EX154

Cavitating Flow Over a Hydrofoil

Cavitating flow over a cambered two-dimensional wing section is simulated using FLUENT. The flow angle over the NACA 66 (MOD) hydrofoil is chosen to represent conditions that are common in water pump and marine propeller applications. Excellent agreement with experimental data is obtained for mid-chord cavitation, and satisfactory agreement is obtained at the trailing edge of the cavitation region.

Cavitation on foil profiles such as marine propellers, pump impellers, ship hulls, and other submerged structures is something equipment designers try to avoid. The onset of cavitation degrades the performance of the foil (in hydraulic machinery) and can lead to erosion of the component surface. It typically occurs when the profile moves through a liquid at high speed and the pressure in the liquid drops below the vaporization pressure, often under isothermal conditions. Vapor bubbles form, but rapidly disappear when they migrate to regions of higher pressure.

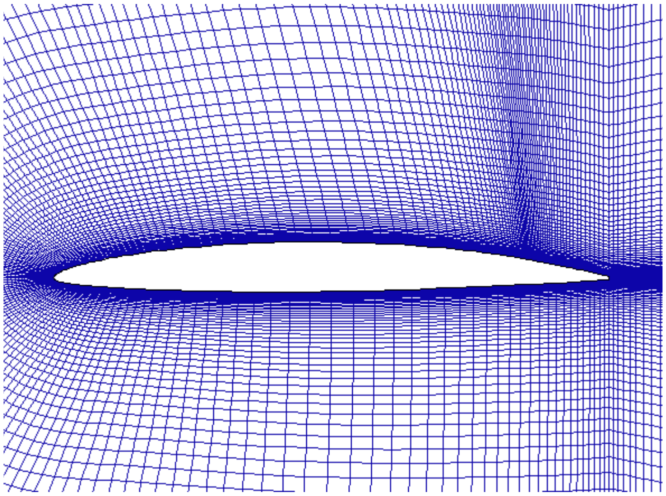

Figure 1: The quadrilateral grid used for the simulation

In this example, the algebraic slip mixture model in FLUENT is used to simulate cavitating flow over a hydrofoil. This multiphase flow model solves a single set of transport equations for a multiphase mixture along with an equation for the slip velocity between the phases. For cavitating flow, inter-phase mass transfer is also included in the calculation. The mass of vapor produced depends on the vapor density and the anticipated average size and density of vapor bubbles.

Liquid water is used as the primary or dominant phase, and water vapor is used as the secondary phase. The density ratio of liquid to vapor is 50,000. Both the liquid and vapor phases are treated as incompressible fluids. The Reynolds number based on the free stream velocity and chord length (the length of the profile along the upper surface) is 3 x 10^6. The angle of attack of the flow approaching the hydrofoil is 1°. A bubble number density of 1 x 10^6 bubbles/m^3 is used. To match the experimental work reported in Reference 1, calculations are carried out for conditions that correspond to a cavitation number of 0.38. The cavitation number is the ratio of the difference of the local static and vapor pressures to the dynamic head. This parameter should correlate with the pressure coefficient in the cavitation region.

The mesh around the NACA 66 (MOD) profile is shown in Figure

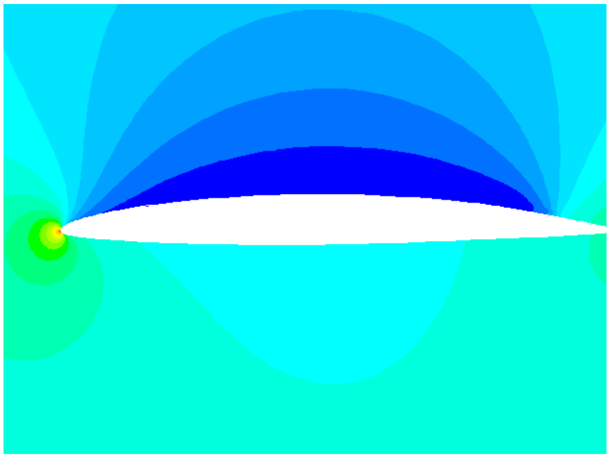

Figure 2: Contours of static pressure surrounding the hydrofoil

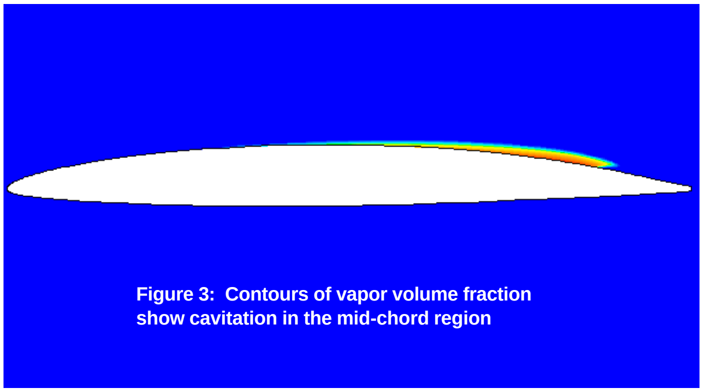

Figure 3: Contours of vapor volume fraction show cavitation in the mid-chord region

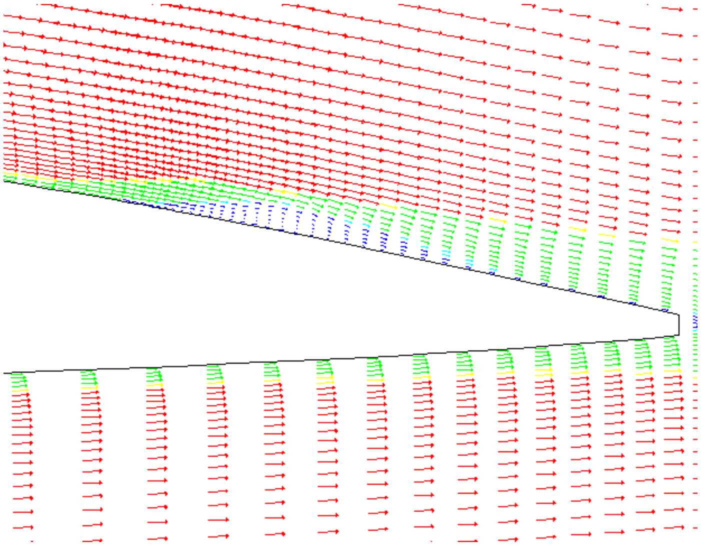

Figure 4: Velocity vectors in the trailing section of the cavitation region

1. A C-type structured quad mesh of 19,490 cells is used for the simulation. Velocity magnitude and direction are specified at the inflow boundary and zero gauge pressure is specified at the outflow boundary. A no-slip condition is specified at the profile surfaces. Experiments suggest that temporal fluctuations in cavity length, though present, are negligible and hence steady-state calculations are performed.

Contours of static pressure around the profile are shown in Figure 2. Lift is indicated by the pressure differential across the foil, as expected. Contours of vapor volume fraction are shown in Figure 3. The cavity begins in the mid-chord region and ends at approximately 86% of the total chord length.

Velocity vectors in the trailing region of the cavity are shown in Figure 4. Reverse flow and separation, which are not normally present in a fully wetted case (i.e. in the absence of cavitation), are evident in the figure.

The negative static pressure coefficient based on the FLUENT predictions is compared to the experimental data (Ref. 1) in Figure 5. The C_p value in the cavity region is 0.38, which is equal to the cavitation number anticipated by the specified flow conditions. This measure of agreement means that the static pressure in the cavity is very well predicted. The agreement is less accurate, however, near the trailing region. Here the collapse of the bubble, corresponding to a rise in pressure, is more abrupt as predicted by the FLUENT simulation than that recorded in the experiment.

In summary, FLUENT has been used to predict mid-chord cavitation around a NACA 66 (MOD) hydrofoil. Excellent agreement for the pressure coefficient in the front and middle sections of the cavity has been demonstrated. At the trailing edge of this region, satisfactory agreement was achieved. Overall, the results suggest that the cavitation model in FLUENT 6.0 can be used with confidence for other cases involving this complex multiphase phenomenon.

Reference 1. Shen, Young T., and Paul E. Dimotakis, “The influence of surface cavitation on hydrodynamic forces”, The 22nd American Towing Tank Conference, St. John, Newfoundland, CANADA, August 1989.

Figure 5: Pressure coefficient as a function of normalized chord length showing FLUENT results compared with data