Numerical Analysis of Heat Transfer and Pressure Drop in a Channel Equipped with Triangular Bodies in Side-By-Side Arrangement

E. Manay  
Department of Mechanical Engineering, Faculty of Engineering, Bayburt University, Bayburt, Turkey

S. Gunes, E.Akcadirci and V.Özceyhan  
Department of Mechanical Engineering, Faculty of Engineering, Erciyes University, Kayseri, Turkey

Abstract-The focus of presented numerical study is to investigate the effect of Reynolds number with respect to both heat transfer and flow characteristics in a channel equipped with two triangular bluff bodies in side by side arrangement. (SST) k-ω turbulence model is used for simulations and the second order upwind numerical scheme and SIMPLE (pressure implicit with splitting of operations) algorithm are utilized to discretize the governing equations. The flow is assumed two-dimensional and the calculations are performed for a Reynolds number range varying from 10,000 to 40,000 under steady state conditions. The calculations are carried out on a four-noded structured mesh near channel wall and a three-noded unstructured mesh near triangular bodies in order to have a better description of the boundary layer. The variation of Nusselt number, skin friction coefficient along the channel and Nusselt number versus Reynolds number were presented. The calculations are also compared to the results obtained by H. Chattopadhyay. The comparison shows that the numerical results are of good agreement with the results obtained from that of Chattopadhyay. H. It was concluded that at Re=40,000 the best achievement in Nusselt number was obtained as 463.

Keywords-Triangular bluff body, side-by-side arrangement, heat transfer enhancement.

I. INTRODUCTION

Bluff bodies are generally used to promote turbulence in channels by disturbing the flow. The studies over bluff bodies mainly consist of cylinders, flat plates, rectangular bars. The flow around bluff bodies has been the subject of many studies in the past. Flow interference caused by bluff bodies placed into the channel is responsible for several changes in the characteristics of heat transfer and flow. As known, there are several coherent structures for different applications according to many kinds of sizes, shapes, flow patterns, etc. respectively. In this study, to examine the effects of body geometry placed into the channel, dual triangles are studied. The studies made in this field showed that lots of parameters like geometry and geometrical arrangements are also affected on heat transfer and flow structure. This paper presents a detailed numerical study of the flow about a pair of triangles in side-by-side arrangement. There are many studies about bluff bodies and the detailed literature review is given below.

Y. Du et al.[1] investigated numerically the effect of the gap ratio of triangular bodies over flow characteristics for different gap ratios varying from 0.12 to 0.48 at Re=470,000. For confirming the observations from numerical study, experimental results using point-to-point method and particle-image velocimetry (PIV) measurements in a close wind tunnel were also carried out. In this study, two types of coherent structure are identified: Low gap ratio 0.12 and High gap ratio 0.22-0.48. The coherent structure is divided by the gap flow into two zones called the primary recirculation zone and the secondary recirculation zone. Results showed that the structure of small gap ratio is different from that of large gap ratio because the interaction between two zones relates to the gap ratio. The flow characteristics of wake and base-bled flow downstream of two bodies, with different cross-sectional geometries in side by side arrangement were studied by C.-Y. Wei and J.-R. Chang [2]. The two-body arrangements were comprised of flat plate and square cylinder, flat plate and circular cylinder, and square cylinder and circular cylinder. The results demonstrated that the characteristics of wake and base-bled flow are significantly related to the cross-sectional geometries of the bluff body. The gap flow tended to deflect toward the narrow-wake side downstream of the two body arrangement. L. Jian-zhong and his friends [3] analyzed the modification of flow by the combined effects of the rotation and the Reynolds number on the flow past two rotating circular cylinders in a side-by-side-arrangement. The analysis were performed using Particle Image Velocimetry (PIV) at a range of 425 ≤ Re ≤130, 0 ≤ s at one gap spacing ratio. The results showed that the vortex shedding was suppressed as rotational speed increased. The flow reached a steady state when the vortex shedding for both cylinders was completely suppressed at critical rotational speed. H. Chattopadhyay [4] studied in the turbulent flow regime up to the Reynolds number of 40,000 As a result, it was reported that the existence of a triangular body in a channel provides approximately 15% heat transfer enhancement.

The interaction among two spheres in tandem formation were studied for a Reynolds number of 300 using both steady and pulsating inflow conditions by L. Prahl et al. [5]. The results showed that the separation distance played a significant role in changing the flow patterns and shedding frequencies at moderate separation distances, whereas effect
on drag was observed even at a separation distance of 12 diameters. The effect of aspect ratio on heat transfer to a cylinder in cross flow of air has been studied experimentally by B.H. Chang et al. [6]. They reported that the heat transfer rates increased with decreasing aspect ratio at the centerplane on the rear of the cylinder. Consequently, the average Nusselt number correlations are presented that account for the effects of aspect ratio, tunnel blockage and free stream turbulence.

Based on the results summarized above, one finds that the relationship between Reynolds number and the biasing characteristic of the gap flow depends strongly on the geometry of the bluff body. Because many practical engineering problems are related to the flow field downstream of bluff bodies with different cross-sectional geometries, the present study focused on flow interaction near two triangular bluff bodies in side by side arrangement and was performed for a Reynolds number range varying from 10,000 to 40,000 under steady state conditions.

II. CFD ANALYSIS

Problem Description

In Figure 1, the main features of the test rig domain are shown schematically. The computational domain mainly consists of two-dimensional channel and dual triangular bodies placed in side by side arrangement. As seen in Figure 1, channel height (H) of 4B, the placement of the bluff bodies (S) was maintained at x = 8B, while the channel length was 36B. For preventing from the negative pressure effects at the outlet sections, the inlet section was selected long enough to get a fully developed flow.

![Figure 1: Computational domain and the triangular bodies in side by side arrangement.](image)

Numerical Procedure

For determining the velocity and temperature distributions, CFD calculations made by the aid of the computational fluid dynamics (CFD) commercial code of FLUENT version 6.1.22 [9] are performed depending on the numerical model, boundary conditions, assumptions, and numerical values. In all the numerical calculations, segregated manner was selected as solver type, due to its advantage which helps to prevent from convergence problems and oscillations in pressure and velocity fields of strong coupling between the velocity and pressure by using, (SST) k-ω turbulence model is used for simulations and the second orderupwind numerical scheme and SIMPLE algorithm being more stable and economical in comparison with the other algorithms are utilized to discretize the governing equations. The converging criterions are thought as $10^{-6}$ for the energy and $10^{-4}$ for other parameters. In momentum and continuity equations, the thermophysical properties are thought as constant, and, the flow is assumed two-dimensional, steady continuity.

Continuity conservation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \cdot \vec{V}) = 0$$  \(1\)

Momentum conservation:

$$\frac{\partial (\rho \cdot \vec{V})}{\partial t} + \nabla \cdot (\rho \cdot \vec{V} \cdot \vec{V}) = \rho g - \nabla P + \nabla \cdot (\mu \nabla \vec{V})$$  \(2\)

Energy conservation:

$$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho \cdot \vec{V} \cdot (\rho E + p)) = \nabla \cdot (k_{eff} \nabla T + (\vec{P} \cdot \vec{V}))$$  \(3\)

Where

$$\vec{P} = \mu \left( \nabla \vec{V} + \nabla \vec{V}^T - \frac{2}{3} \nabla \cdot \vec{V} \right)$$  \(4\)

$$E = h - \frac{P}{\rho} + \frac{v^2}{2}$$  \(5\)

and $k_{eff}$ is the effective conductivity.

F.R. Menter [7] developed the shear-stress transport (SST) k–ω model by blending k–ω model and k-ε model formulation effectively. In literature, there exist various numerical studies concerning with channel flow which are performed by using (SST) k-ω turbulence model (S. Eiamsaard and P. Promvonge [8], Kamali and Binesh [10], Nasiruddin and Siddiqui [11] investigated the heat transfer enhancement in a heat exchanger tube by installing a baffle numerically and he reported the (SST) k-ω model predicts successfully and accurately the flow modification due to the baffle according to other turbulent models. The (SST) k-ω model is able to calculate speedly two-dimensional flow and also predict the interactions with the wall. This model is also advantageous because the model equations behave compatible in both the near-wall and far-field regions.

In the derivation of the k–ω model, the flow is assumed to be fully turbulent. The Shear-Stress Transport (SST) k-ω model, the turbulence kinetic energy, k, and the specific dissipation rate, ω, are obtained from the following transport equations:
\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G_k - Y_k
\]

(6)

and

\[
\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega
\]

(7)

In these equations, \( G_k \) represents the generation of turbulence kinetic energy due to mean velocity gradients. \( G_\omega \) represents the generation of \( \omega \). \( \Gamma_k \) and \( \Gamma_\omega \) represent the effective diffusivity of \( k \) and \( \omega \), respectively. \( Y_k \) and \( Y_\omega \) represent the dissipation of \( k \) and \( \omega \) due to turbulence and \( D_\omega \) represents the cross-diffusion term. The calculation of all of the above terms is given detailed in FLUENT 6.1.22 [9].

Two parameters of interest for this study are the skin friction coefficient and the Nusselt number. The skin friction coefficient \( C_f \) is defined by

\[
C_f = \frac{\tau_w}{\frac{1}{2} \rho U_m^2}
\]

(8)

The heat transfer performance is evaluated by Nusselt number which can be obtained by the local temperature gradient as:

\[
Nu = -\frac{\partial T}{\partial Z}
\]

(9)

The average Nusselt number can be calculated as follows:

\[
Nu_{av} = \int \frac{Nu_x \, \delta x}{L}
\]

(10)

where \( L \) is the length of computational domain. The friction factor is determined from:

\[
f = \frac{\Delta P}{\frac{1}{2} \rho U_m^2 \frac{L}{H}}
\]

(11)

in which \( \Delta P \) is pressure difference between the channel inlet and exit:

Boundary Conditions

The solution domain of the considered 2D channel flow is geometrically quite simple, which is a rectangle on the \( x-z \) plane, enclosed by the inlet, outlet and wall boundaries. The working fluid in all cases is water. The inlet temperature of water is considered to be uniform at 300 K. On walls, no-slip conditions are used for the momentum equations. A constant surface temperature of 400 K is applied to the bottom wall of the channel. The upper wall is assumed to be adiabatic.

Uniform velocity is imposed to inlet plane and the Reynolds number varies from 10000 to 40000. The outlet boundary condition is natural condition which implies zero-gradient conditions at the outlet.

III. RESULTS

In this work, the used numerical method to obtain results is validated with the study of Chattopadhyay [9] in which the augmentation of heat transfer in a channel using a single triangular prism is investigated. The results of Nusselt number and skin friction coefficient obtained from CFD analyses are compared with the results obtained from Chattopadhyay [9]. Figures 2(a) and 2(b) show comparison between the results of the used CFD model and Chattopadhyay [9]. As observed from these figures that, there is a good agreement between the results of the used numerical model and Chattopadhyay [9]. These results give confidence that the numerical method used is accurate.
The distribution of local Nusselt number along the channel length is shown for different Re numbers is illustrated in Fig. 3. The local Nusselt number takes a local maximum at the placed position of the upstream body. After the peak towards to the exit the Nusselt number decreases strongly due the effect of the periodically shedding vortices do not occur from the bodies for all cases. Separated shear layer from downstream body and impinging of vortex formed from upstream body strongly increases heat transfer on downstream.

However, the peak becomes obvious with increasing Reynolds number. As expected local Nusselt number increases with increasing Reynolds number, the heat transfer enhances especially for upstream flow region concerning with the generation of vortices due to the triangular bodies as well as the role of turbulence in better mixing brings in the enhancement in heat transfer from the channel wall.

On the other hand, as expected, these bodies cause a significant fluid friction as well, in comparison with the smooth channel. The increase in local Nusselt number results in an increase in pressure drop, the friction factor increases with increasing Reynolds number due to the fact that interaction between the bodies placed as side by side disturb the entire flow field and cause more friction than the smooth channel. The local skin friction coefficient distributions on the bottom channel wall are shown in Figure 4 for different Reynolds numbers. Similarly with local Nusselt number, the local skin friction coefficient takes a maximum at the placed position of the triangular bodies in side by side arrangement. In Fig. 4, the local distribution of the skin friction coefficient for smooth channel, Re=10000 is also shown for comparison. The magnitude of skin friction is the highest at Re=40000. It is evident from Figure 4 that the local skin friction coefficient increases with increasing Reynolds number when the flow encounters the blockages created by equilateral triangular bodies.

IV. CONCLUSIONS

The local Nusselt number and skin friction coefficient take a local maximum at the placed position of the upstream body, and at the placed position of the downstream body. The heat transfer enhances especially for upstream flow region concerning with the generation of vortices due to the first body as well as the role of turbulence in better mixing brings in the enhancement in heat transfer from the channel wall.

The increase in local Nusselt number results in an increase in pressure drop, the friction factor increases with decreasing spacing between the bodies due to the fact that close interaction between the bodies disturb the entire flow field and cause more friction.

V. ACKNOWLEDGMENTS

Authors would like to thank for the financial support of the TUBITAK (The Scientific and Technological Research Council of Turkey) under the contract: 107M508.

REFERENCES


Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>base of the triangular body</td>
</tr>
<tr>
<td>C_f</td>
<td>skin friction coefficient</td>
</tr>
<tr>
<td>f</td>
<td>friction factor</td>
</tr>
<tr>
<td>H</td>
<td>height of the computational domain</td>
</tr>
<tr>
<td>k</td>
<td>kinetic energy</td>
</tr>
<tr>
<td>L</td>
<td>length of domain in x direction</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>P</td>
<td>Pressure</td>
</tr>
<tr>
<td>S</td>
<td>location of the triangular bodies</td>
</tr>
<tr>
<td>U_m</td>
<td>mean velocity component in x direction</td>
</tr>
</tbody>
</table>

Greek symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ρ</td>
<td>Fluid density</td>
</tr>
<tr>
<td>τ</td>
<td>shear stress</td>
</tr>
<tr>
<td>ω</td>
<td>specific dissipation rate</td>
</tr>
</tbody>
</table>

Subscripts

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>m</td>
<td>mean</td>
</tr>
<tr>
<td>s</td>
<td>Smooth channel</td>
</tr>
<tr>
<td>tb</td>
<td>triangular body</td>
</tr>
</tbody>
</table>