Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2015,27(1):105-111
DOI: 10.1016/S1001-6058(15)60462-0
www.sciencedirect.com/
science/journal/10016058

Numerical research on the performances of slot hydrofoil*

WEI Qun (魏群) [1,2], CHEN Hong-xun (陈红勋) [1], ZHANG Rui (张睿) [1]
1. Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University, Shanghai 200072, China
2. Institute of Civil Engineering, Nanchang University, Nanchang 330031, China, E-mail: qunwei@shu.edu.cn

(Received December 21, 2013, Revised July 3, 2014)

Abstract: The paper presents an numerical study of the hydraulic and cavitating characteristics of a slot hydrofoil at the angle of incidence of 6^{o} . Results indicate that the performance of this slot hydrofoil is better than the original hydrofoil at non-cavitation condition, but deteriorates sharply once cavitation occurred. To improve the performance, a new splitting scheme was put forward and optimization research was carried out at the same indicent angle, numerical results show that the optimized slot hydrofoil achieves better hydraulic and cavitation performances.

Key words: hydrofoil, performance, optimal design, cavitation inhibition

Introduction

Hydrofoil is the key factor of axial-flow pump's design, foil section has direct effect on the performance. The high efficiency working area of axial-flow pump is very narrow, with the change of external operation environment it may work on low efficiency condition, companying by greater hydraulic loss and system instability. To improve the performance of axial-flow pumps at low flow rate, Chen applied slotted technology[1] on 791 hydrofoil and Zhang designed a slot hydrofoil[2] by hydraulic optimization design, which applied in the design of an axial-flow pump. It's validated that the efficiency of this axial pump with slot blades increases at low flow rate by numerical and experimental means.

Cavitation can give rise to hydraulic performance deterioration, noise, vibration and erosion damage, so cavitating character is another indicator evaluating the performance of a pump. In present paper, the research aiming at cavitating characteristics of this slot hydrofoil at the angle of incidence of 6° will be performed.

Cavitation flow around a hydrofoil is often a multiphase flow associated with turbulence, unsteady flows and phase change, etc. Many researches have been carried out to simulate the cavitation flow and noticeable progresses have been made in recent years. A common approach to modeling cavitation is the homogeneous flow theory, where the mixture density is introduced and only one single set of mass and momentum equations is solved. Different approaches have been proposed to generate the variable density field. One of those is adopting arbitrary barotropic equation of state for density[3-5]. Another approach is transport equation model(TEM), fulfilled by a supplementary equation or the simplified Rayleigh-Plesset equation controlling the convection of the vapor[6-10]. TEM models have similar format but with different source terms. Frikh et al.[11] analyzed the influence of different models on the simulation of cloud cavitation on 2-D foil section, the results showed a large resemblance between most of models and modifying parameters change the cavity shape and structure. Many experiences[12-14] have proved that CFD simulation can be used to analyze the cavitating behavior successfully with coupling suitable cavitation and turbulence models.

In this paper, the cavitating behavior of the slot

* Project supported by the National Natural Science Foundation of China (Grant No. 51379120), the Science and Technology Plan of Zhejiang Province (Grant No. 2011C11068), and the Shanghai Program for Innovative Research Team in University.
Biography: WEI Qun (1978-), Female, Ph. D., Lecturer
Corresponding author: CHEN Hong-xun,
E-mail: chenhx@shu.edu.cn

hydrofoil will be investigated by a homogeneous model on comparison with 791 hydrofoil for 6^{o} attack angle. Furthermore, to achieve good hydraulic and cavitation performances, optimal research of the slot hydrofoil will be conducted with Ansys Workbench12.1, in which the cavitation index is introduced as another optimization goal besides hydraulic parameters.

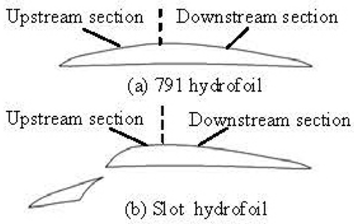

Fig.1 Geometry of hydrofoils

1. Geometry of hydrofoil

The research object of this work is an optimal slot hydrofoil based on 791 hydrofoil $(C = 0.158\ \text{m})$. 791 hydrofoil (Fig.1(a)) was verified to have good hydraulic performance and cavitation resistence through production practice, which was proposed by Guan[15]. The slot hydrofoil (Fig.1(b)) was designed by Zhang through optimization for maximum lift-drag ratio with constraint of lift.

2. Numerical methods

In order to figure out the cavitation performance of the slot hydrofoil, numerical simulations were carried out. The numerical model solved the unsteady Navier-Stokes equations, coupled with SST turbulence model and the Bakir[16] Rayleigh Plesset cavitation model with automatic near-wall treatment which can automatically switch from wall functions to a low-Re near wall formulation as the mesh is refined. The saturation pressure was 3 574 Pa. All simulations were done with general purpose CFD code ANSYS CFX 12.1. Equations were discretized on the element-based finite-volume, the second-order high resolution scheme and second-order backward Euler scheme were used separately for the advection term and transient term. Unsteady simulations were carried out with the initial values of corresponding steady simulation results.

2.1 *Validation of the numerical methods*

Leroux et al.[17] had taken researches on NACA 66 mod hydrofoil by both experimental and numerical means. To verify the validity of numerical methods, the adopted numerical methods were used to simulate the cavitation flow around NACA66 mod hydrofoil at two same conditions to experiments done by Leroux. Geometry was simplified to 2-D problem with 6^{o} attack angle, and the grid was generated with structured grid with 27 006 nodes (Fig.2). Figure 3 shows the predicted pressure distribution on the suction side of the hydrofoil for $\sigma = 1.41$, in which the pressure distribution agrees well with the experimental data. For $\sigma = 1.25$, the calculation results display that the cavity undergoes the growth and shedding process with main frequency of 3.57 Hz, and the experimental value is 3.625 Hz. From the above, it can be concluded that the numerical methods adopted in this paper can correctly simulate the cavitation flow around hydrofoil.

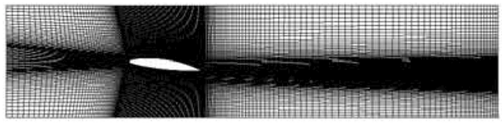

Fig.2 General view of the grid around NACA 66 mod foil

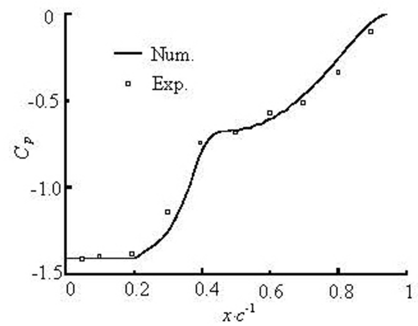

Fig.3 Pressure distribution on suction side of NACA 66 ($\alpha = 6^{o}$, $\sigma = 1.41$)

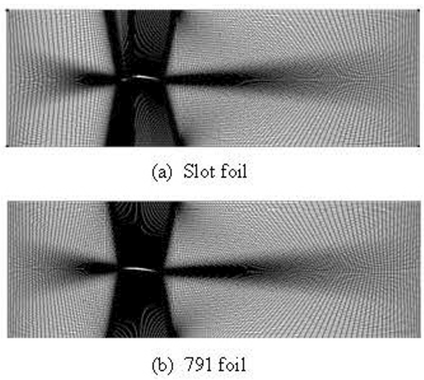

Fig.4 General view of the grid around foils

2.2 *Boundary condition , grid and time resolution*

The geometry was simplified to 2-D problem, which is fixed within a $11C$ long and $4C$ wide square cross test section. The slot foil was designed to improving the hydraulic performance at low flow rate, so the angle of incidence of the foil in this research is set