A NUMERICAL STUDY OF CAVITATING FLOWS BASED ON PLIC-VOF METHOD AND ARBITRARY UNSTRUCTURED MESHES

By

Dezhi Dai

Thesis Advisor: Dr. Albert Y. Tong

1:00 P.M. – 3:00 P.M., Monday, July 29

WH, Room 200

Abstract

Piecewise Linear Interface Calculation (PLIC) schemes have been extensively employed in the Volume of Fluid (VOF) method for interface capturing in numerical simulations of multiphase flows. Polyhedral unstructured meshes are often adopted due to their geometric flexibility and superiority in gradient calculation. Four analytical interface reconstruction algorithms in the PLIC-VOF method for arbitrary convex polyhedral cells have been proposed in the present study. The plane interface at a given orientation within a polyhedral cell is located by four different analytical techniques. They have been tested successfully for six different geometric shapes that are common in polyhedral meshes. The computational efficiencies of the algorithms have been compared with two other published schemes in the literature. The proposed algorithms have been shown to yield smaller truncation errors with reduction in computational complexity. A numerical simulation of a 3D dam-breaking problem has been successfully performed using the proposed interface reconstruction scheme on a polyhedral mesh. The percentage of the overall computational time consumed has been assessed to justify its optimization in a real multiphase flow simulation.

The effect of cavitation on the dynamics of a two-dimensional hydrofoil and three-dimensional propeller have been studied numerically. Analytical PLIC-VOF methods on overset unstructured meshes has been employed to capture the interface between liquid and vapor phases in cavitating flows around a NACA 66 (MOD) hydrofoil and PPTC propeller. A modified version of the
Schnerr-Sauer cavitation model has been adopted to compute the local mass transport rate for the vaporization and condensation processes. Turbulent eddy viscosity is evaluated by using the Spalart-Allmaras (SA) one-equation model. Simulations are performed under the framework of OpenFOAM with a modified flow solver. Stable sheet and unstable cloud cavitations have been investigated. The findings in the present study are consistent with experimental studies reported in the literature.