoters,” NASA CR-3837, 1984.
[6] Dalle Donne, M., & Meyer, L. (1977). Turbulent convective heat transfer from rough surfaces with two-dimensional rectangular ribs. International Journal of Heat and Mass transfer, 20(6), 583-620.
[7] Liou, T. M., & Hwang, J. J. (1992). Developing heat transfer and friction in a ribbed rectangular duct with flow separation at inlet.
[8] Lorenz, S., Mukomilow, D., & Leiner, W. (1995). Distribution of the heat transfer coefficient in a channel with periodic transverse grooves. Experimental Thermal and Fluid Science, 11(3), 234-242.
[9] Yang, Y. T., & Hwang, C. W. (2004). Numerical calculations of heat transfer and friction characteristics in rectangular ducts with slit and solid ribs mounted on one wall. Numerical Heat Transfer, Part A: Applications, 45(4), 363-375.
[10] Mukherjee, A., & Saha, S. (2021). Effect of fillet on fluid flow and heat transfer characteristics in a grooved channel: A Numerical study. In Proceedings of the 26th National and 4th International ISHMT-ASTFE Heat and Mass Transfer Conference December 17-20, 2021, IIT Madras, Chennai-600036, Tamil Nadu, India. Begel House Inc
[11] Eiamsa-ard, S., & Promvonge, P. (2008). Numerical study on heat transfer of turbulent channel flow over periodic grooves. International Communications in Heat and Mass Transfer, 35(7), 844-852.
[12] Bernard, P. S. (1986). Limitations of the near-wall k-epsilon turbulence model. AIAA journal, 24(4), 619-622.
[13] Kim, W. S., He, S., & Jackson, J. D. (2008). Assessment by comparison with DNS data of turbulence models used in simulations of mixed convection. International journal of heat and mass transfer, 51(5-6), 1293-1312.

[14] Sagaut, P., & Germano, M. (2005). On the filtering paradigm for LES of flows with discontinuities. Journal of Turbulence, (6), N23.

[15] Chaube, A., Sahoo, P. K., & Solanki, S. C. (2006). Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater. Renewable Energy, 31(3), 317-331.

[16] Kumari, N., & Chakraborty, A. (2022, February). The hydrodynamic interaction of turbulent flow with tandem hydrofoils in presence of a free surface. In OCEANS 2022-Chennai (pp. 1-6). IEEE.

[17] Kumari, N., & Chakraborty, A. (2021, December). A Numerical Study of Flow Around Different Hydrofoil Systems In Presence of the Free Surface. In Gas Turbine India Conference (Vol. 85536, p. V001T09A002). American Society of Mechanical Engineers.

[18] Chakraborty, A., & Warrior, H. V. (2020). Study of turbulent flow past a square cylinder using partially-averaged Navier–Stokes method in OpenFOAM. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 234(14), 2821-2832.

[19] Kumari N, Chakraborty A, Jangam S. The Hydrodynamic Analysis of Multiple Foils Translating in Tandem in Presence of a Free Surface. Proceedings of the Institution of Mechanical Engineers, Part M: Journal of Engineering for the Maritime Environment. June 2022.

[20] Song, K. W., & Wang, L. B. (2008). Relationship between heat transfer intensity and absolute vorticity flux intensity in flat tube bank fin channels with vortex generators. Progress in Computational Fluid Dynamics, an International Journal, 8(7-8), 496-502.