A Study of Various Numerical Turbulence Modeling Methods in Boundary Layer Excitation of a Square Ribbed Channel

Hojjat Saberinejad, Adel Hashiehbaf, Ehsan Afrasiabian

Abstract—Among the various cooling processes in industrial applications such as electronic devices, heat exchangers, gas turbines, etc. Gas turbine blades cooling is the most challenging one. One of the most common practices is using ribbed wall because of the boundary layer excitation and therefore making the ultimate cooling. Vortex formation between rib and channel wall will result in a complicated behavior of flow regime. At the other hand, selecting the most efficient method for capturing the best results comparing to experimental works would be a fascinating issue. In this paper 4 common methods in turbulence modeling: standard k-ε, rationalized k-ε with enhanced wall boundary layer treatment, k-w and RSM (Reynolds stress model) are employed to a square ribbed channel to investigate the separation and thermal behavior of the flow in the channel. Finally, all results from different methods which are used in this paper will be compared with experimental data available in literature to ensure the numerical method accuracy.

Keywords—boundary layer, turbulence, numerical method, rib cooling

I. INTRODUCTION

VARIOUS cooling methods in gas turbine industries are widely used and related researches are in demand. Different ribbed internal cooling ducts with difference in shape and configuration are studied in recently researches. Whereas it has a turbulent nature with some complicated phenomena such as separation and secondary flows, it is very important to predict behavior of the flows considering the cooling capability.

It seems that Computational Fluid Dynamics (CFD) is the best way to simulate and predict such flows. Considering that high computational expense for simulating such flows is a major obstacle, it would be advantageous to reduce costs. It is possible with engaging economic methods with low computational costs. It is useful to show which method is powerful with low expense such as $k-\varepsilon$, $k-\varepsilon$–$\omega$ and LES model in comparison with expensive methods such as Large Eddy Simulation (LES) or Direct Numerical Simulation (DNS).

Simulation (LES) or Direct Numerical Simulation (DNS). Regarding to this fact that LES and DNS consider more details of the flow, but they are cumbersome and need powerful facilities and are applicable only for low Reynolds flows. These issues have limited use of these turbulence methods in the industries and have reduced it to the confined research area. For the purpose of applying turbulence methods in cooling, it is important consider which one is more capable to predict thermal effects.

Some heat transfer experiments have been done in both developing flow by Han and Park [1], Liou and Hwang [2], Wagner et al. [3], Wang et al. [4], Chang and Morris [6], fully developed flow by Han [7], Liou and Hwang [8], Baughn and Yan [9], Fann et al. [10], Park et al. [11], Mochizuki et al. [12], Eckard et al. [13], Chen et al. [14], Islam et al. [15], Gao et al. [16]. Also some experimental efforts have been done with Laser Doppler Velocimeter (LDV) such as Rau et al. [16], Sato et al. [17], Liou et al. [18], and Graham et al. [19] and hot-wire anemometer measurements by Hirota et al. [20].

Studies using the economic turbulence models have been presented such as $k-\varepsilon$ model by Durst et al. [21], Liou et al. [22], Acharya et al. [23], Prakash and Zerkle [24], Zhao and Tao [25], Ooi et al. [26], $v^2-f$ model by Ooi et al. [26] and LES model (Experimental validation of large eddy simulations of flow and heat transfer in a stationary ribbed duct) Sewall et al. [27], Saha and Acharya [28], Tafti [29], Abdel-Wahab and Tafti [30]. Other studies to comparing various methods by Bonhoff et al. [31], Lin et al. [32], Jiang et al. [33], Al-Qahtani et al. [34-35], Murata and Mochizuki [36].

In our work both developing and fully developed flows have been consdered in a stationary squared duct with two smooth walls and two ribbed walls with an inline configuration. Rib height to hydraulic diameter $(e/D_h)=0.1$ and the rib pitch to height are $(p/e)=10$ and Reynolds on bulk velocity and hydraulic diameter is 20000. There are 9 squared ribs, Walls temperature has been set to $1200^0k$ and entrance flow is uniform.

Results show that some methods are more suitable and more efficient in thermal modeling in comparison with other ones. In this Paper results are validated with experiments at Re=10000 and 20000 captured with LDV and LES modeling in a stationary ribbed duct. It shows good analogy and leads us to choose an optimum method for internal cooling phenomena with less costs and applicable results. Results show that $k-\varepsilon$ realizable with enhanced wall treatment (two layer approach)
and fine meshes near the walls have a great capability and acceptable tolerances in according the low computational expenses and could be engaged for higher Reynolds number flows which LES or DNS are not practical.

II. THEORETICAL METHOD

The turbulent flow and heat transfer simulation of the three dimensional channel with square ribs were presented by the steady and unsteady-state Navier-stokes and energy equations. Among the four methods for capturing the flow and temperature field inside the channel, the $k-\varepsilon$ methods and $k-\omega$ methods were solved in steady mode and Reynolds stress model (RSM) in unsteady mode. The average Reynolds governing equation that describe the turbulent flow and energy were defined as follow

$$\rho \left( \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \frac{\partial u_i}{\partial x_j} \right] + \frac{\partial}{\partial x_j} \left[ \sigma_k \frac{\partial u_i}{\partial x_j} \right] + \frac{\partial}{\partial x_j} \left[ \sigma_t \frac{\partial T}{\partial x_j} \right] + S_i$$

(1)

In which the $C_1, C_2, C_3$ are experimental constants and $\sigma_k, \sigma_t$ and $\sigma_\varepsilon$ are also turbulent prandtl numbers for $k$ and $\varepsilon$, respectively. The terms $C_1(1+\frac{k}{\varepsilon}) \sigma_k, C_2(1+\frac{\varepsilon}{k}) \sigma_t, C_3(1+\frac{\varepsilon}{k}) \sigma_\varepsilon$ are representatives of shear generation and viscose dissipation processes, respectively. The term $C_1(1+\frac{\varepsilon}{k}) \frac{\varepsilon}{k}$ is also representative of buoyancy effects. The generation of turbulent kinetic energy due to mean velocity gradients, ($G_k$) is defined as follow:

$$G_k = -\rho \frac{\partial u_i}{\partial x_j} \frac{\partial u_i}{\partial x_j} \approx \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j}$$

(2)

The generation of turbulent kinetic energy due to buoyancy, ($G_b$) for constant density will be obtained as follow:

$$G_b = \nu_t \left[ \frac{\partial T}{\partial x_i} \frac{\partial T}{\partial x_i} \right]$$

(3)

The terms, $S_k$ and $S_\varepsilon$ are source terms. The $C_1$ will be defined as below:

$$C_1 = \max[0, 4.3, \frac{\eta}{\eta + 5}]$$

(5)

According to Eq. (4), the transport equation for $k$ is the same as the in the standard $k-\varepsilon$ model except for the model constants. Two important features of Eq. (4) are as below:

The first term doesn’t involve the $G_k$ term (like standard $k-\varepsilon$ model) and this property predicts the energy transfer much better than the standard model.

The second term in the right hand side of transport equation in Eq. (4) never vanishes although the $K$ vanishes and thus doesn’t have singularity.

The model for predicting the turbulent viscosity is also modeled as follow:

$$\nu_t = \frac{\mu_k}{\varepsilon}$$

$$C_\mu = \frac{1}{A_0 + C_\mu kU^*}$$

$$U^* = \sqrt{S_{ij} S_{ij} + \Omega_{ij} \Omega_{ij}}$$

(6)

$$\Omega_{ij} = \Omega_{ij} - 2 \varepsilon_{ijk} \omega_k$$

$$\Omega_{ij} = \Omega_{ij} - \varepsilon_{ijk} \omega_k$$

Where $\Omega_{ij}$ is the mean rate-of-rotation tensor viewed in a rotating reference frame with the angular velocity $\omega_k$. The constants are represented in Eq. (7).
\[ A_0 = 4.04 \quad A_k = \sqrt{6} \cos \phi \]
\[ \phi = \frac{1}{3} \cos^{-1} \left( \sqrt{6}W \right) \quad W = \frac{S_{ij}S_{jk}S_{ki}}{S^3} \]
\[ \ddot{S} = \sqrt{S_{ij}S_{ij}} \quad S_{ij} = \frac{1}{2} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \]

B. \( k-\omega \) Model

The next method used in this paper is Wilcox \( k-\omega \) model. From the comparison aspect, this model is stronger than the standard \( k-\varepsilon \) model in flows containing the deceleration and also separation flow due to adverse pressure gradient. It should be noticed that this model is powerful when the Re number is low. The constitutive Equations are as below:

\[ \rho \left( \frac{\partial \omega}{\partial t} + \frac{\partial u_j}{\partial x_j} \right) = (\mu + \frac{\mu_t}{\sigma_\varepsilon}) (\frac{\partial k}{\partial x_j} + \frac{\partial \varepsilon}{\partial x_j}) + \frac{1}{2} \frac{\partial}{\partial x_j} (\rho \varepsilon \omega) \]
\[ + C_1 \frac{\omega}{k} \left( G_k + C_1 (1-C_3) \right) \frac{\omega}{k} G_b - C_2 \rho \omega^2 \]

The constants are also illustrated in Tab.2.

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>( \sigma_k )</th>
<th>( \sigma_\varepsilon )</th>
<th>( \varepsilon_\mu )</th>
<th>( C_1 )</th>
<th>( C_2 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>2</td>
<td>2</td>
<td>0.99</td>
<td>0.555</td>
<td>0.8333</td>
</tr>
</tbody>
</table>

As it is clear in Eq. (8), the general form of transport equations are so similar to Eq. (1). And instead of using \( \varepsilon \), the \( \omega \) is replaced. So it is clear that the results will be nearly the same for the \( k-\varepsilon \) model and \( k-\omega \) model.

C. Reynolds Stress Model

The eddy viscosity models in the attached boundary layer flow are valid when just one element of Reynolds stress tensor is dominant (\( u'^2 \)). Therefore, the next model was dedicated to RSM model but with the same wall function for near wall regions as for standard \( k-\varepsilon \). This was done because of comparison with standard \( k-\varepsilon \) model and the importance of other elements of stress tensor for the case study involved in this paper.

The transport equation for turbulence kinetic energy is written as below:

\[ \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho u_i k) = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j} + \frac{1}{2} \frac{\partial}{\partial x_j} (P_H + G_{hi}) + S_k \right] \]

Where \( S_k \) is the source term, as it is clear, the Eq. (9) is the same as Eq. (1). But the difference is in the calculation of \( k \). The \( k \) obtained from Eq. (9) is valid wherever in the boundary condition and in any other cases the \( k \) would be obtained as Eq. (10).

\[ k = \frac{1}{2} u_i u_i' \]

The dissipation tensor would be followed as below:

\[ \varepsilon_{ij} = \frac{2}{3} \delta_{ij} \rho \varepsilon \]

And the scalar dissipation rate, \( \varepsilon \) would be also according to Eq. (12) same as for the standard \( k-\varepsilon \) model:

\[ P_H + C_{\varepsilon 3} G_{hi} \frac{\varepsilon}{k} \frac{\varepsilon}{k} S_{\varepsilon} \]

Where the \( S_{\varepsilon} \) is the source term (if any) and constants are listed in Table 3.

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>( \sigma_k )</th>
<th>( \sigma_\varepsilon )</th>
<th>( C_{\varepsilon 1} )</th>
<th>( C_{\varepsilon 2} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>0.82</td>
<td>1</td>
<td>1.44</td>
<td>1.92</td>
</tr>
</tbody>
</table>

The term, \( C_{\varepsilon 3} \) is also evaluated as a function of local flow direction like standard \( k-\varepsilon \) model as Eq. (13).

\[ C_{\varepsilon 3} = \tanh \frac{\psi}{\psi_0} \]

III. NUMERICAL METHOD

A (CFD) finite volume based commercial code (Fluent 6.3), was employed to discrete the momentum, Energy and turbulence transport equations for the case study involved in this paper. The schematic view of the channel is shown in Fig. (1).

![Fig. 1 Schematic view of channel and closer view with tetrahedral/hybrid meshes](image-url)
In all four methods the meshes were selected as tetrahedral/hybrid scheme and the $y^+$ criterion were satisfied in each method. The $y^+$ was satisfied between 30 to 50 for $k-\varepsilon$ and $k-\omega$ model, between 2 to 10 for $k-\varepsilon$ realizable and between 1 to 10 for RSM method, near the wall boundary regions. The boundary conditions for inlet and outlet were velocity inlet and pressure outlet, respectively. In addition the all of the walls temperature was fixed in 1200°K. The Re=20000 in all cases except the graphs involving the centerline velocity and root mean-square velocities with two other methods predicts the lowest values and RSM model is a low Reynolds model and $k-\varepsilon$ is suitable for high Re number (Re>50000). Indeed, this is not a strong rule and as it will be seen later, the $k-\varepsilon$ model has strong ability in predicting the thermal flux at the wall boundary. Another attractive matter is the behavior of RSM model in predicting the axial velocity; the plot for RSM shows more severe deviation from the mean velocity rather than two other models. The amplitude is high and seems to have difference in this matter with the experimental data but the value of maximum velocity reported by RSM model is somewhere closer to experimental data rather than two other models.

Order of the role of wall function. In fact, the RSM model had the most meshes in compare with three other models but near the wall region was treated by wall function. But the best results are reported by $k-\varepsilon$ realizable model.

One of the main reasons of three other models seems to be in rule and as it will be seen later, the $k-\varepsilon$ model has strong ability in predicting the thermal flux at the wall boundary. Another attractive matter is the behavior of RSM model in predicting the axial velocity; the plot for RSM shows more severe deviation from the mean velocity rather than two other models. The amplitude is high and seems to have difference in this matter with the experimental data but the value of maximum velocity reported by RSM model is somewhere closer to experimental data rather than two other models.

Other comparisons were done in strength of the models employed in the prediction of the heat transfer coefficient between two ribs. Regarding to this goal, a non dimensional Nu number which is defined as Eq. (14),

$$\frac{Nu}{Nu_0} = \left(\frac{\dot{q}^*}{T_s - T_b} \frac{D_b}{k} \right) \left(0.023 Re^{-0.8} Pr^4\right)$$

was applied to investigate the thermal flux between ribs 2-3. This definition is in accordance with the reported definition by Sewall [27]. In addition, due to high wall temperature and variation of specific heat capacity with temperature, a temperature dependent polynomial was considered as Eq. (15).

$$C_P = 1005.7 - 0.44T + 0.001407T^2$$

$$- 7.99 \times 10^{-7}T^3 + 1.9327 \times 10^{-10}T^4$$

The non dimensional Nu number is illustrated in Fig. (3).

As it is shown in Fig. (2), the $k-\varepsilon$ model in comparison with two other methods predicts the lowest values and RSM method has the best results. The main reason that the $k-\omega$ model predicts the velocity better than $k-\varepsilon$, seems to be because of natural characteristics of the model, i.e., the $k-\varepsilon$ model is a low Reynolds model and $k-\varepsilon$ is suitable for high Re number (Re>50000). Indeed, this is not a strong rule and as it will be seen later, the $k-\varepsilon$ model has strong ability in predicting the thermal flux at the wall boundary. Another attractive matter is the behavior of RSM model in predicting the axial velocity; the plot for RSM shows more severe deviation from the mean velocity rather than two other models. The amplitude is high and seems to have difference in this matter with the experimental data but the value of maximum velocity reported by RSM model is somewhere closer to experimental data rather than two other models.

### IV. RESULTS AND DISCUSSIONS

Many efforts and detailed measurements have been done in the field of turbine blade cooling and separation flow around different ribs, but a comparison of time consuming and academic approaches with industrial and practical ones is of interest in this paper.

Because of validation of current results, some plots like the centerline velocities, mean-square velocity at the centerline and Nu number between ribs 2-3 and 5-6 were validated with previous work by Sewall et all [27]. The centerline stream wise velocity of current work with the experiments done with Sewall is shown in Fig. (2).

$$\text{Fig. 2 The centerline stream wise velocity (Re=100000)}$$

$$\text{Fig. 3 Non dimensional Nu number between ribs 2 and 3}$$

According to Fig. (3), the predicted values of thermal flux by $k-\varepsilon$ model are better than the $k-\omega$ and RSM model. But the best results are reported by $k-\varepsilon$ realizable model. One of the main reasons of three other models seems to be in order of the role of wall function. In fact, the RSM model had the most meshes in compare with three other models but near wall region was treated by wall function. But the $k-\varepsilon$ realizable model employs a two layer approach and when there are some computational nodes in the viscose sub layer, takes advantage of them to predict more precise results.
on solid boundaries. This manner was repeatedly done between ribs 5 and 6 and results are demonstrated in Fig. (4).

![Fig. 4 Non dimensional Nu number between ribs 5 and 6](image)

As shown in Fig. (5), the same manner was observed and $k - \varepsilon$ realizable model had the best results.

After validation of numerical results with experimental ones in literature, some attractive characteristics like pressure drop along the channel and the velocity profile and kinetic energy among selected ribs were plotted.

The first graph is dedicated to pressure drop and is illustrated in Fig. (5).

![Fig. 5 Pressure drop along the channel](image)

As shown in Fig. (5), the $k - \varepsilon$ and $k - \omega$ model failed to predict the correct pressure drop along the channel centerline and two other models were approximately predicted the same value. One of the reasons of uncertainty of reported values by $k - \varepsilon$ and $k - \omega$ model was related to the steep of the plot. Due to existence of multiple ribs in the channel walls, the slight steep is not reasonable and the two other models manner seems to be in more accordance with reality.

One of the interesting properties of a turbulent flow is the behavior of kinetic energy among the ribs. Therefore the plot of kinetic energy was plotted between ribs 1, 2 and 5, 6 and after the last rib according to Figs. (6-8).

![Fig. 6 Plot of kinetic energy between ribs 1 and 2](image)

As shown in Figs. (6-8), the turbulent kinetic energy increases along the channel but this manner only is valid just before the last rib and after that the kinetic energy decreases dramatically. This shows the effect of ribs in making the severe turbulence among ribs. Another issue is related to the strength of the models in predicting the TKE. As seen, between ribs 1 and 2, the results of RSM and $k - \varepsilon$ non
equilibrium are so similar to each other and between ribs 5 and 6 the kinetic energy value reported by RSM, $k-\varepsilon$ and $k-\varepsilon$ non equilibrium was coincident. This behavior completely changes after the last rib. The results of RSM model have the similarity with the results of $k-\varepsilon$ realizable near the channel wall and have similarity with $k-\omega$ model near the centerline. Generally speaking, the best results seem to be reported by RSM model and weakest ones by $k-\omega$ model.

Due to better comprehension of behavior of turbulence, the velocity profiles are plotted among the above mentioned ribs as for kinetic energy.

As shown in Fig. (9), the turbulent flow is not yet formed and upper surface of plot is flat but by moving forward along the channel, the profile becomes completely parabolic as shown in Fig. (10). In addition the RSM model has reported the highest value rather than three other models.

As seen in Fig. (11), the velocity profile tends to become flat and this means the fully developed flow after the ribs. The fascinating matter is in the behavior of $k-\varepsilon$ realizable in which the velocity values near the channel wall are completely in accordance with the values of $k-\varepsilon$ and in the core region are completely matched with $k-\omega$ model. It seems that the best results are reported by $k-\varepsilon$ realizable model and results of RSM model are not so satisfactory.

It should be noticed that the velocity profiles are plotted in the line which has connected the left wall to the right one. If the graphs were suppose to be plotted in the line vertical to this line, no special behavior was found and a flat profiles were obtained for all of the situations and actually the formation of fully turbulent flow was not observable.

V. CONCLUSIONS

A (CFD) Finite Volume method was applied to investigate the effect of turbulence models in predicting the fluid flow and heat transfer in a three dimensional square-ribbed channel with the constant wall temperature. The results show that there is not a specific model to predict all of the turbulence properties accurately. The $k-\varepsilon$ realizable model was considered as the best model for predicting the heat transfer coefficient and after that the $k-\varepsilon$ model was the best, in addition, the $k-\omega$ and $k-\varepsilon$ realizable model were powerful in prediction of velocity profile, the RSM model was the best in predicting the kinetic energy

ACKNOWLEDGMENT

With especial thanks to Mr. Farajpour because of his guides for RSM method section.

REFERENCES


