Prediction of secondary flow structure in turbulent Couette-Poiseuille flows inside a square duct

Wei Lo[a], and Chao-An Lin[a]*

[a]Department of Power Mechanical Engineering, National Tsing Hua University, Hsinchu 300, TAIWAN

1. Introduction

The turbulent Poiseuille or Couette-Poiesuille flows inside a square or rectangular cross-sectional duct are of considerable engineering interest because their relevance to the compact heat exchangers and gas turbine cooling systems. The most studied flow is the turbulent Poiseuille type inside a square duct and is characterized by the existence of secondary flow of Prandtl's [1] second kind which is not observed in circular ducts nor in laminar rectangular ducts. The secondary flow is a mean circulatory motion perpendicular to the streamwise direction driven by the turbulence. Although weak in magnitude (only a few percent of the streamwise bulk velocity), secondary flow is very significant on the momentum and hear transfer.

Fukushima and Kasagi [2] studied turbulent Poiseuille flow through square and diamond ducts and found different vortex pairs between the obtuse-angle and acute-angle corners in the diamond duct. A pair of larger counter-rotating vortices presented near the corners of acute-angle with their centers further away from the corner. Heating at the wall in a square duct was found to have evident effect on the secondary flow structure as well ([3] [4] [5]). Salinas-Vázquez and Métais [5] simulated a turbulent Poiseuille flow in a square duct with one high temperature wall and observed that the size and intensity of secondary flow increase in tandem with the heating. In the corner formed by heated and non-heated walls there exhibited a non-equalized vortex pair with vortex near the heated wall much larger than the other. Study that combined the bounding wall geometry and heating on the secondary flow in a non-circular duct can be directed to Salinas-Vázquez et al. [6]. By numerically studied the heated turbulent flow through a square duct with one ridged wall, new secondary flow were revealed near the ridges. Rotation around an axis perpendicular to the streamwise flow direction were also found to greatly modified the secondary flow structure in a LES calculation conducted by Pallares and Davidson [7].

The above investigations have implied that with careful manipulation, the secondary flow is very much promising on enhancement of particle transport or single phase heat transfer in different industrial devices. They also demonstrated that the turbulence gen-

---

*Email address: calin@pme.nthu.edu.tw

erated secondary flow in non-circular ducts could be modified by bounding wall geometry, heating and system rotation. However, little is known about the effect of moving wall on the secondary flow structure. In the present study, the assessment to this question is investigated by simulating the turbulent Couette-Poisueille flows in a square duct. One of the bounding walls is moving at the positive streamwise direction with different speed. The secondary flow patterns are first studied through mean velocity contours and vectors. Correlations between the vortex position and global flow parameters then been examined for an easy estimation of the secondary flow structure.

2. Governing Equations and Modeling

The governing equations are grid-filtered, incompressible continuity and Navier-Stokes equations. In the present study, the Smagorinsky model[8] has been used for the sub-grid stress(SGS). A Van Driest damping function accounts for the effect of the wall on sub-grid scales is adopted. Although other models which employed dynamic procedures on determining the Smagorinsky constant (C_s) might be more general and rigorous, the Smagorinsky model is computationally cheaper among other eddy viscosity type LES models. An investigation carried out by Breuer and Rodi [9] on the turbulent Poiseuille flow through a straight and bent square duct has indicated that, the difference between obtained turbulence statistics of dynamic models and Smagorinsky model was only negligible.

3. Numerical and parallel Algorithms

A semi-implicit, fractional step method proposed by Kim and Moin [10] and the finite volume method are employed to solve the incompressible Navier-Stokes equations. Spatial derivatives are approximated using second-order central difference schemes. The non-linear terms are advanced with the Adams-Bashfoth scheme in time, whereas the Crank-Nicholson scheme is adopted for the diffusion terms. The grid size employed in the present study is (128x128x96) in the spanwise,normal, and streamwise direction, respectively.

In the present parallel implementation, the single program multiple data (SPMD) environment is adopted. The domain decomposition is done on the last dimension of the three dimensional computation domain due to the explicit numerical treatment on that direction. The simulation is conducted on the HP Integrity rx2600 server(192 Nodes) with about 80 percent efficiency when 48 CPUs are employed in the computation.

4. Results

Schematic picture of the flows simulated is shown in Figure 1, where D is the duct width. We consider fully developed, incompressible turbulent Couette-Poiseuille flow in a square duct where the basic flow parameters are summarized in Table 1. Reynolds number based on the bulk velocity (Re_{bulk}) is kept around 10000 for all cases simulated and the importance of Couette effect in this combined flow field can be indicated by the ratio $r = (W_w/W_{Bulk})$ and $-\frac{D}{\rho W_w^2}\frac{\partial P}{\partial z}$. The secondary flow structure and their effect on transferring the mean streamwise momentum can be clearly identified in Figure 3. Below $y/D = 0.5$ the secondary flow structure is similar to that in turbulent Poiseuille flow which two contour rotating vortices exhibit near the corner. However, vortex above the bottom corner bisector is distorted by the large vortex near the top corners ($y/D > 0.5$). From Figure 3 secondary flow vector shows an inflow along the corner bisector at bottom half of the duct. Therefore, the advection effect of secondary flow near the bottom of the duct ($y/D < 0.5$) is to increase the momentum or energy in the corner region. Secondary flow near the top of the duct($y/D > 0.5$), consists of one dominate vortex and a relatively small vortex near the moving wall. With the speed of moving wall increased from case CP1 to CP4, the secondary flow gradually blocks the high momentum core fluid from the corner region. By measuring the angle between the top two vortices relative to the horizontal x axis, it is interesting to note that a linear relation exits between the angle and the parameter r, as shown in Figure 2. By the least square fit of the four simulation data a relation of $\Theta = -13.56 + 38.26r$ is obtained. Distance between the vortex cores of each vortex pair is also observed to remain approximately constant.

In summary, numerical investigation of the turbulent Couette-Poiseuille flow has been conducted to study the effect of moving wall on the secondary flow structure inside a square duct. We demonstrated that the secondary flow could be greatly affected by the speed of moving wall. An interesting correlation between the parameter $r = (W_w/W_{Bulk})$ and vortex core position near the corner formed by the moving and stationary wall is revealed in the present results.

REFERENCES

1. L. Prandtl, Uber die ausgebildete turbulenz, Verh. 2nd Intl. Kong. Fur Tech. Mech., Zurich,[English translation NACA Tech. Memo. 435, 62]. (1926)
2. N. Fukushima, N. Kasagi, (2001). The effect of walls on turbulent flow and temperature fields, in: Proceedings of Turbulent Heat Transfer III, Anchorage, USA.
3. M. Hirota, H. Fujita, H. Yokosawa, et al (1997). "Turbulent heat transfer in a square duct", Int. J. Heat and Fluid flow Vol. **18**, (1): pp. 170-180.
4. M. Piller, E. Nobile, (2002). "Direct numerical simulation of turbulent heat transfer in a square duct", Int. J. Numerical methods for heat and fluid flow, **12** (6): pp. 658-686.
5. M. Salinas-Vázquez and O. Métais (2002). "Large eddy simulation of the turbulent flow through a heated square duct", J. Fluid Mech. Vol. **453**,pp. 201-238.
6. M. Salinas-Vázquez, W. Vicente Rodriguez , R. Issa, (2005). "Effects of ridged walls on the heat transfer in a heated square duct," International Journal of Heat and Mass Transfer **48**, pp.2050-2063
7. J. Pallares and L. Davidson, (2002). "Large eddy simulations of turbulent heat transfer in stationary and rotating square ducts", Phys. Fluids **14**, 2804-2816.
8. Smagorinsky, J., (1963).General circulation experiments with the primitive equations. I. The basic experiment. Mon. Weather Rev. **91**, pp.499-164.
9. M. Breuer, W. Rodi, (1994). "Large eddy simulation of turbulent flow through a straight square duct and a 180 degree bend," Direct and Large-Eddy Simulation I, eds. P. R. Voke et al., Kluwer Academic Publishers, pp. 273-285.
10. Kim, J. and Moin, P., (1987) "Application of a fractional-step method to incompressible Navier-Stokes equations", J. Comput. Phys. **177**, pp.133-166.

Table 1
The flow conditions for simulated cases; Re_τ is defined by mean friction velocity averaged over four solid walls (t=top,b=bottom wall); W_w denotes the velocity of the moving wall and W_{Bulk} is the bulk velocity; $Re_c = \frac{W_w D}{\nu}$; $r = \frac{W_w}{W_{Bulk}}$.

	$Re_{\tau t}$	$Re_{\tau b}$	Re_τ	Re_{Bulk}	Re_c	r	$-\frac{D}{\rho W_w^2}\frac{\partial P}{\partial z}$
Case CP1	441	605	571	9972	4568	0.47	0.0621
Case CP2	305	591	538	10021	9136	0.94	0.0138
Case CP3	284	587	533	10029	11420	1.17	0.0083
Case CP4	364	581	553	10067	13704	1.36	0.0054

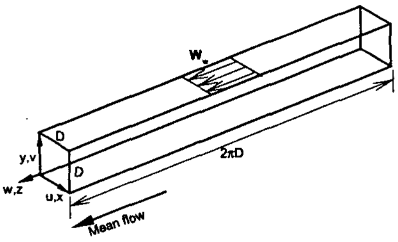

Figure 1. Schematic plot of the computational domain.

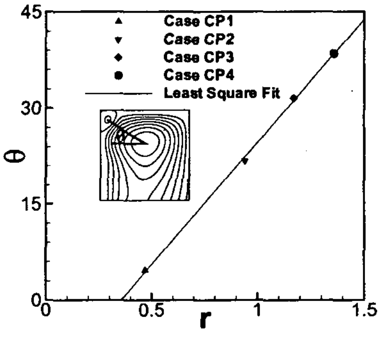

Figure 2. Angles between vortex cores near the top corners. Inset shows the streamline of case CP3 as a demonstration for the definition of Θ.

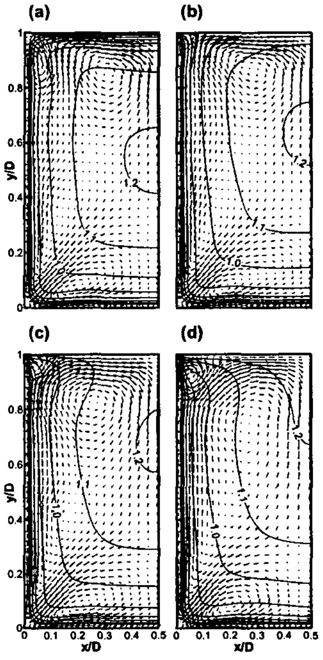

Figure 3. Secondary flow vector (every three vectors is drawn for clarity) overlapped by mean streamwise velocity contour (normalized by the bulk velocity). (a) Case CP1; (b) Case CP2; (c) Case CP3; (d) Case CP4.