Evaluation of the cavitatingFoam solver for low Mach number flows around 2D hydrofoils

Dr. Anne Gosset, anne.gosset@udc.es **Dr. Vicente Diaz Cásas**, vicente.diaz.casas@udc.es **Dr. Fernando López Peña**, flop@udc.es	Universidade da Coruña Campus de Esteiro, Mendizábal s/n 15403 Ferrol, Spain.

Abstract

Cavitation on hydrofoils can take a wide variety of forms, ranging from bubble and patch cavitation to sheet cavitation. In the latter, the front part of the cavity remains attached, while the rear part is more or less unsteady depending on the cavitation number σ. For the lowest values of σ, the cavity tail sheds clouds of bubbles ("cloud cavitation"), inducing the formation of a reentrant jet. In all cases, vapor cavities on hydrofoils are generally large and feature important dynamics, which explains that all approaches for cavitation modeling are not necessarily adapted.

The different approaches for cavitation fall into two main categories: two fluids methods, in which the liquid and the vapor phases are solved separately in combination with mass transfer models for vaporization and condensation, and single fluid methods, in which the liquid-vapor mixture is considered as a homogenous fluid with variable density. In the latter, differences between the various existing models lie in the relation between the pressure and density fields: this coupling is generally treated through a barotropic equation of state, or through a transport equation for the vapor mass fraction. The single fluid approach has received more attention up to now, because of its lower computational cost and its easier coupling with turbulence.

The cavitatingFoam solver implemented in OpenFoam uses such a homogeneous equilibrium approach, in which a barotropic equation of state is injected in the continuity equation to produce a pressure equation. The main difficulty arises from the strong coupling between pressure and the equations of state and continuity, and the rapid changes in compressibility between the two-phase mixture and the liquid. The solver has originally been designed for high-speed flows [1], in such a way that the time step limiter is the non-acoustic Courant number. However, its numerical implementation in OpenFoam has been generalized to allow treating lower speed flows. The severe time step restriction due to the acoustic Courant number is reduced by treating acoustic terms implicitly. We propose here to evaluate the capabilities of cavitatingFoam for low Mach number flows, such as flows past hydrofoils.

For that purpose, a test case is chosen, characterized by a rather stable sheet cavitation regime on a NACA 0015 foil [2] (Re=6.5.105, σ=1.3, AoA=7°). The predictions of two phase simulations using interPhaseChangeFoam with Kunz´s mass transfer model and implicit LES are compared to the predictions of cavitatingFoam. Provided that the computation is started from a well converged non cavitating flow field and that the time step is small enough, cavitatingFoam yields a qualitatively correct pattern of cavitation, as evidenced in figure 1. However, the dynamics of the cavity is strongly affected by the use of a RANS model for turbulence, and as a result, the average lift coefficient is underestimated of almost 20%.

In this contribution, the difference in behavior of the vapor cavity (mean length and dynamics) with the single and two-phase approaches will be pointed out, as well as its impact on the average lift and drag coefficients. Emphasis will be given to the compromise that can be reached between the quality of results and the CPU requirements for both cavitatingFoam and InterPhaseChangeFoam. Finally, the parameterization of cavitatingFoam that allows reaching physical results will be detailed.

Key words: Cavitation, Hydrofoil

References

[1] F. Peng Kärrholm, H. Weller and N. Nordin, 2007, "Modelling injector flow including cavitation effects for diesel applications", Proc. FEDSM2007, 5th Joint ASME-JSME Fluids Engineering Conference, July 30 - August 2, 2007, San Diego, California, USA.

[2] E. Amromin, J. Kopriva, R.E.A. Arndt, M. Wosnik, 2006, "Hydrofoil drag reduction by partial cavitation", J. Fluids Eng. 128, 931-936.