Proceedings of the Eighth Asian Congress of Fluid Mechanics
December 6-10, 1999, Shenzhen, China

MULTI-SCALE COMPLEX FLOW FIELDS AND NUMERICAL SIMULATION OF TURBULENT FLOW

Zhuang Fenggan

(China Aerospace Corporation, Beijing 100830, China)

Multi-scale complex flows, such as turbulence, exist almost everywhere in the nature. Some achievements have been reached recently in the study of this kind of flows. With developments of computational technique the numerical simulations become one of the important tools for investigating the complex flow fields.

The complex flow fields, like turbulence, are characterized by the following properties: they are unstable and unsteady, they have multi-scale characters, and they are three dimensional. The nonlinear effect in the complex flow fields is a predominent featare. For correct simulation of such kind of problems the key point is capability of methods for capturing the multi-scale structures. We have successfully simulated the steady state flows with macroscopic spatial structures, like shocks, by using second-order accurate schemes. But for the unsteady, unstable field flows and flow with unsteady bifurcation, like turbulent flows, the multi-scale structures are changing in advancement of time, and usually the small structures are random and chaotic. The nonlinear effect is very complicated. All these physical phenomena should be captured well by numerical methods. Special attention should be paid on correct simulation of phase relation in the physical problems. Numerical dissipation may smooth out the small flow field structures, and the numerical dispersion may lead to wrong structures.

The flow fields with a range of scales are very complicated. In the paper we pay more attention on direct numerical simulation. There are three methods for improving the resolution. One of them is to refine the grid system, but the resource of computers is limited. The another is to construct a reasonable grid system with grid generation technique. With this method usually is difficult to use high order accurate scheme. In this paper we will discuss the third method, the method with high order accuracy.

The behavior of numerical solutions is analyzed. From Fourier analysis for the one dimensional model equation it can be seen that the Fourier components with lower wave numbers can be simulated very well even with the second order accurate scheme. In practical applications we have successfully solved large scale structure, like shock, with lower accurate TVD schemes. With Fourier analysis it also can be seen that with lower accurate schemes the high frequency components may smooth out or even have wrong traveling direction. This is not important if we are interested only in steady and stable large structures of flow fields. But it is important for solving the multi-scale problems. In the numerical solutions with low accurate schemes the high frequency components may go ahead, behind, or in opposite direction according to the corresponding physical waves. The behaviors of numerical solutions are also analyzed for the linearized two-dimensional Euless equations.

For multi-dimensional problems the influence of dispersion and dissipation errors exhibits anisotropic characters. The Fourier analysis shows that the resolution of numerical solution is getting better with increasing the order of accuracy of approximations. Three methods for improving the resolutions of solutions for solving the multi-scale problems. They are the method of increasing the order of accuracy of approximation, the method of accuracy balance between different Fourier components, and the method of group velocity control. The high order accurate schemes are used to solve physical problems.