Computational Investigation of Turbulent Flow Development in 180° Channel with Circular Cross Section

Md. Safayet Hossain, Md. Ishtiaque Hossain

Abstract—The present study focuses on the analysis of flow characteristics inside a 180° bent channel by adopting two distinct RANS model namely, Realizable k-± and Reynolds Stress Model (RSM). The computation results, obtained from both case study have been validated against experimental data at different cross-sections throughout the bend region and downstream tangent for velocity distribution. The anisotropic behavior of turbulent flow was illustrated for both case study inside the bend region and it has been established that after 3° the flow gradually became more intense at the outer core. Pressure coefficient throughout the u-channel was depicted for both turbulence model and a characteristic feature has been obtained. Due to centrifugal force and high inlet Reynolds number, a pair of counter-rotating Dean vortices were constructed at different stations inside the bend region. From both demonstrations, it was revealed that, Realizable k-± model provided relatively better approximation.

Index Terms—Dean Vortices; 180° Bent Pipe; Turbulent Flow; Realizable k-± Model; RSM; Pressure Coefficient.

I. INTRODUCTION

Fluid flow in curved pipe has a wide range of industrial application such as HVAC appliance, Turbomachinery, heat exchangers, CANDU type reactor, High Temperature Gas Cooled Reactor (HTGR), Pressurized Water Reactor (PWR), torus reactor along with providing more flexibility in pipeline network. Analysis of fluid flow in pipe bend also plays a significant role in medical science, as the flow pattern is similar to the blood flow in vain or artery.

A substantial amount of work has been done on developing turbulent flow in 90° pipe bends by means of experimental and numerical methods [1]-[15]. Most of these works has been solved by RANS equation and it was established that RNG k-± model provides relatively better approximation along with being relatively cost-efficient. Usually in a bend channel, a fluid starts to be influenced at the bend inlet because of the sudden change in flow path. Due to the sudden conversion of the path from straight to a curvilinear region a centrifugal force act on that particular curved section and the working fluid passing through the section changes its direction. For the change of its direction secondary cross-stream flow reversal occurs whereas variation in the Reynolds number as well as curvature ratio influence the formation of the secondary flow [1]-[4]. While experimenting with the bend pipe design strategy, the effects curvature ratio is analyzed in several experimental as well as numerical study and it has been substantially established that bringing changes in the curvature ratio provide significant changes in the fluid flow. Depending upon the curvature ratio, a notable pressure drop will occur throughout the bent tube [5]-[8] and [14]. In the analysis of bend pipe, flow separation plays a vital role on the efficiency of the pipe besides secondary flow, pressure and velocity distribution. This flow separation phenomenon is directly a function of curvature ratio as well as bulk velocity which was demonstrated by numerous investigations [3] and [9]-[11]. Strong curvature ratio could result in a significant separation region. A rough pipe with several repeated ribs inside of it along with the possible presence of guide vane, could drastically affects different characteristics of fluid flow [13]-[14].

Although a number of experimental work on 90° pipe bends has been validated numerically, there has not been much work done on 180° channel illustrating the flow characteristics by means of numerical method. Similar flow characteristics like, generation of cross flow reversal at downstream tangent, stronger flow on the outside wall at the bend region was found from the numerical study of u channel. Azzola and Humphrey took assistance of Laser-Doppler Measurement to calculate the longitudinal and circumferential velocity components of a developed turbulent flow through a 180° strongly curved pipe [16]. They also checked the velocity profile and turbulent intensity at different cross-sections throughout the problem geometry. According to their study after 90°, the mean longitudinal velocity component changed a little but a fully developed state was never achieved by the circumferential component. Some authors alongside Lacovides and Launder investigated [21] the behavior of the turbulent flow using Standard k-± model and then the calculated results showed a good agreement with their experimental work. Cuming [19] analyzed a fully developed laminar flow and for equal radius of curvature to diameter ratio and dimensionless stream wise pressure drop, the ratio of the relative intensity of the secondary flow in a duct of square cross section to that in a pipe of circular cross section is 2.47 at the center line location. Rowe conducted an experiment for the turbulent flow of 180° pipe. He concluded that at about 30°, secondary flow is the most intense and the curved flow is essentially fully developed past 90° [20]. He demonstrated the secondary cross flow reversal along the bend symmetry plane between 90° and X/D=5. Sugiyama et al. [22] examined the turbulent flow separation and the possibility of improving the wall function method in the separated flow region by replacing the log-law velocity profile for a smooth wall with that for a rough wall. RSM model and rectangular
duct with a sharp 180° turn were chosen for the investigation. Sugiyama and Hitomi [17] used the experimental apparatus of proposed by Sudou and Takami for the numerical analysis of the turbulent flow through a 180° curved pipe with an algebraic Reynolds stress model. They concluded a rigorous investigation on the behavior of anisotropic turbulent characteristics and the distribution of Reynolds stresses. They managed to obtain the location of maximum streamwise velocity formed at circumferential region. Guan and Martonen used FIDAP to examine the axial and secondary motions in both the developing and fully developed regions of flow [18]. After validation of their results with the experimental data they suggested that, flow patterns and their commensurate effects on particle and gas transport can be simulated by the modeling technique.

Turbulent flow is highly anisotropic in nature. In each scenario, any or all of the turbulent properties could show unique behavior. The Dean motion, for starter, will greatly affect the streamwise flow with higher Reynolds number at the bend region. Due to its large category of behavioral change, the present study constrained its focus to analyze the velocity distribution and formation of secondary flow pattern throughout the bent channel. To validate the computational data, it is compared with an experimental work [16]. In order to do so, two different RANS model, namely, Realizable k-ε and Reynolds stress model were used to further cross-check the reliability of computational study. Later, results obtained from both case studies were compared to illustrate the pressure coefficient along the problem section.

II. PROBLEM DEFINITION

Fig. 1a shows the schematic diagram of the basic component of the experimental setup of Azzola and Humphrey [16]. Two pipes having length of 54.7D are at attached as the upstream and downstream tangent with a bent section. The radius of bend mean curvature (Rc) to pipe diameter is 3.375D. The whole channel has a constant circular cross-section with a diameter (D) of 4.45 cm. The associated Dean number (De) is about 22000, when the, De = Re√D/(2Rc). To facilitate the CFD modeling, the arrow direction in Fig. 1a was defined as the positive direction in this study. At the upstream tangent channel, XD=54.7 is the starting point of the main flow.

III. NUMERICAL METHODOLOGY

Although the length of the downstream tangent was set as 54.7D in the experiment, it was considered 20D in the current study to attenuate some computational effort. As the wall thickness of the experimental apparatus [16] is almost negligible (±0.03 cm), it was ignored in is study to refine the geometry which in turns, also alleviated the computational effort. The inlet Reynolds number was set at 5.7E04, which was same as the experiment. At the inlet, turbulent intensity, which can be estimated as, I = 0.16Re°(1/8), was set as 0.041 based on the hydraulic diameter. Since this problem condition could be considered as high Reynolds number model, the standard wall function (SWF) method was employed. And this restricts the y* to a certain range. But the SWF does not resolve the boundary layer, henceforth it’s mostly valid for the core region of turbulent flow. Which means, any derived wall region variables like wall shear stress or dynamic pressure, could represent crude values. For both case study, a second-order upwind discretization scheme was used for the discretion of the governing equations. This ensure more accuracy, stability as well as convergence. SIMPLE algorithm described by Patankar and Spalding [25] was used for pressure-velocity coupling. In both case study, the under relaxation parameter was set as 0.8 to facilitate the convergence to a more reasonable approximation.

A. Governing Equations and Turbulence Modelling

In this analysis, three dimensional Reynolds Averaged Navier–Stokes (RANS) equations are solved using the segregated implicit solver [23] while an incompressible fluid with constant properties is considered. Equation (1) (2) describe the continuity and momentum respectively.

\[
\frac{\partial p}{\partial t} + \frac{\partial (p\vec{u}_i)}{\partial x_i} = 0
\]  

\[
\frac{\partial}{\partial t}(\rho \vec{u}_j) + \frac{\partial}{\partial x_i}(\rho \vec{u}_i \vec{u}_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}\left[\mu \left(\frac{\partial \vec{u}_i}{\partial x_j} + \frac{\partial \vec{u}_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial \vec{u}_k}{\partial x_k}\right)\right] + \frac{\partial}{\partial x_i}(-\rho \vec{u}_i \vec{u}_j)
\]  

k-ε model is probably the most widely used turbulence model in industrial application because of its cost efficiency and reasonable accuracy. k-ε model solve two transport equation for turbulent kinetic energy and dissipation rate respectively. It’s a semi-empirical model that assumes the flow is fully turbulent. In this approach, Reynolds stresses are modeled based on Boussinesq hypothesis which is quite reasonable for channel flows. The Realizable k-ε model contains an alternative formulation of eddy viscosity. It has been found from previous case study, that the performance of this model is substantially better than its predecessor Standard k-ε model. Detailed derivations of transport equations are provided by Shih et al. [26].

Fig. 1. Schematic representation of problem geometry and sample view of computational grid.
The Reynolds Stress Model (RSM) is a higher order second moment closure turbulence model and it is the most complete turbulence model. It discards the eddy viscosity hypothesis and directly solve the transport equations for Reynolds stresses, which means, in total it solves seven transport equations in a three dimensional problem strategy. This model is the more advantageous choice when the flow is highly anisotropic, for example when strong swirling flow occurs. However, this model will require much greater time to converge the solution as it will solve additional transport equations for Reynolds stresses and so it is discernible that it demands a relatively heavier hardware support which turns it into a potentially cost-intensive model. Detailed calculation on the closure equations can be found in [27]-[29].

B. Grid Configuration

A completely structured grid was generated for this study with about 1.7 million hexahedral elements. The computational domain is divided into several subdomains. The grid was carefully designed to cluster a greater number of elements into the bend region to obtain more explicit prediction from that area. Primarily a completely structured H-grid mesh was generated as shown in Fig. 1b which is kept same throughout the whole channel. This grid resemblance the mesh designed by Sugiyama and Hitomi in their work [17]. But this method, produced a highly skewed mesh (>0.9) on four corners as marked in Fig. 1b where vertices of the hexahedral blocks meet the curved surface. Afterward investigating the mesh quality statistics, it has been found that the poorest elements were situated throughout the whole geometry on those four corners.

The equiangular skewness is based on the deviation from a normalized equilateral angle and can be defined as, 
\[
\text{max}[(\theta_{\text{max}} - \theta_{\text{e}})/(180 - \theta_{\text{e}}), (\theta_{\text{e}} - \theta_{\text{min}})/\theta_{\text{e}}]
\]
was checked to compare the mesh quality. Here, \(\theta_{\text{max}}\) is the largest angle in face or cell, \(\theta_{\text{min}}\) is the smallest angle in face or cell and \(\theta_{\text{e}}\) is the angle for equiangular face or cell. To resolve this issue, an O-grid topology (see Fig. 1c) was adopted in the present study, which results in producing more elements but reduced the skewness at a much lower
value (see Fig. 2b). This method rearranged the elements as an annular shaped design.

The $y^+$ is a non-dimensional entity, which determines whether the influences in wall adjacent cells are laminar or turbulent [24] and the term $y^+$ can be defined as, $y^+ = \frac{u_\tau y}{\nu}$. Here, $u_\tau$, $\nu$ and $y$ is the friction velocity, kinematic viscosity and first cell height respectively. Whether it's relatively easy to control the $y^+$ along the wall throughout the whole geometry in case of 2D geometry, it's become more difficult in case of 3D geometry. This complexity increases with having cylindrical wall and curvilinear region. As mentioned earlier, a wall function was employed in this study. The required $y^+$ value for SWF lies in between 30 and 300 whereas in the present study it was restricted in between 35 and 70 (see Fig. 2a) for both case study.

IV. VALIDATIONS AND DISCUSSIONS

The results obtained from this numerical study is compared with experimental data (see Fig. 3) at the bend region and downstream tangent. The local velocity sample obtained from different cross-section is normalized by the inlet bulk velocity. The results obtained from both case study, has been well validated with the experiment. The predicted data, up to 45° is almost similar to the experimental data. At 90°, the RSM model showed a slight discrepancy, whereas k-$\epsilon$ model provided almost accurate approximation. Continuing from $\theta=90^\circ$ to the downstream tangent, this discrepancy found from RSM model, started to diminish. It has been found that, the k-$\epsilon$ model throughout the bend region, gives slightly better prediction. In the experiment [16], it was explored that, after $\theta=90^\circ$, the streamwise flow became strongest at the outer wall. After performing a more in-depth investigation, it was revealed that, the circumferential flow distribution throughout the downstream tangent, became constant which validates the experimental investigation [16].

The contour plots of different circumferential cross-section illustrate the velocity distribution with pressure coefficient at the bend region. Each of these contour plots (see Fig. 4 and 5) is analogous to the left to right side of the curved channel whereas left and right side are indicated as inner and outer wall respectively in this work (see Fig. 1). Fig. 4 and 5 show variations in streamwise mean velocity at different circumferential sections inside the bend region for both turbulence models. And Fig. 6 illustrates the variation in streamwise mean velocity on the central plane for both turbulence models. After leaving the upstream tangent, at $\theta=3^\circ$, the flow is relatively more intense at the inner wall. Both of the case studies showed the similar flow pattern at this station and, Azzola et al. [21] also obtained the same behavior. Logically, at this station, maximum pressure coefficient is produced at the outer wall of 45° section.

After leaving the $\theta=45^\circ$ station, the core flow start to deform heavily. Similar phenomena were found in [17] and [21]. This was happened due to formation of secondary flow. A slight deviation in flow pattern was found at $\theta=90^\circ$ station. At this station, the contour line from both case study differs from each other and they also differ from the numerical work of Azzola et al [21] and Sugiyama [17]. Now this could happen for a number of reasons. In any numerical analyses, the accuracy of the results depends on various factors such as geometry refinement, grid independency study, convergence criterion, discretization scheme employed, wall treatment applied, turbulence model used, appropriate boundary condition set along with other factors. Even the solver itself plays an important role as different code could use different sorts of algorithm. For example, Fluent and CFX use cell-centered and vertex-centered formulations respectively.

Fig. 4. Distribution of velocity and pressure coefficient at different bend section under k-$\epsilon$ model.
Although none of these works [17] and [21], showed the exactly identical flow pattern with the present case study, but the basic concept is same. From this section, the flow largely starts to deform from inward to outward direction. Fig. 6 shows that streamwise mean flow is slightly dissimilar to each other, but the velocity gradient is higher at the outer wall.

Starting from $\theta=90^\circ$ to bend outlet the inner core flow was gaining a slight increase in velocity as shown in Fig. 6. This happens with the increase in strength of induced secondary flow at the inner region. At the downstream tangent, after $X=+2D$ distance, the core flow becomes relatively stable as shown is Fig. 3 and 6.

Fig. 1. Pressure distribution along the bent channel.

Fig. 5. Distribution of velocity and pressure coefficient at different bend section under RSM model.

Fig. 6. Mean velocity distribution under Realizable $k$-$\epsilon$ (a) and Reynolds stress model (b) at the central plane.

Fig. 7 explains the pressure distribution at the inner wall, outer wall and the centerline of the bent channel along the streamwise flow. Results obtained from both case studies show the similar pressure distribution throughout the bent channel. Both at the upstream and downstream tangent pressure coefficient ($C_p$) decrease uniformly along the direction of axial flow. While exiting the upstream tangent, at the bend inlet, $C_p$ started to rise at outer wall and on the other hand it started to decrease at the inner wall. A characteristic feature, found from the both case study that, at the outer wall, the pressure coefficient continues to increase.
up to 45°. After 135°, the pressure coefficient starts to rise at the outer wall and up to X/D=+1, this phenomenon continues. Similar pattern has been found on both the 90° elbow bend [4,8,14] and u-channel [17].

Because of the existence of the curved channel, the fluid is affected by a centrifugal force starting from bend inlet. This is why pressure started to increase at the outer wall (see Fig. 7). With the increase in streamwise velocity, a pair of counter-rotating vortices are generated. Fig. 8 and 9 show the formation of Dean vortices under different turbulence models across the bend region and downstream tangent at different cross-sectional stations.

Starting from the bend inlet, θ=0°, a secondary flow uniformly starts to generate at the inner core due to the influence of curvature. With the increase in pressure at the outer wall, low velocity fluid starts to develop a pair of counter-rotating Dean vortices at the symmetrical sections of the bent channel (see Fig. 8 and 9). At θ=45°, fully developed Dean vortices are found. With the increase in centrifugal force, this effect continues up to θ=90°. Up to this station, the Dean motion pattern found from k-ε model is in good agreement [4]. After θ=90°, the streamwise velocity slightly starts to increase at the inner core, hence the intensity of secondary flow starts to diminish and along with that, the Dean vortices start to shrink. After θ=135°, the low velocity fluid is started to delocalize in unique manner under both turbulence models which in turns, differently affects the secondary flow pattern.

![Fig. 8. Formation of Dean Vortices under Realizable k-ε (a) and Reynolds stress model (b) at different stations inside the bend region.](image)

![Fig. 9. Formation of Dean Vortices under Realizable k-ε (a) and Reynolds stress model (b) at different stations inside the downstream tangent.](image)

V. REMARKS

A numerical study has been conducted for developing turbulent flow in a 180° bent channel by utilizing Realizable k-ε and Reynolds stress model. Computational results are validated against experiment [16] and they also showed good agreement with [17] and [21]. The pressure and velocity distribution along with secondary flow formation throughout the bent channel and outlet cylinder are studied in this work.

A pair of unique characteristic fact is established from this study. From bend inlet to the outlet of downstream tangent, maximum streamwise velocity is formed at the outer core. This flow becomes stronger at the same vicinity of downstream tangent. At the curvilinear zone, with the increase in circumferential velocity, the trace of cross-stream flow reversal becomes stronger. The flow distribution at each cross-sectional station is found almost identical at both case study. But, at 90° station, a slight disparity in radial direction, is found in between the case studies.
The pressure coefficient decreases throughout the straight inlet and outlet cylinder and also along the centerline with the bulk velocity. But at the outer wall of bent channel, the pressure suddenly started to rise and this continued up to 45° cross-sectional station. After that, the pressure coefficient started to decrease and at, θD=±1° position, it became identical with inner and outer wall. While on the other hand, the pressure coefficient started to decrease and it reached its lowest value at about 135° cross-sectional station. This phenomenon occurred due to the variation in centrifugal force. After that, it started to increase and at, θD=±1° position, it became identical with the pressure coefficient of the rest of the outlet cylinder. The pressure coefficient obtained from both case study, showed almost identical behavior except at 90°, very minor deviation was seen. This finding, confirmed the disparity seen in circumferential velocity distribution at 90° station.

It is found from the numerical study, under the impression of high inlet Reynolds number, a secondary flow pattern will be induced in any curved channel. In a u-channel, with the increase in centrifugal force, the Dean vortices continue to develop toward the inner core up to θ=90° station. After that, they start to contract and disappear completely at θ=180°. The secondary flow pattern is very complicated in nature, moreover there is no surefire way to determine a specific turbulence model for a specific types of problem. In each unique case, a different turbulence model could predict better result. So with a slight oscillation in inlet parameter or grid construction mechanism, the flow pattern could represent a very different result.

This study had kept down its investigation to pressure and velocity distribution along with the propagation of secondary flow throughout a u-channel. Additional work can be done on flow separation or secondary flow mechanism under different turbulence model. A more in-depth investigation is required by changing the curvature ratio and Reynolds number to fully understand the nature of flow. Also the comparison in degree of flow separation in between circular and quadrilateral cross-section in a u-channel is kept for future work.

REFERENCES

Md. Safayet Hossain was born in Bangladesh. He obtained BSc in Mechanical Engineering from Chittagong University of Engineering & Technology in 2017. His field of interest includes aerodynamics, computational fluid dynamics and renewable energy. He has published a number of academic journal in the past.

Md. Ishtiaque Hossain was born in Bangladesh. He accomplished BSc in Mechanical Engineering from Chittagong University of Engineering & Technology in 2017. His research interest is in computational fluid dynamics, heat transfer, renewable energy, sustainable energy and thermodynamics.