Modelling and Computation of Cavitation Erosion on a Ship Propeller

Andreas Peters, Department of Mechanical and Process Engineering, University of Duisburg-Essen, Duisburg, Germany

Udo Lantermann, Department of Mechanical and Process Engineering, University of Duisburg-Essen, Duisburg, Germany

Ould el Moctar, Department of Mechanical and Process Engineering, University of Duisburg-Essen, Duisburg, Germany

Long Abstract

Introduction

High flow velocities, induced by high rotation rates of ship propellers, generate pressures below the vapour pressure of water at common sea temperatures. As soon as small gas-filled bubbles, so called cavitation nuclei, reach these regions, they grow to vapour-filled cavitation bubbles. Once the cavitation bubbles are exposed to higher pressures, they rapidly collapse and radiate pressure waves of high amplitudes. These pressure waves may force the collapse of further cavitation structures in the flow field. When bubbles collapse close to a rigid boundary, this process is always asymmetrical, because of the boundary's influence on flow. These bubbles then form high velocity waterjets – microjets –, which are able to cause pressures beyond the plastic resistance of commonly used materials in technical applications. In marine technology propellers and rudders of ships are exposed to cavitation erosion under certain conditions of the cavitating flow. This may first lead to surface deformation during an incubation period and later even to material erosion.

In the present work, the cavitating flow around a ship propeller is numerically simulated and the potential erosion on the propeller blades is predicted with a developed erosion model. The method to predict cavitation erosion is validated by comparing numerical results of the flow over a NACA 0015 hydrofoil to experimental erosion prediction using a new coating technique.

1. Methods

The cavitating flows are simulated using an implicit, pressure-based flow solver out of the open source package OpenFOAM [1]. An Euler-Euler approach is used, which assumes the phases of liquid and vapour to be continuous. The flow is supposed to be a homogeneous mixture of the two phases. The equations of conservation of the mixture are the equations of continuity and momentum. The interfaces between the phases are tracked with a Volume of Fluid (VoF) method. The processes of evaporation and condensation are modeled by the cavitation model of Schnerr and Sauer [2], which is based on a simplified Rayleigh-Plesset equation for bubble dynamics. To simulate unsteady forms of cavitation, like the shedding process of cloud cavitation, the correction of turbulence by Reboud et al. [3] is applied. It accounts for the reduction of the turbulent kinetic energy in highly compressible regions.

Erosion is predicted with a developed model, which assumes that the presence of microjets on a solid surface leads to erosion based on the hypothesis by Dular and Coutier-Delgosha [4]. The information from the flow solution is used, to assess the possibility of microjet occurrence and material damage. The erosion potential is stated to be a product of both the amount of impacts in a certain area, as well as the intensity of these impacts. Semi-empirical formulas are used to obtain a critical velocity, based on material properties and a local jet velocity, based on the flow conditions. When the local jet velocity exceeds the critical velocity, a pit-shaped surface deformation is supposed to take place. The erosion model is able to identify erosion sensitive areas and to distinguish their erosion potential.

2. Results

The cavitating flow over a NACA 0015 hydrofoil at 5° angle of attack is investigated for different cavitation numbers. Due to a low pressure region at the leading edge of the foil a sheet cavitation develops downstream with the flow (see Fig. 1 and 2). For a cavitation number of 1.19 the sheet cavitation differs periodically in form and length, though it does not form a cloud cavitation. This leads to the prediction of erosion at the closure region of the sheet cavitation, which is in agreement with first experimental observations. The cavitating flow shows a different behaviour for a lower cavitation number of 0.95. Here, the sheet cavitation rolls up downstream and forms a cloud cavitation, which detaches from the foil and travels beyond the foil's surface. Since the clouds mostly collapse downstream behind the foil they do not lead to an erosion being predicted in the aft part of the foil. Though, erosion is being predicted at the leading edge, where the sheet cavitation starts to detach from the hydrofoil.

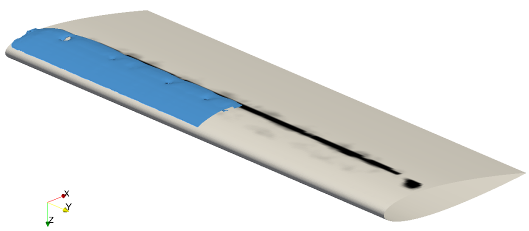

Figure 1. Cavitation erosion at $\sigma = 1.19$

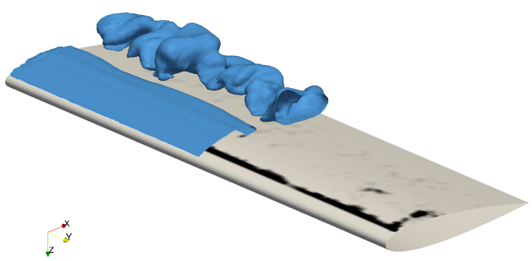

Figure 2. Cavitation erosion at $\sigma = 0.95$

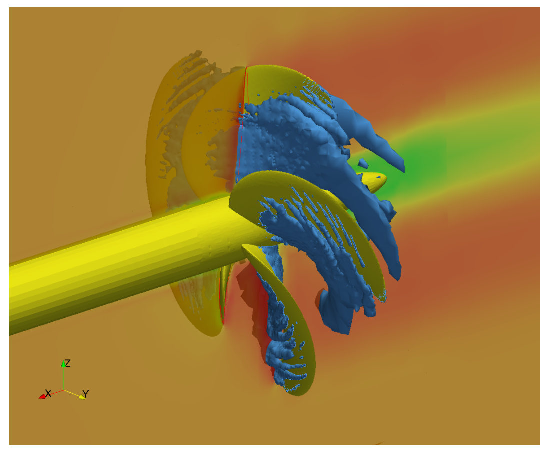

Figure 3. Cavitating propeller from perspective view

Figure 3 shows sheet cavitation on the blades of a rotating propeller. Both the suction side of the blades, as well as the pressure side are exposed to cavitation. As soon as this type of sheet cavitation gets unsteady in terms of variations in form and length, while touching the blades, erosion is supposed to happen. In this work, erosion on the propeller blades is predicted, by using the developed erosion model. The prediction of cavitation erosion is then compared to experimental measurements, where an erosion coating technique was applied on the propeller.

References

[1] OpenFOAM Foundation. OpenFOAM. [http://www.openfoam.org/], 2014.

[2] J. Sauer and G. H. Schnerr. Unsteady Cavitating Flow - a New Cavitation Model Based on Modified Front Capturing Method and Bubble Dynamics. In *Proceedings of FEDSM, 4th Fluids Engineering Summer Conference*, 2000.

[3] J.-L. Reboud, B. Stutz, and O. Coutier-Delgosha. Two phase flow structure of cavitation: Experiment and modeling of unsteady effects. In *Proceedings of the 3rd International Symposium on Cavitation, Grenoble, France*, 1998.

[4] M. Dular and O. Coutier-Delgosha. Numerical Modelling of Cavitation Erosion. *International Journal for Numerical Methods in Fluids*, 61:1388–1410, February 2009.