Prediction of the Cavitation Behavior Around Hydrofoils

***Byung-Jae Ahn* [1], *Youn J. Kim* [2]**

1. HYUNDAI Mobis, 80-10 Guseong-Eup, Yongin-Shi, Gyunggi-Do 449-910, Korea, bjahn@mobis.co.kr
2. School of Mechanical Engineering, SungKyunKwan University, 300 Cheoncheon-dong, Suwon 400-746, Korea, yjk@skku.ac.kr

Corresponding Author: Byung-Jae Ahn

Abstract

One of the most important technical requirements imposed on a hydraulic machine is due to cavitation. In the stages of heavily developed cavitation and flashing of liquid to vapor phase, the flow is choked and unresponsive to the downstream pressure, and the downstream is affected seriously because of unsteady column collapse owing to the vapor condensation [1]. Also collapsing bubbles and cavities are responsible for the characteristics noises of cavitation. Therefore, cavitation is important because it may not only restrict the flow capacity of a system, but also generate unacceptable noise and cause erosion and failure. Furthermore, in extreme cases it may cause instabilities that may damage or destroy a system.

The present work is an investigation of the cavitation behavior around several types of hydrofoil with the general-purpose CFD package Fluent V.6.0. Several practical hydrofoils are presented: on an isolated NACA 0015, 16012 and 64108 foils. The model presented here aims to be flexible and simple: flexible to allow it to be used on a variety of conditions, simple also to allow unsteady or three-dimensional flows. And it can be made to fit experimental results in the near future or to fit the result of others.

The governing equations make use of full Navier-Stokes equations with the realizable $k-\varepsilon$ model for calculation of incompressible, steady and transient cavitating flow. These equations are discretized by finite volume method and solved by SIMPLEC algorithm.

There are two approaches for the numerical calculation of multiphase flows; the Euler-Lagrange approach and the Euler-Euler approach in Fluent code [2]. In this paper, the mixture model of the Euler-Euler approach was used. In this method, the phases are treated as interpenetrating continua. It solves for the mixture momentum equation and prescribes relative velocities to describe the dispersed phases. Also it can be used without relative velocities for the dispersed phases to model homogeneous multiphase flow. The computation will be done for two types of the grid: type "H" and type "O".

The aim of the present study is to obtain an overall view of the cavitating flow by examining in hydrodynamic characteristics of several types of hydrofoil. For quantitative and qualitative assessments of cavitation characteristics with cavity length, void fraction and prediction of lift breakdown around foils are depicted and discussed.

Keywords: Cavitation, Hydrofoil, Navier-Stokes equation, Cavity length, Void fraction

References

[1] S. C. Li, *"Cavitation of Hydraulic Machinery"*, Imperial College Press, London (2000).
[2] *"Fluent 6.0 User's Guide"* Vol. 1~5, Fluent Inc. (2001).