Present State and Future of CFD Based on Three-Dimensional RANS analysis

Kwang-Yong KIM
Inha University, Mechanical Engineering Dept., 253 Yonghyun-Dong Nam-Gu Incheon 402-751, kykim@inha.ac.kr

Abstract

Computational Fluid Dynamics (CFD) based on Navier-Stokes equations has been developed rapidly for several decades with the developments of high speed computers and numerical algorithms, and presently is regarded as an essential analysis tool in the engineering applications containing fluid flow and convective heat transfer. It is known that for turbulent flow the Navier-Stokes equations can be calculated precisely by Direct Numerical Simulation (DNS). However, DNS needs huge computing time even for simple low-Reynolds number flows, and thus is not practical. Large Eddy Simulation (LES) can be an alternative. But, LES also needs considerable computing time for the analysis of engineering flows, and have some problem in the methods. Therefore, the analysis methods using Reynolds-averaged Navier-stokes equations (RANS) and turbulence closure models are still regarded as the major techniques for the analysis of turbulent flows in spite of the inaccuracy of the prediction. In this presentation, the present state and the prospect of CFD based on three-dimensional RANS analysis are introduced for physical models and numerical algorithms with the engineering examples. Especially, for the analysis of two-phase flows in nuclear reactor, the recently developed techniques are also introduced. And, the presentation includes the methods of design optimization using RANS analysis and numerical optimization techniques with variety of the applications.