行政院國家科學委員會專題研究計畫 成果報告

直接數值模擬三度空間具波型邊界之流場特性分析
研究成果報告(精簡版)

計畫類別：個別型
計畫編號：NSC 97-2221-E-216-034-
執行期間：97年08月01日至98年07月31日
執行單位：中華大學機械與航太工程研究所

計畫主持人：蔡永培

計畫參與人員：博士班研究生-兼任助理人員：周中祺

處理方式：本計畫涉及專利或其他智慧財產權，2年後可公開查詢

中華民國 98年10月26日

國科會專題研究計畫成果報告撰寫格式

一、說明

國科會基於學術公開之立場，鼓勵一般專題研究計畫主持人發表其研究成果，但主持人對於研究成果之內容應負完全責任。計畫內容及研究成果如涉及專利或其他智慧財產權、違異現行醫藥衛生規範、影響公序良俗或政治社會安定等顧慮者，應事先通知國科會不宜將所繳交之成果報告蒐錄於學門成果報告彙編或公開查詢，以免造成無謂之困擾。另外，各學門在製作成果報告彙編時，將直接使用主持人提供的成果報告，因此主持人在繳交報告之前，應對內容詳細校對，以確定其正確性。

本格式說明僅為統一成果報告之格式，以供撰寫之參考，並非限制研究成果之呈現方式。精簡報告之篇幅（不含封面之頁數）以 4 至 10 頁為原則，完整報告之篇幅則不限制頁數。

成果報告繳交之期限及種類（精簡報告、完整報告或期中報告等），應依本會補助專題研究計畫作業要點及專題研究計畫經費核定清單之規定辦理。

二、內容格式：依序為封面、中英文摘要、目錄（精簡報告得省略）、報告內容、參考文獻、計畫成果自評、可供推廣之研發成果資料表、附錄。

(一)報告封面：請至本會網站（http：//www.nsc.gov.tw）下載製作（格式如附件一）。

(二)中、英文摘要及關鍵詞(keywords)。

(三)報告內容：請包括前言、研究目的、文獻探討、研究方法、結果與討論（含結論與建議）…等。若該計畫已有論文發表者，可以 A4 紙影印，作為成果報告內容或附錄，並請註明發表刊物名稱、卷期及出版日期。若有與執行本計畫相關之著作、專利、技術報告、或學生畢業論文等，請在參考文獻內註明之，俾可供進一步查考。

(四)頁碼編寫：請對摘要及目錄部分用羅馬字 I 、II、 III……標在每頁下方中央；報告內容至附錄部分請以阿拉伯數字 1.2.3.……順序標在每頁下方中央。

(五)附表及附圖可列在文中或參考文獻之後，各表、圖請說明內容。

(六)計畫成果自評部份，請就研究內容與原計畫相符程度、達成預期目標情況、研究成果之學術或應用價值、是否適合在學術期刊發表或申請專利、主要發現或其他有關價值等，作一綜合評估。

(七)可供推廣之研發成果資料表：凡研究性質屬***應用研究***及***技術發展***之計畫，請依本會提供之表格（如附件二），每項研發成果填寫一份。

三、計畫中獲補助國外或大陸地區差旅費、出席國際學術會議差旅費或國際合作研究計畫差旅費者，須依規定撰寫心得報告，以附件方式併同成果報告繳交，並請於成果報告封面註記。

四、打字編印注意事項

1. 用紙

使用 A4 紙，即長 29.7 公分，寬 21 公分。

2. 格式

中文打字規格為每行繕打（行間不另留間距），英文打字規格為 Single Space。

3. 字體

報告之正文以中英文撰寫均可。在字體之使用方面，英文使用 Times New Roman Font，中文使用標楷體，字體大小請以 12 號為主。

附件一

行政院國家科學委員會補助專題研究計畫 ■成果報告 □期中進度報告

（計畫名稱）

計畫類別：■ 個別型計畫 □ 整合型計畫
計畫編號：NSC－97－2221－E－216－034
執行期間： 97 年 08 月 01 日至 98 年 07 月 31 日

計畫主持人：蔡永培
共同主持人：
計畫參與人員：
兼任研究助理 周中祺

成果報告類型(依經費核定清單規定繳交)：■精簡報告 □完整報告

本成果報告包括以下應繳交之附件：
□赴國外出差或研習心得報告一份
□赴大陸地區出差或研習心得報告一份
□出席國際學術會議心得報告及發表之論文各一份
□國際合作研究計畫國外研究報告書一份

處理方式：除產學合作研究計畫、提升產業技術及人才培育研究計畫、列管計畫及下列情形者外，得立即公開查詢
□涉及專利或其他智慧財產權，□一年■二年後可公開查詢

執行單位：中華大學機械系

中 華 民 國 98 年 10 月 26 日

附件二

可供推廣之研發成果資料表

□ 可申請專利　　□ 可技術移轉　　　　　　　　日期：__年__月__日

國科會補助計畫	計畫名稱： 計畫主持人： 計畫編號：　　　　學門領域：
技術/創作名稱	
發明人/創作人	
技術說明	中文： （100~500 字）
	英文：
可利用之產業 及 可開發之產品	
技術特點	
推廣及運用的價值	

※　1. 每項研發成果請填寫一式二份，一份隨成果報告送繳本會，一份送　貴單位研發成果推廣單位（如技術移轉中心）。

※　**2. 本項研發成果若尚未申請專利，請勿揭露可申請專利之主要內容。**

※　3. 本表若不敷使用，請自行影印使用。

Direct Numerical Simulation of 3D Fully Developed Turbulent Flow over a Wavy Wall

NSC Project No. NSC 97-2221-E-216-034

Abstract

This paper of this investigation is to develop a 3D numerical code and compute flow field properties generated by turbulent channel flow over a wavy wall. The flows over a wavy wall are very important phenomena in nature. They play quite important roles in engineering and scientific applications. Since wavy wall can increase boundary's surface and speed up radiation, therefore wavy wall flows can be applied in the area of heat conduction. Moreover a concave-convex thin film coating was developed and applied on the external surface of an Airbus A320 test airplane. The experiment results indicate that a surface coverage of 70% could achieve a fuel savings of 1-2% . In the direct simulation methods of flows over a wavy wall, Weighted Essentially Non-Oscillatory by the compressible flow domain was selected to solve Navier-Stokes equation. And the LU-SSOR method was applied to increase numerical stability and converging of the formula. A good agreement was found between simulation analysis results based on the Upwind method and the experimental values generated by P. Cherukat, Y. Na, and T. J. Hanratty & J. B. Mc Laughlin(1998). The simulation analysis has accurately predicted the fluid separation and superposition of the return flow in the wavy wall downhill areas. As the amplitude doubles, it was predicted that the detachment point of the fluid can occur earlier near the profile wall and the region of return flow can increase.

Keywords: Lower drag, Economy energy, 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 turbulent flow structure in the near wall areas, 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[4], Zilker et al.[5] 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[6] has improved simulation for several kinds of near wall with low Reynolds number $k-\varepsilon$ model. Patel and Chen[7] have 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.[8] 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 were in a good agreement with the 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 [9] 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 (A/λ). Montalbano and McCready [10] 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 [11] 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[12] the dimensionless Navier-Stokes equations are solved in the whole range of the laminar flow regime. Numerical predictions are compared with available experimental data for very low Reynolds numbers. The emphasis in the discussion of results is given in the presentation of free surface profiles, streamlines, velocity, and pressure distributions along the free surface and the wall. The interaction of the dimensionless numbers of the flow is studied, criteria for flow reversal are established, and a resonance phenomenon at high Reynolds numbers is investigated. Boersma[13]. The evolution in space and time of particles are released in this flow will be examined. It will be shown that small waves on the channel bottom can generate large longitudinal vortices similar to Langmuir vortices that are observed in flows with waves at the free-surface. The simulation results show that the concentration of the particles is maximal on the downstream side of the wave crest. Nakagawa et al.[14], Measurements of turbulence with laser Doppler velocimetry (LDV) are compared for turbulent flow over a flat surface and a surface with sinusoidal waves of small wavelength. The wavy boundary was highly rough in that the flow separated. The Reynolds number based on the half-height of the channel and the bulk velocity was 46,000.

The wavelength was 5 mm and the height to wavelength ratio was 0.1. The root-mean-squares of the velocity fluctuations are approximately equal if normalized with the friction velocity. This can be explained as a consequence of the approximate equality of the correlation coefficients of the Reynolds shear stress. Calculations with a direct numerical simulation (DNS) are used to show that the fluid interacts with the wall in quite different ways for flat and wavy surfaces. They show similarity in that large quadrant 2 events in the outer flow, for both cases, are associated with plumes that emerge from the wall region and extend over large distances. Measurements of skewness of the streamwise and wall-normal velocity fluctuations and quadrant analyses of the Reynolds shear stresses are qualitatively similar for flat and wavy surfaces. However, the skewness magnitudes and the ratio of the quadrant 2 to quadrant 4 contributions are larger for the wavy surface. Thus, there is evidence that turbulent structures are universal in the outer flow and for quantitative differences in the statistics that reflect differences in the way in which the fluid interacts with the wall. Zilker et al.[5] measurements of the shear-stress variation along and the velocity profiles above a solid wavy wall bounding a turbulent flow are presented for waves with height-to-length ratios of $2a/\lambda=0.0312$ and 0.05. These are compared with previous measurements of the wall shear stress reported by Thorsness (1975) and by Morrisroe (1970) for $2a/\lambda = 0.012$. The investigation covered a range of conditions from those for which a linear behaviour is observed to those for which a separated flow is just being initiated.

In the direct simulation methods of flows over a wavy wall, Weighted Essentially Non-Oscillatory by the compressible flow domain was selected to solve Navier-Stokes equation. And the LU-SSOR method was applied to increase numerical stability and converging of the formula. A good agreement was found between simulation analysis results based on the WENO method and the experimental values generated by Tsai[1] “Direct Numerical Simulation of a Fully Developed Turbulent Flow over a Wavy Wall” and Kim[2] “Numerical investigation of confined supersonic mixing flow” and Cherukat et al.[3] The simulation analysis has accurately predicted the fluid separation and superposition of the return flow in the wavy wall downhill areas. As the amplitude doubles, it was predicted that the detachment point of the fluid can occur earlier near the profile wall and the region of return flow can increase. This research then carries on the velocity field take this as the datum in view of the wavy wall phenomenon of flow, geometry outlook. The wavy wall applies major part three carries on the system discussion.

2. Numerical Method

2.1 Flow configuration

The parameters of the simulation are the same as used by Tsai[1] and Cherukat et al.[3]. In their numerical, the amplitude and the wavelength of the wavy wall were 2.54 mm and 50.8 mm, respectively. The distance between the mean location of the wavy surface and the flat wall was 50.8 mm. A schematic diagram of the three-dimensional computational domain is shown in Fig.1 and Fig.2. It consists of a channel which is unbounded in both the streamwise (x) and spanwise (z) directions. The lower wall has $N_\omega (= 4)$ waves with sinusoidal shape and a mean position at the y = 0 plane (y is the vertical direction). The flat wall is located at y = h. The location of the wavy wall, y_ω, is given by $y_\omega = a\cos(\frac{2\pi x}{\lambda})$ where a is the amplitude of the wave and is the wavelength. The mean flow in the streamwise direction is pressure driven. In the present study, wavelength ¸ and amplitude a were set equal to h and 0:05h to match the parameters of Hudson's (1993) measurements. The flow is assumed to be homogeneous in the spanwise direction, justifying the use of periodic boundary conditions. The flow is also assumed to be periodic in the streamwise direction. Thus, the computational box size in the streamwise $(\Lambda x = N_\omega \lambda)$ and spanwise directions (Λx) should be large enough to include the largest length scale of the turbulent structures. The extents of the computational domain were, respectively, chosen to be 4h in the streamwise direction and 2h in the spanwise direction.

2.2 Governing equation

3D Navier-Stokes differential equation：

$$\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = \frac{\partial Ev}{\partial x} + \frac{\partial Fv}{\partial y} + \frac{\partial Gv}{\partial z} \quad (2\text{-}1)$$

where Q is a conservation variable in equation：

$$Q = \begin{bmatrix} p \\ u \\ v \\ w \end{bmatrix} \quad (2\text{-}2a)$$

where ρ is density; u and v are quantities of velocity; e is unit volume energy in the above equation. E and F are Non-stickiness fluxes：

$$E = \begin{bmatrix} u\beta \\ u^2 + p \\ uv \\ uw \end{bmatrix} \quad (2\text{-}2b)$$

$$F = \begin{bmatrix} v\beta \\ uv \\ v^2 + p \\ vw \end{bmatrix} \quad (2\text{-}2c)$$

$$G = \begin{bmatrix} w\beta \\ uw \\ vw \\ w^2 + p \end{bmatrix} \qquad (2\text{-}2d)$$

E_v and F_v are stickiness fluxes defined as:

$$Ev = \begin{bmatrix} 0 \\ 2\nu u_x \\ \nu\left(u_y + v_x\right) \\ \nu\left(u_z + w_x\right) \end{bmatrix} \qquad (2\text{-}3a)$$

$$Fv = \begin{bmatrix} 0 \\ \nu\left(v_x + u_y\right) \\ 2\nu v_y \\ \nu\left(v_z + w_y\right) \end{bmatrix} \qquad (2\text{-}3b)$$

$$Gv = \begin{bmatrix} 0 \\ \nu\left(w_x + u_z\right) \\ \nu\left(w_y + v_z\right) \\ 2\nu w_z \end{bmatrix} \qquad (2\text{-}3b)$$

3. RESULTS

A number of studies of flow over a period train of solid waves have been carries out based on both laboratory experiment and numerical simulation. A principal finding of these work is that the flow is characterized by an outer flow and by an inner flow extending a distance from the mean level of the surface. The mechanism of turbulence production in the inner region is fundamentally different from flow over a flat surface. Turbulence production in the inner region for turbulent flow over a flat plat is related to the formation of streamwise vortices in the wall region. In the turbulent channel flow with lower sinusoidal wall, however, turbulence production is mainly associated with a shear layer that separates from the back of the wave. Turbulence generation and sustainment by flow oriented vortices such as found for flat walls appears to be unimportant in a channel with wavy wall. This important, but poorly understood flow configuration is a focus of numerous studies because of its applicability as a reference flow for complex flows. Early numerical and analytical studies of flow over wavy surface considered waves of infinitesimal amplitude. A good agreement was found between our 2D simulation results based on the WENO method and the experimental data measured by Vyas, Saurabh (2005); Nakagawa(2003); P. Cherukat, et al.(1998) and Hudson’s (1993).

The parameters of the simulation are the same as used by Cherukat et al.[3] for comparison purposes. In their numerical, the amplitude and the wavelength of the wavy wall were 2.54 mm and 50.8 mm, respectively. The distance between the mean location of the wavy surface and the flat wall was 50.8 mm. A schematic diagram of the two-dimensional physics domain is shown in Fig. 1. The Computational domain shows the Figure2. It consists of a channel which is unbounded in both the streamwise (x) and spanwise (y) directions. And flat wall is located at y = h. The location of the wavy wall, y_ω, is given by $y_\omega = a\cos(\frac{2\pi x}{\lambda})$ where a is the amplitude of the wave and is the wavelength. The mean flow in the streamwise direction is pressure driven. In the present study, wavelength, and amplitude a were set equal to h and 0.05h to match the parameters of Hudson’s (1993) measurements. The flow is assumed to be homogeneous in the spanwise direction, justifying the use of periodic boundary conditions. The flow is also assumed to be periodic in the streamwise direction. Thus, the computational box size in the streamwise $(\Lambda x = N_{\omega}\lambda)$ and spanwise directions (Λx) should be large enough to include the largest length scale of the turbulent structures. The extents of the computational domain were, respectively, chosen to be 4h in the streamwise direction and 2h in the spanwise direction.

The flat boundary, single wavy wall and bilateral wavy wall, three different instantaneous streamwise velocity chart are shown in Fig.3. 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 Fig.4 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.

The Fig.5 and Fig.6 shows three different streamwise velocity contour in the X-Y and Y-Z planes. The by way of velocity contour our to be clear about in the flow field the celocity change different has a difference along with the boundary, especially of bilateral wavy wall.

Mean velocity profiles at two streamwise locations (the crest and trough), in Fig.7 show good agreement. Since streamlines can be obtained by an integration of mean streamwise velocity, the good agreement of the mean velocity profile shown in Figure 6 is consistent with the observation that the thickness of the recirculating region or separation bubble is in fair agreement with measurements.

The mean pressure distribution in the full flow over a wavy channel is shown in Figure 8. The mean pressure is a combination of the linear pressure drop and a wave-induced variation. Pressures shown in Figure 8 were obtained by subtracting the contribution from the mean pressure gradient; the pressure at the crest is taken as a reference. Large vertical pressure gradients exist in the vicinity of the separation point. The spatial pattern of mean pressure is periodic in the streamwise direction, indicating that the size of the streamwise domain is adequate.

5. Conclusions

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 grid of 125x41 x41 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 shows the turbulent flow over a wavy wall have a Lower drag and Economy energy phenomenon.

This research is to obtain systematic structural information about the turbulent flow field. Features of the turbulence structure are discussed.

6. Acknowledgements

This work is funded by National Science Council under grant NSC-96-2221-E-212-021.

7. References

1. Yeong-Pei Tsai and Chung-Chyi Chou 2007. “Direct Numerical Simulation of a Fully Developed Turbulent Flow

over a Wavy Wall” Chung Hua Journal of Science and Engineering, Vol.5, No.2, pp.9-14.

2. S. K. Kim 2001. “An Experimental Study of Flow in a Wavy Channel by PIV” The 6TH Aslan Symposium on Visualization.
3. P. Cherukat, Y. Na, T.J. Hanratty, and J.B. McLaughlin. 1998. “Direct Numerical Simulation of a Fully Developed Turbulent Flow over a Wavy Wall” Theoretical and Computational Fluid Dynamics 11,109–134.
4. P.Zilker, J.Hanratty. 1979. “Influence of the amplitude of a solid wavy wall on a turbulence flow. Part 2” separated flows, J. Fluid Mech 90, 257-271.
5. P.Zilker, W.Cook, and J. Hanratty. 1977. “Influence of the amplitude of a solid wavy wall on a turbulent flow. Part 1” Non-separated flows 82 , 29-51.
6. V. C. Patel, W. Rodi and G. Scheuerer. 1985. “Turbulence models for near-wall and low Reynolds number flows: a review” AIAA J. 23. 1308-1319.
7. V. C. Patel and H. C. Chen. 1988. “Near-wall turbulence models for complex flows including separation” AIAA J. 26. 641-648.
8. V. C. Patel, J. T. Chon and J. Y. Yoon. 1991 “Turbulent flow in a channel with a wavy wall”, J. of Fluids Eng. 113, 579-586.
9. A. D. Ferreira and A. M. G. Lopes etc. 1995. “Experimental and numerical simulation of flow around two-dimensional hills”, J. of Wind Eng. and Industrial Aerodynamics. (54/55), 173-181.
10. E. D. Montalbano and M. J. McCready. 1998. “Laminar channel flow over long and moderate waves”, J. Comp. Phys. 6. 1-23.
11. C. Airiau and A. Giovannini. 1999. Vorticity evolution on a separated wavy wall flow. Third International Workshop on Vortex Flows and Related Numerical Methods. 7. 1-11.
12. Daniel P, Zilker, Gerald W. Cook and Thomas J. Hanrattya1 2006. “Influence of the amplitude of a solid wavy wall on a turbulent flow. Part 1. Non-separated flows” Cambridge University Press 12Apr.
13. N. A. Malamataris1 and V. Bontozoglou. 1999 “Computer Aided Analysis of Viscous Film Flow along an Inclined Wavy Wall” Journal of Computational Physics 154, 372–392
14. Bendiks Jan Boersma 2000. “Particle distributions in the flow over a wavy wall” Center for Turbulence Research Proceedings of the Summer Program 109.

8. Figures

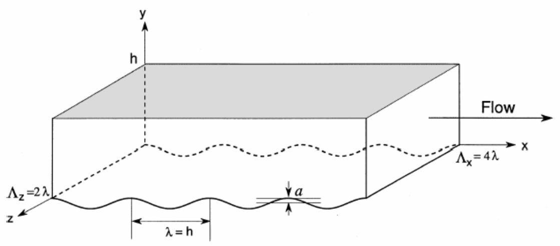

Fig.1 Flow over a wavy wall of a computational domain (Cherukat et al.[3]).

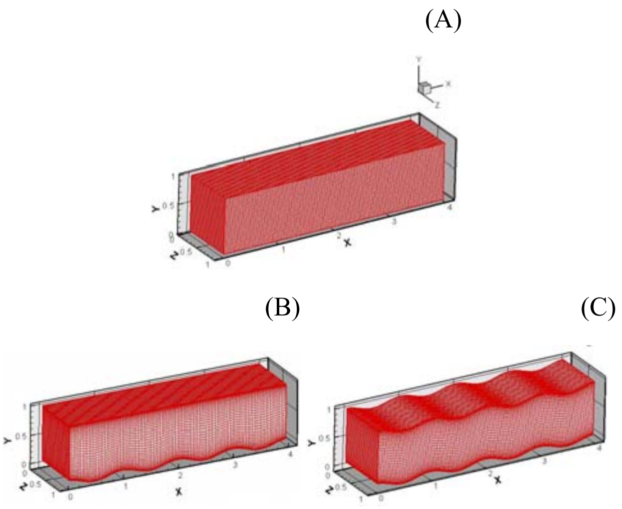

Fig.2 The computational grid of (A)Flat wall, (B)Single wavy wall and (C)bilateral wavy wall.

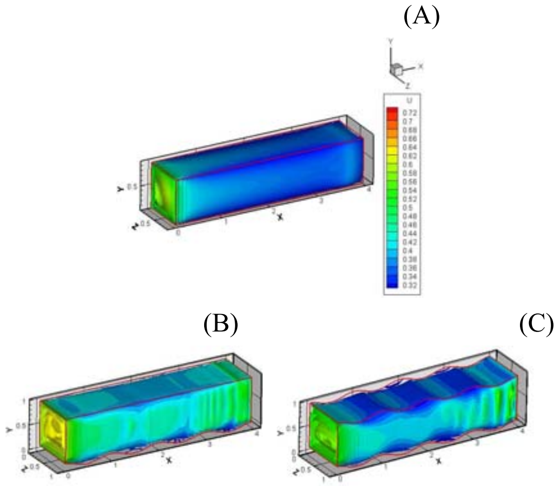

Fig.3 Three-dimensional instantaneous velocity chart for (A)Flat wall, (B)Single wavy wall and (C)bilateral wavy wall in the mean streamwise and normal velocity.

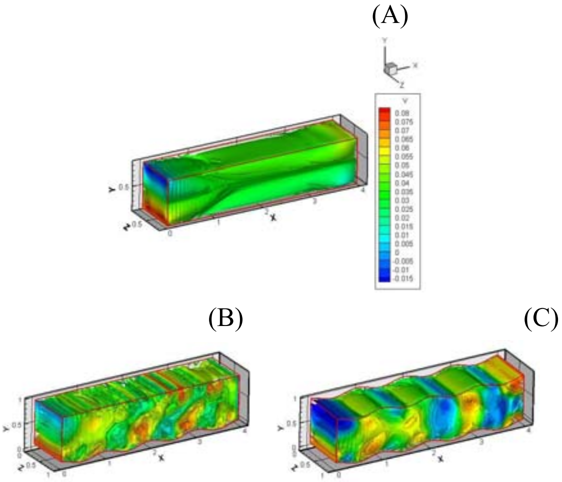

Fig.4 Three-dimensional instantaneous velocity chart for (A)Flat wall, (B)Single wavy wall and (C)bilateral wavy wall in the cross-stream direction.

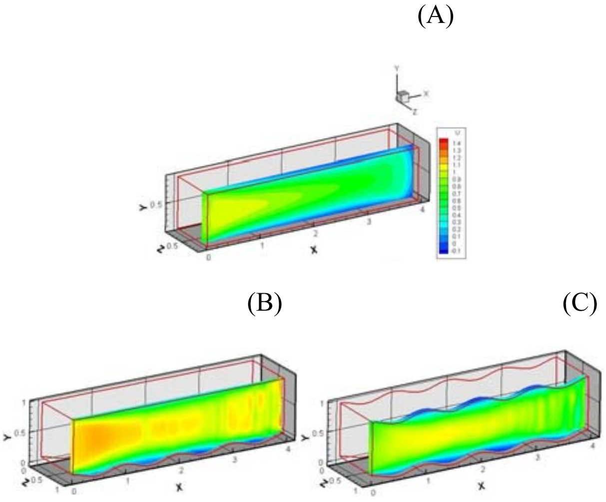

Fig.5 Distributions of instantaneous streamwise velocity contours for flat boundary in the X-Y planes, in (A)Flat wall, (B)Single wavy wall and (C)bilateral wavy wall.

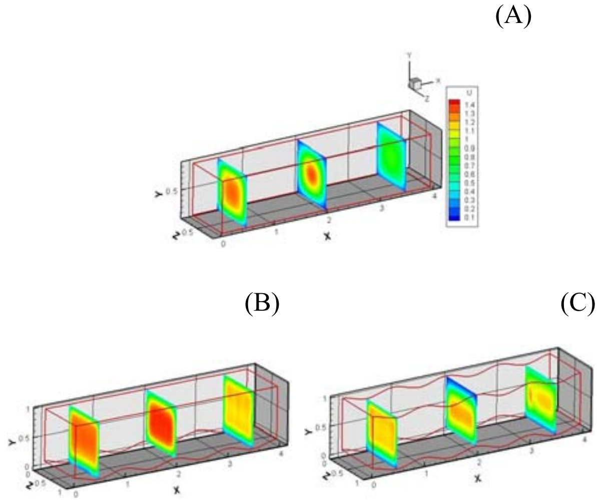

Fig.6 Distributions of instantaneous streamwise velocity contours for (A)Flat wall, (B)Single wavy wall and (C)bilateral wavy wall. In the Y-Z plane, A=2.54mm.

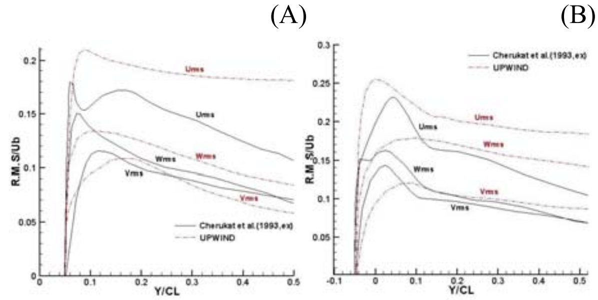

Fig.7 Comparison with Cherukat et al.: —, and present: --, (A) x/h=0.0 (crest); (B) x/h=0.5 (trough).

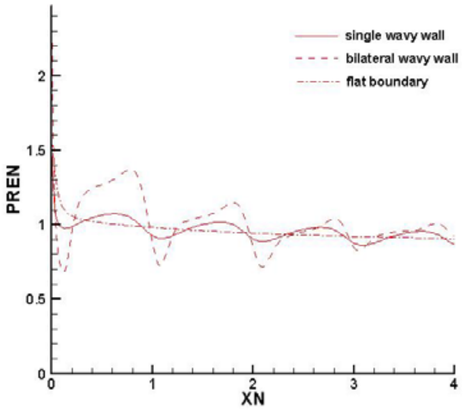

Fig.8 Pressure curve from the studies for Reynolds number is 3460. Comparison with flat boundary: •－•,single wavy wall:─, bilateral wavy wall:•••.

直接數值模擬三度空間具波型邊界之流場特性分析

國科會計畫編號：NSC 97-2221-E-216-034

摘要

本研究之目的是要發展出一個程式，可以用來計算由超音速流體所形成之三維管道具週期性波形邊界其紊流流場之物理特性。波形壁流場是自然界中相當重要的一種流動現象，它在工程與科學領域上有相當重要的應用與重要的地位。亦可應用於熱傳導方面，波形壁有助於表面積的增加，可利加快散熱速度。再者亦可發展溝槽薄膜，法國空中巴士A320試驗機將其貼覆於飛機70%表面達到節省1~2%之燃料之功效。因此針對該特性進行數值模擬研究，在數值方法的選擇上，以加權型基本不震盪算則(Weighted Essentially Non-Oscillatory)直接求解奈維爾-史托克(Navier-Stokes)方程式。而時間離散上採用LU-SSOR method 以增強數值之穩定性及加速程式之收斂，二維研究中已與P. Cherukat, Y.Na, 和 T.J. Hanratty & J.B.Mc Laughlin(1998)實驗值進行比較。波形管壁具有提升曳力及減低阻力已達節能之效益，在量測平均速度分佈，瞬間流場圖形，紊流強度及壓力分佈等，本研究將更近一步針對三維留場做一有系統之計算與分析。此外雷諾數及波形管壁之振幅對紊流場結構及紊流強度之影響亦將討論。以期對管流具波形邊界形成之紊流場有清楚的了解。

*關鍵字：*減阻；節能；波形壁