



THE HONG KONG UNIVERSITY OF SCIENCE & TECHNOLOGY

Department of Mathematics

SEMINAR ON SCIENTIFIC COMPUTATION

**A robust and efficient fluid-solid thermal coupling solver based
on second-order Finite Volume method**

by

Prof. Xiaoquan YANG

Shanghai University

Abstract

Cell-centered finite volume (FV) method has been widely used in computational fluid dynamics (CFD). Compared with finite difference (FD) method, it is more suitable for simulations of complex configurations using unstructured grids, but is hardly to extend to high-order accuracy calculations (by which we mean third-order accuracy and above). Considering the computational cost and the computational accuracy we need for our simulations, most of the numerical simulations for engineering applications are based on second-order accuracy FV method for now.

However, there are still some challenges in solving the compressible turbulent flows with second-order FV method, such as the robustness and efficiency of the reconstruction methods and implicit schemes. In order to overcome these defects, a robust and efficient fluid-solid thermal coupling solver based on second-order FV method is developed by Aerodynamics Research Center of Modern Aircraft (ARCMA). In this solver, a compact Least-Squares reconstruction method is used to calculate the gradients for the distribution of flow field variables approximation, which shows better robustness and accuracy than traditional reconstruction methods. Meanwhile, an exact Jacobian solving strategy is developed for implicit temporal discretization. The accurate processing of Jacobian matrix can extensively improve the invertibility of the Jacobian matrix and avoid introducing extra numerical errors. In addition, by coupling the heat transform of solid, the fluid-solid thermal coupling calculation can be realized. Besides, sliding mesh technology is applied to deal with the calculations of rotational motion. Numerical results of several benchmarks indicate that the solver performs well in robustness, efficiency, accuracy, and have good application potential in turbulent flow simulations of complex configurations.

Date : 17 April 2023 (Monday)

Time : 11:00am – 12:00nn

Venue : Room 5510 (Lifts 25/26)

All are Welcome!