Direct Numerical Simulation of a Turbulent Flow over a Wavy Wall With Lower Drag and Economy Energy Analysis

蔡永培,周中祺
Mechanical Engineering
Engineering
yptsai@chu.edu.tw

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.

Keyword：Lower drag, Economy energy, Wavy wall.