Simulation of a 3D Turbulent Wavy Channel based on the High-order WENO Scheme

Bor-Jang Tsai 1, Chung-Chyi Chou 2†, Yeong-Pei Tsai 3, Ying Hung Chuang 4
Ph.D. Program in Engineering Science, Chung Hua, University 1, 3
Department of Fire and Safety, Da-Yeh University, 2†, 4
dustin@cloud.dyu.edu.tw, 2†

Abstract. Passive interest turbulent drag reduction, effective means to improve air vehicle fuel consumption costs. Most turbulent problems happening to the nature and engineering applications were exactly the turbulence problem frequently caused by one or more turbulent shear flows. This study was operated with incompressible 3-D channels with cyclic wavy boundary to explore the physical properties of turbulence flow. This research measures the distribution of average velocity, instant flowing field shapes, turbulence and pressure distribution, etc. Furthermore, the systematic computation and analysis for the 3-D flow field was also implemented. It was aimed to clearly understand the turbulence fields formed by wavy boundary of tube flow. The purpose of this research is to obtain systematic structural information about the turbulent flow field and features of the turbulence structure are discussed.

Keyword: Lower drag, Navier-Stokes, Wavy wall.

1. Introduction
Recent researches on wavy channel concentrate on the areas of revelation of fluxion structure and the control and reduction of anti-obstruction mechanism. The research was aimed to theorize this phenomenon and use it in the engineering applications to reduce obstruction. However the effects of fluid structure change and the wall surface pressure gradient on the change of wall surface obstruction in the near wall area are very complex. The reduction in obstruction from the wavy channel is apparently due to the improvement of the partial In the near-wall region of turbulence structure, which has reduced the friction in the fluids and between the fluid the solid wall. Therefore in order to further the revelation of wavy channel in obstruction reduction, it is believed that the near wall flowing should be characterized.

Experimental research of flow over the wavy wall was performed as early as 1932. The theory of the surface wall variation has been developed. Zilker and Hanratty[3], Zilker et al.[4] continued to conduct studies including the experiments of gauging wall surface pressure, measuring mean velocity of stream and the wall surface shearing stress and analyzing the mobile characteristics. In 1985 Patel[5] has improved simulation for several kinds of near wall with model (low Reynolds number). Patel and Chen[6] used double-decked model to solve turbulent complex stalled flow. Since this model saves the grids, therefore it can effectively save the computation space and time and enhances the computation feasibility. Patel et al.[7] calculation of the wavy wall turbulent flow was conducted on two kinds of wave height ratios in 1991. The simulated streamlines picture in the near wall vortex area and the cross section velocity distribution, the numerical simulation is consistent with experimental results.
Comparison of friction coefficient curves with pressure coefficients was also made between the wavy wall surface and flat surface. The effects of the profile on flows in the near wall area were analyzed. Ferrira and Lopes [8] carried on the multi-group wind tunnel experiments on the unimodular flows of sinusoidal wavy wall flow field. The low Reynolds number k-ε model with the control volumetric method was used to compute wavy wall near zone stalled flow for various wave height ratios. Montalbano and McCready [9] used the wave stability theory and added small perturbation quantity to Orr-Sommerfeld equation to develop the relation between the laminar wall surface pressure and the shear stresses.

Airiau and Giovannini [10] utilized statistical simulation to obtain the stream function and vortex charts at different time for the sinusoidal wavy wall. Also the average stream function and the average wall surface pressure coefficient curves were calculated using the time average method. With these tools vorticity and variation of the pressure gradient along the wall surface were analyzed. Malamataris1 and Bontozoglou [11] Dimensional Navier-Stokes equations within the entire range of laminar flow conditions. Relatively very low Reynolds numerical and experimental data. Focus of discussion in the results is due to the free surface contour, flow lines, velocity and pressure distribution along the free surface and the wall of the exhibit. Number of dimensionless flow interactions were studied for flow inversion condition is established, and the resonance phenomenon in the high Reynolds number is studied. Boersma [12]. Release particles in the flow field waveform observed flow trajectories of particles in the flow field. This will show on channel bottom small waves can generate large longitudinal vortex is similar to that observed at the free surface vortex flow Langmuir. The simulation results show that the particle has the highest concentration in the trough. Nakagawa et al. [13]. Turbulence using laser Doppler velocimetry (LDV) measurements were compared, for the turbulence over one flat surface, and with the small wavelength of the sine wave surface. Wavy border, the height of the flow separation roughened. Reynolds set to 46,000.

2. NUMERICAL METHOD

The numerical code in the current work is intended for the direct calculation of solving 3D Navier-Stokes equations with no subgrid scale turbulence model. Flow field contains multiple regions, with large gradients, we need to be carefully distinguished. Grid near the wall in the vertical direction requires encryption, to resolve the boundary layer. The separated shear layer, which is located the separation bubble, also need special attention. Because of these requirements, a fine resolution should be maintained until the middle of the channel, in order to resolve the separated shear layer normal. The present flow domain consists of three wavelengths compared with four in A. Nakayama and Maass & Schumann. For the present configuration of the flow, one of the coordinates is taken to conform to the wavy surface on the bottom and the flat upper boundary and the second coordinate is taken so that it is close to orthogonal to the first as close as possible without much distortion in the main flow domain. Approximate orthogonally near the solid boundary is important to capture the motion accurately in this region. Boundary conditions are set: no slip on the bottom of the wave-shaped walls, no slip boundary, and regularly flow direction and cross flow. In order to shorten the development time of initial flow, the current calculation first began with our previous DNS straight open channel flow as a result of the calculation of the initial flow of a cycle of big waves.

2.1. Grid setup

The (x, y, z) direction was discretized into 210125 elements. The number of grid points:125*125*41. The turbulent flow occurs mostly in the flow field nearby the wavy wall surface. In order to better reveal the turbulence movement, this study densifies the grid at both sides in the physical space Figure1.
2.2. Initial and boundary conditions

The inflow speed is at Mach number 0.6, and the Reynolds number is $3.66 \times 10^5$. The pressure is in the static atmospheric condition and the density is $1.2 \text{kg/m}^3$. Boundary value of the flow field is assumed to be the initial flow field exit condition and kept constant.

\[
\frac{\partial p}{\partial y} = 0, \quad \frac{\partial u}{\partial y} = 0, \quad v = 0, \quad \frac{\partial w}{\partial y} = 0
\]

The boundary condition is subsonic of sound and expressed as follows:

\[
\rho u = C_i, \quad \frac{y}{\gamma - 1} \frac{p}{\rho} + \frac{1}{2} (u^2 + v^2) = C_i
\]

3. RESULTS

Turbulent Flow Field in this study can be obtained from the sinusoidal wall having a lower wavy channel, however, mainly the production of turbulent shear layer is used to separate the back of the associated wave. Turbulent flow generation and maintenance guide whirlpool, found the flat wall seems to be unimportant wave wall of the channel. This is important, but little is known about many of flow structure, because it is a priority for the complex flow of applicability of the reference flow. Early numerical simulation and analysis of flow through the infinitely small amplitude wave surface wave think. The numerical simulation results and Cherukat [1] for comparison. Flow field contains multiple regions, with large gradients, we need to be carefully distinguished. Grid encryption requires the flow field in the vertical direction, in order to resolve the boundary layer. The separated shear layer, which is located the separation bubble, also need special attention. Because of these requirements, a fine resolution should be maintained until the middle of the channel, in order to resolve the separated shear layer normal.

Comparisons of streamlines, $\psi$, of the mean flow fields are presented in Figure 2. For computing grid test. (A) Mesh: 100 * 20 * 20; (B) Mesh: 125 * 41 * 41; (C) Grid: 251 * 41 * 41. B and C at the downstream of the vortex can correctly observed phenomena, and then to B Streamwise velocity distribution grid computing. There are obvious vortex phenomenon downstream of middle layer and the average speed is higher than the waveform wall.

Comparisons of Cherukat like, and Hudson (1993) present results are compared both flow position means both peaks and troughs velocity curve, showing good agreement in Figure 3. The flat boundary,
single wavy wall and bilateral wavy wall, three different instantaneous streamwise velocity chart are shown in Figure 4. At the flat boundary, both upper and lower parts of inner flow field were flat. It was impossible for bilateral flat wall to trigger vortex happening between tube walls and flowing bodies. Thus, the apparent separation phenomenon no happening nearby flowing fields. And Figure 5 through Figure 6 shows three different instantaneous cross-stream and spanwise velocity contour chart. At the single wavy wall, because the upper wall was flat, it was unavailable to trigger vortex happening between flowing bodies and tube wall. The separation phenomenon could be clearly seen. The lower wall part was a sinusoidal wavy one with apparent vortex happening to the wave trough. At the bilateral wavy wall, both upper and lower wall parts of inner flow field were sinusoidal wavy walls with the amplitude(A) rated at 2.54mm. Because both upper and lower wall parts were sinusoidal ones, there were vortex and pressure changes could be seen nearby the wave trough.

Figure 3 Comparison with measured profiles of mean streamwise velocities at locations for x=0.0, 0.2, 0.5, 0.6 and 0.8: Nakagawa(2003) LDV measurement; ●, Nakagawa(2003) DNS; —, Present mean streamwise; ---.

Figure 4 Three-dimensional instantaneous velocity chart for (A) Flat boundary; (B) Single wavy wall; (C) Bilateral wavy wall in the mean streamwise velocity.

Figure 5 Three-dimensional instantaneous velocity chart for (A) Flat boundary; (B) Single wavy wall; (C) Bilateral wavy wavy wall in the cross-stream direction.

Figure 6 Three-dimensional instantaneous velocity chart for (A) Flat boundary; (B) Single wavy wall; (C) Bilateral wavy wavy wall in the spanwise direction.
4. Conclusion
This study successfully simulates the flow field characteristics of the two-dimensional profile wall using the Upwind and LU-SSOR schemes for space-time discretization and a 3D grid of 251x41x41 with both sides densified, the numerical results agree well with the experimental results, and this agreement can be used to verify the numerical codes and numerical results. Due to the strong mixing of the core flow and turbulence intensities, Reynolds stresses, turbulent energy production, instantaneous and mean flow field, and the pressure distribution of the turbulent channel flow with a wavy wall are calculated and investigated. The purpose of this research is to obtain systematic structural information about the turbulent flow field. Features of the turbulence structure are discussed.

References
[1] S. Nakagawa, Y. Na, T.J. Hanratty., “Influence of a wavy boundary on turbulence I”, Highly rough surface Experiments in Fluids. 2003. 35. 422–436
[2] N. A. Malamataris1 and V. Bontozoglou. “Computer Aided Analysis of Viscous Film Flow along an Inclined Wavy Wall”, Journal of Computational Physics. 1999. 154, 372–392
[3] E. D. Montalbano and M. J. McCready. Laminar channel flow over long and moderate waves, J. Comp. Phys.1998. 6. 1-23.
[4] A. D. Ferreira etc., “Experimental and numerical simulation of flow around two-dimensional hills”, J. of Wind Eng. and Industrial Aerodynamics. 1995, (54/55), 173-181.
[5] Liu, et al., “Weighted Essentially Non-oscillatory Schemes”, Journal of Computational Physics. 1994. 115. 200-212.
[6] V. C. Patel, etc. “Turbulent flow in a channel with a wavy wall”, J. of Fluids Eng. 1991, 113, 579-586.
[7] V. C. Patel and H. C. Chen. “Near-wall turbulence models for complex flows including separation”. AIAA J. 1988. 26. 641-648.
[8] V. C. Patel, W. Rodi and G. Scheuerer. “Turbulence models for near-wall and low Reynolds number flows: a review”, AIAA J. 1985. 23. 1308-1319.
[9] P.Zilker, J. Hanratty. “Influence of the amplitude of a solid wavy wall on a turbulence flow Part 2, separated flows”, J. Fluid Mech., 1979. 90, 257-271.
[10] P.Zilker, W.Cook, and J. Hanratty. “Influence of the amplitude of a solid wavy wall on a turbulent flow. Part 1, Non-separated flows”. 1977. 82 , 29-51.
[11] A. Nakayama and K. Sakio “Simulation of flows over wavy rough boundaries”, Center for Turbulence Research Annual Research Briefs, 2002. pp.313-324.
[12] Bendiks Jan Boersma, “Particle distributions in the flow over a wavy wall”, Center for Turbulence Research Proceedeings of the Summer Program., 2000. 109.
[13] C. Airiau and A. Giovannini. “Vorticity evolution on a separated wavy wall flow”, Third International Workshop on Vortex Flows and Related Numerical Methods. 1999. 7. 1-11.
[14] P. Cherukat., “Direct Numerical Simulation of a Fully Developed Turbulent Flow over a Wavy Wall”, Theoretical and Computational Fluid Dynamics, 1998. 11, 109 – 134. Reports, Theses, and Individual Papers
[15] Marie Denison (2015) "A Brief Study of the Effect of Rigid Swept Surface Waves on Turbulent Drag" Advanced Modeling & Simulation (AMS) Seminar Series, August 27, 2015
[16] G. Lodato, etc. (2016) Direct numerical simulation of shock wavy-wall interaction: Analysis of cellular shock structures and flow patterns, J. Fluid Mech, 789: 221-258.
[17] B. Denet, etc. (2015) Model equation for the dynamics of wrinkled shock waves. Comparison with DNS and experiments. Combust. Sci. Tech., 187(1-2): 296-323.
[18] O. Zanotti and M. Dumbser, A high order special relativistic hydrodynamic and magnetohydrodynamic code with space-time adaptive mesh refinement, Computer Physics Communications, 188 (2015), 110-127.
[19] Y. Yang, M. Wan, Y. Shi, K. Yang and S. Chen, A hybrid scheme for compressible magnetohydrodynamic turbulence, Journal of Computational Physics, 306 (2016), 73- 91.