Wear 338-339 (2015) 189–201

Contents lists available at ScienceDirect

Wear

journal homepage: www.elsevier.com/locate/wear

Numerical modelling and prediction of cavitation erosion

Andreas Peters*, Hemant Sagar, Udo Lantermann, Ould el Moctar

Institute of Ship Technology, Ocean Engineering and Transport Systems, University of Duisburg-Essen, Bismarckstr. 69, 47057 Duisburg, Germany

ARTICLE INFO

Article history:
Received 26 March 2015
Received in revised form
15 June 2015
Accepted 15 June 2015
Available online 23 June 2015

Keywords:
Cavitation
Erosion
Multiphase flow
Computational fluid dynamics
Erosion model
Reboud's correction

ABSTRACT

This paper addresses the prediction of cavitation erosion using a numerical flow solver together with a new erosion model. Numerical flow simulations were conducted with an implicit, pressure-based Euler–Euler multiphase flow solver in combination with the developed erosion model. The erosion model refers to the microjet hypothesis and uses information from the flow solution to assess the occurrence of microjets in specific areas. The ability of the numerical code to simulate cavitating flows was shown by comparison with experimental tests of sheet cavitation over a NACA 0009 hydrofoil. The numerical prediction of cavitation erosion was compared to measured erosion in experimental tests of an axi-symmetric nozzle and shows good agreement regarding the erosive areas in general and the areas of highest erosion. Aim of this work is the assessment of erosion sensitive areas, as well as the erosion potential of cavitational flow during the incubation period.

1. Introduction

Cavitation occurs when small gas filled cavitation nuclei reach low pressure regions. When the local field pressure approaches the saturation pressure of the fluid, an evaporation process is started. This causes the cavitation nuclei to grow to vapour filled "cavitation bubbles". As soon as the pressure inside the bubbles exceeds the surrounding field pressure, the bubbles will suddenly collapse and condensate. In addition to cavitation bubbles, different types of cavitation may be identified, containing accumulations of these single cavitation bubbles. They can be distinguished depending on their shape and dynamical behaviour. "Sheet cavitation" is often formed at the leading edge, on the suction side of a hydrofoil. This type of cavitation can be more or less stationary, where the change of its form is only marginal. Depending on the flow conditions, though, the sheet cavitation may also show a strong transient behaviour, where its growth and shrinkage are harmonic. The so-called "cloud cavitation" may be shed from an unsteady sheet cavitation by a "re-entrant jet" as described by Callenaere et al. [1] and Decaix and Goncalvès [2]. This re-entrant jet is a result of the cavitation due to a blocking effect on the liquid flow. The flow is going to be reversed at the end of the sheet cavitation. Analogue to the dynamics of single bubbles, the volume of a vapour cloud oscillates. The collapse of the cloud cavitation occurs further downstream in higher pressure regions.

The instantaneous implosion of these cavitation clouds leads to the generation of pressure waves of high amplitudes. These pressure waves are regarded as a main mechanism giving rise to erosion. Fortes-Patella et al. [3] suggested a cavitation erosion model, where the potential energy of the macroscopic cavitation structures is regarded as the main factor that generates erosion. The potential energy of a cloud cavitation is supposed to be converted into acoustic energy of pressure waves, which travel through the fluid and are able to damage a surface directly.

The mechanism of cloud collapse is also regarded as the main damaging mechanism by Wang and Brennen [4]. In their model, the fully non-linear Rayleigh–Plesset equation is used to simulate the interactions of a spherical bubble cloud, consisting of single cavitation bubbles, with the liquid. The acoustic pressure radiated by the spherical cloud is calculated and employed to quantify the damage potential of the cloud.

Further erosion models concentrate on different flow properties, which may be calculated by using numerical methods. Li [5] stated a numerical erosion model, where the absolute pressure needs to exceed a threshold pressure for erosion to be predicted. Nohmi et al. [6] proposed multiple indices based on pressure, vapour volume fraction and their derivatives. These indices may then be used to assess the local erosion from the flow simulation.

Another hypothesis implies that the radiated pressure waves are not able to damage a surface directly since their pressure amplitudes are strongly attenuated when moving through the fluid. It is believed that these pressure waves initiate the

* Corresponding author. Fax: +49 20 33792 779.
E-mail addresses: andreas.peters@uni-due.de (A. Peters), hemant.sagar@uni-due.de (H. Sagar), udo.lantermann@uni-due.de (U. Lantermann), ould.el-moctar@uni-due.de (O. el Moctar).

http://dx.doi.org/10.1016/j.wear.2015.06.009
0043-1648/

oscillation and collapse of other cavitation bubbles. When bubbles collapse close enough to a material surface, this process is always asymmetrical as the flow through the bubble is disturbed by the surface itself [7]. This, in turn, produces a liquid waterjet, also called "microjet", which flows through the bubble and breaks it up into partial cavities. The collapse process was investigated experimentally and theoretically by Brujan et al. [8] as well as numerically by Lauer et al. [9]. Because of the asymmetrical collapse process, the microjet is almost always pointing towards the material surface. As shown by Field [10] and Haller Knežević [11], the impact of the jet onto the wall leads to the generation of a shock wave radiated perpendicularly away from the wall. This phenomenon induces a very high pressure near the wall, the so-called "water hammer pressure", which may be higher than the yield strength of common steel materials and able to damage a solid surface [12]. Dular and Coutier-Delgosha [13] and Dular et al. [14] stated an erosion model, where the velocity of a microjet needs to exceed a certain velocity threshold to be erosive for a certain material. Both the amount of pits on a surface, as well as the totally damaged surface is calculated further on.

In the present approach, a numerical erosion model has been developed following the work of Dular and Coutier-Delgosha [13] to predict the most threatened areas of erosion – the erosion sensitive areas – as well as the intensity of erosive impacts during the incubation period – the erosion potential.

2. Numerical method

In the present work, the open source CFD software package *OpenFOAM* [15] was used to simulate cavitating flows and develop models to predict cavitation erosion.

2.1. Euler–Euler two phase flow

Cavitating flows are multiphase flows involving phase changes. In the present work, the Euler–Euler approach is adopted, which deals with both liquid and vapour phase on a fixed Eulerian grid, where the flow is treated as a homogeneous mixture of the two incompressible, isothermal phases. A Volume of Fluid (VoF) method is utilized to track interfaces between the phases. For this approach, the equations for conservation of mass and momentum of the mixture are defined as

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} + \frac{\partial p}{\partial x_j} = \mu \frac{\partial}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + S_{\mathrm{vf}}. \tag{2}$$

u_i is the velocity in the coordinate direction x_i and p is the pressure. t is the time and ρ and μ are the density and dynamic viscosity of the homogeneous mixture. S_{vf} are source terms due to volume forces like gravitation.

Characteristic for the Euler–Euler approach with a VoF method is the introduction of a volume fraction α, which defines the volume of vapour or liquid occupied in the current cell. The vapour volume fraction is given by

$$\alpha = \frac{V_{\mathrm{v}}}{V_{\mathrm{v}} + V_{\mathrm{l}}}, \tag{3}$$

with the volumes of liquid V_{l} and vapour V_{v} of the cell. The densities and viscosities of the pure phases are constant. The mixture properties of density and dynamic viscosity can then be calculated with the volume fraction:

$$\rho = \alpha\rho_{\mathrm{v}} + (1-\alpha)\rho_{\mathrm{l}}, \quad \mu = \alpha\mu_{\mathrm{v}} + (1-\alpha)\mu_{\mathrm{l}}. \tag{4}$$

ρ_{l} and ρ_{v} are the densities μ_{l} and μ_{v} are the viscosities of pure liquid and pure vapour, respectively. α is obtained from an additional convective transport equation:

$$\frac{\partial \alpha}{\partial t} + \frac{\partial \alpha u_i}{\partial x_i} = S, \tag{5}$$

where S is the source term of the net phase change, defined as

$$S = S_{\mathrm{e}} - S_{\mathrm{c}}. \tag{6}$$

The terms on the right hand side are the phase change rates of evaporation S_{e} and condensation S_{c}. These source terms are obtained from a cavitation model.

2.2. Schnerr–Sauer cavitation model

In the present work, the cavitation model by Sauer and Schnerr [16] is applied. The model is based on the fact that the vapour phase can be defined by a finite number of single bubbles per volume of liquid. The vapour volume is then a function of the number of bubbles per liquid and of their size. Therefore, a definition of the vapour volume fraction is stated as

$$\alpha = \frac{n_{\mathrm{b}}\frac{4}{3}\pi R_{\mathrm{b}}^3}{1 + n_{\mathrm{b}}\frac{4}{3}\pi R_{\mathrm{b}}^3}, \tag{7}$$

with n_{b} as the number of bubbles per volume of liquid and R_{b} the bubble radius. Furthermore, the dynamics of bubbles are embedded by using a simplified form of the Rayleigh–Plesset equation. In the Rayleigh–Plesset equation the pressure difference $p_{\mathrm{b}} - p$ is the dominant term, with p_{b} being the pressure inside the bubble. Therefore, effects due to inertia, surface tension, viscosity and relative velocities can be neglected, which leads to the following simplified form:

$$\frac{dR_{\mathrm{b}}}{dt} = \sqrt{\frac{2}{3}\frac{p_{\mathrm{b}} - p}{\rho_{\mathrm{l}}}}. \tag{8}$$

The source terms for evaporation and condensation are then deduced from the continuity equation by using expressions (7) and (8):

$$S_{\mathrm{e}} = \frac{\rho_{\mathrm{v}}\rho_{\mathrm{l}}}{\rho}\frac{3\alpha(1-\alpha)}{R_{\mathrm{b}}}\sqrt{\frac{2}{3}\frac{p_{\mathrm{b}} - p}{\rho_{\mathrm{l}}}}, \quad \text{for } p < p_{\mathrm{v}},$$
$$S_{\mathrm{c}} = \frac{\rho_{\mathrm{v}}\rho_{\mathrm{l}}}{\rho}\frac{3\alpha(1-\alpha)}{R_{\mathrm{b}}}\sqrt{\frac{2}{3}\frac{p - p_{\mathrm{b}}}{\rho_{\mathrm{l}}}}, \quad \text{for } p \geq p_{\mathrm{v}}. \tag{9}$$

Depending on the local field pressure, a process of evaporation or condensation is started, causing the vapour volume to either grow or shrink.

2.3. Turbulence effects on cavitation

The turbulence in a flow may have an essential influence on the dynamics of cavitation. The standard turbulence models are not able to model unsteady cavitating flows with incompressible flow solvers. They do not standardly account for turbulence effects on the vapour pressure. Additionally, they do often fail to enable the re-entrant jet, which causes the harmonic cloud shedding. This leads to a steady sheet cavitation most of the time, as first noticed by Reboud et al. [17]. Therefore, some additional effects of turbulence have to be considered to calculate cavitating flows.

According to Singhal et al. [18] it could be shown in experimental investigations that turbulent pressure variations have an effect on the local vapour pressure. This can be accounted for by