Numerical Simulation of Various Mechanisms of Cavitation Erosion

Ignacijo Biluš\(^1\), Luka Lešnik\(^1\), Matevž Dular\(^2\)

Abstract
The results of numerical simulations of cavitation at Venturi section are presented in the contribution. A fully compressible, cavitating flow simulations was performed to resolve the formation of the shock waves at cloud collapse which are believed to be directly related to the formation of the damage. Good agreements were noticed between calculations and tests performed in previous work. The numerical simulations with short time step were performed to investigate the processes of cavitation collapse and shedding off in detail. The results indicate the importance of different individual cavitation structures collapse on erosion potential. Present analysis results shows a great potential for future development of techniques for accurate predictions of cavitation erosion by numerical means only.

Keywords
Cavitation, Numerical simulation, Erosion

\(^1\)Faculty of Mechanical Engineering, University of Maribor, Slovenia
\(^2\)Faculty of Mechanical Engineering, University of Ljubljana, Slovenia

Corresponding author: matevz.dular@fs.uni-lj.si

INTRODUCTION

Since Rayleigh reported on the cavitation erosion issue on ship propellers in 1917, considerable progress has been made to discover its hydrodynamic mechanisms [1-4]. Two main mechanisms are usually discussed: the micro-jet and pressure shock wave. For the case of the micro-jet it is believed that the liquid jet penetrates the bubble as the surrounding pressure is imbalanced. The jet velocity can reach a magnitude order of 100 m/s, and when it impacts the solid wall enormous stresses that cause pit formation occur [5]. On the other hand the pressure shock wave approach, considers the bubbles to remain spherical during the collapse what, at the final stage causes a shock wave generation with a magnitude order of several MPa [6]. Fortes-Patella et al. [7] studied the interaction between shock wave emitted by the implosions of a spherical bubble and material deformation. They concluded that the cavitation induced damage is directly related to the pressure shock wave and the characteristics of material. Subsequently, they proposed a so-called energy cascade theory. It postulated that potential energy contained in a macro cavitation cloud, considering the conservation of energy, would transfer into the radiation of acoustic pressure wave, which might be emitted either by spherical bubble or vortex collapse as well as micro jet.

The rapidly development of computational power and numerical simulation technology, increase the potential and the accuracy of the methods for the prediction of cavitating flow and cavitation erosion power. Dular & Coutier-Delgosha [8] presented a prediction method of cavitation damage based on the micro-jet theory, by coupling a computational fluid dynamics (CFD) and the proposed erosion model.

In the present paper we are comparing results of numerical simulations against experimental data obtained by Petkovšek & Dular [9, 10]. We performed fully compressible, cavitating flow simulations to resolve the formation of the shock waves during cavitation collapse phase which is directly related to the formation of the damage in the vicinity of the solid material surfaces. We have confirmed that cavitation cloud shedding causes different typical hydrodynamic mechanisms and results in extreme conditions which are connected with material erosion. The results are valuable for further numerical investigation and development of cavitation erosion prediction models.

1. EXPERIMENT

Numerical simulation was performed on the case of Venturi geometry installed into a small cavitation tunnel at the Laboratory for Water and Turbine Machines, University of Ljubljana. The experimental analysis of cavitation erosion process was performed in the past and extensively reported [9, 10]. According to this, only a brief description with key features of experimental set-up is given in the following.

The Venturi geometry (Fig. 1a), which was 10 mm wide with a converging angle of 18° and diverging angle of 8°. The throat dimensions were 10×10mm². The test section was made out of transparent plexi glass so that observation from all directions was made possible. The section was mounted into a cavitation tunnel (Fig. 1b) with a closed loop circle what enables to vary both the flow rate and the system pressure. This way a wide range of operating points can be achieved.
The idea was to obtain sufficient damage in a very short period of time (in about 1 second) experimentally. To set the operating conditions, first the test rig pressure was set to a desired value (490000 Pa, absolute pressure), then the valve upstream of the test section was closed and the pump was switched on to a determined rotating frequency. As the upstream valve was rapidly opened the velocity increased from 0 to 27.4 m/s in about 0.05 s achieving cavitation number $\sigma = 1.3$.

The idea of the experiment was to simultaneously record images of cavitation structures and cavitation erosion and by doing this to determine the mechanisms that lead to formation of pits. For this purpose several high speed cameras were used to simultaneously record the sequences of the cavitation dynamics and the damage sustained on the surface of the solid body.

The upper side of the foil is covered by vapour structures that obstruct the view, hence one needs to look at the foil from the bottom side to see the damage. Consequently the whole test section had to be made of transparent material and, equally important, the foil had to be thin enough so that the cavitation damage, which occurs on the side exposed to cavitation was also visible on the other side.

For the present experiment high speed cameras were synchronized and recorded at 30000fps at a reduced (608×86 pixels) resolution. One camera was used for observation of the aluminium foil while the other one captured images of cavitation structures.

Erosion was evaluated in image pairs: image at time $t+\Delta t$ was subtracted from image at time $t$. This way the change between times $t$ and $t+\Delta t$ could be detected and recognized as damage [9].

1.1 Results - cavitation erosion mechanisms

The general behavior of the developed cavitating flow follows a distinctive pattern where cavitation structures of different shapes and sizes are shed from the attached cavity (Fig. 2).

The pressure difference between the outer flow and the inside of the attached cavity, forces the streamlines to curve towards the cavity and the surface beneath it. This causes attached cavity to close and the formation of a stagnation point at which the flow is split into outer flow which reattaches to the wall and the re-entrant jet which travels upstream, carrying a small quantity of the liquid to the inside the cavity. As the re-entrant jet travels upstream it loses momentum due to shear forces, turns upwards and “cuts” the attached cavity, causing cavitation cloud separation (shedding). The cloud is then entrained downstream by the main flow and can violently collapse in a region of pressure recovery. During the separation, circulation around the structure appears. It causes the reshape of separated cloud and formation of horseshoe cavitation structure which results in damage in the region where vortex leg touches solid surface during break up phase. Meanwhile the attached cavity begins to grow and the process is periodically repeated [14]. The instants (C, E, H, L and O) when the damage occurs are highlighted by the darker colour.

Not less than 5 different mechanisms that lead to occurrence of cavitation erosion. [10].
Figure 3 shows the actual instants when the damage occurred during the measurements – the damage is enlarged (3 times) for better representation and appears smaller in reality (its position and shape are not altered).

The observed mechanisms were spherical cavitation cloud collapse (H), horseshoe cavitation cloud collapse (L), the “twister” cavitation cloud collapse (O) and in addition it was found that pits also appear at the moment of cavitation cloud separation (E) and near the stagnation point at the closure of the attached cavity (C).

2. MATHEMATICAL MODEL
2.1 The governing equations and homogenous flow model

The applied governing equations were based on the conservation form of the Reynolds averaged Navier-Stokes equations, including mass continuity (1), momentum equation (2) and energy equation (3):

\[
\begin{align*}
\frac{\partial \rho_m}{\partial t} + \frac{\partial}{\partial x_j} (\rho_m u_j) &= 0 \\
\frac{\partial \rho_m u_i}{\partial t} + \frac{\partial}{\partial x_j} \left[ (\rho_m + \rho_i) (u_i u_j + \delta_{ij} \rho_i) \right] &= -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \left( \mu_{ml} + \mu_i \right) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] \\
\frac{\partial (\rho_m E)}{\partial t} + \nabla \cdot \left[ \tilde{u} (\rho_m E + p) \right] &= \nabla \cdot \left( k_{eff} \nabla T \right)
\end{align*}
\]

The liquid phase and vapor phase are treated as a homogeneous mixture based on the volume of fraction. The mixture density and viscosity are defined as a function of vapor volume fraction:

\[
\rho_m = \rho_v \alpha_v + \rho_i (1-\alpha_v)
\]

\[
\mu_m = \mu_v \alpha_v + \mu_i (1-\alpha_v)
\]

where \( p \) is the pressure, \( \rho \) is the density, \( u \) is the velocity, \( \mu \) and \( \mu_i \) stand for the laminar viscosity and turbulent viscosity, \( \delta_{ij} \) is the Kronecker delta function, \( E = h - p \left[ \rho_v + u_i^2 / 2 \right] \), \( h \) is the entropy, \( k_{eff} \) is the effective conductivity, \( \alpha \) is the volume fraction. The subscripts \( mlv \) indicate the mixture, liquid and vapor, respectively.

2.2 Turbulence model

As known, the turbulence model plays a significant role in the prediction of cavitating flow. Since the standard \( k-\varepsilon \) model is over-estimating the eddy viscosity in the mixture region, it cannot effectively resolve the detachment of the cavity from solid surface and excessively attenuates the cavitation instability. However, the shedding motion and subsequent collapse are the primary reason causing cavitation erosion. Therefore, a modified Re-normalized group (RNG) \( k-\varepsilon \) model, proposed by Coutier-Delgosha et al. [11], was employed in this work. It can successfully reduce the eddy viscosity by defining the turbulent viscosity as:

\[
\mu_i = f(\rho_m)C_m \frac{k^2}{\varepsilon}
\]

\[
f(\rho_m) = \rho_v + \left( \frac{\rho_m - \rho_i}{\rho_i} \right)^n
\]

where the coefficient \( C_m=0.09 \), identical with \( k-\varepsilon \) model, and the exponent \( n=10 \), recommended by Coutier-Delgosha et al. [11].

2.3 Cavitation model

The vapor generation and disappearance are controlled by a mass transport equation model (TEM) based on the vapor volume fraction, expressed as:

\[
\frac{\partial \alpha_v}{\partial t} + \frac{\partial (\alpha_v u_j)}{\partial x_j} = \dot{m}^+ + \dot{m}^-
\]

The source term \( \dot{m}^+ \) and \( \dot{m}^- \) represent the mass rates of liquid evaporation and vapor condensation. In this paper, the Zwart-Gerber-Belamri model [12], deduced from the Rayleigh-Plesset equation, was applied, since it has, based on our previous experience, a precise cavitating prediction performance and a good convergence behavior. It is defined as:
\[
\dot{m}^+ = F_{\text{vap}} \frac{3r_{\text{nu}} (1 - \alpha_r) \rho_v}{R_B} \sqrt{\frac{2}{3}} \frac{p - p}{\rho_i} \quad \text{if } p < p_v \tag{9}
\]
\[
\dot{m}^- = F_{\text{cond}} \frac{3 \alpha_r \rho_v}{R_B} \sqrt{\frac{2}{3}} \frac{p - p}{\rho_i} \quad \text{if } p > p_v \tag{10}
\]

where \( F_{\text{vap}} \) and \( F_{\text{cond}} \) are the empirical calibration coefficients of evaporation and condensation, respectively, \( r_{\text{nu}} \) stands for the nucleation site volume fraction, \( R_B \) is the nucleation site radius. Vaporization is initiated at nucleation sites, which can be regarded as the non-condensible gases. \( \rho_v \) represents the water vaporization pressure. The recommended values of these coefficients are: \( F_{\text{vap}}=50 \), \( F_{\text{cond}}=0.01 \), \( r_{\text{nu}}=5 \times 10^{-4} \), \( R_B=2 \times 10^{-6} \)m and \( \rho_v=3574 \)Pa.

As compressible approach was adopted the vapour obeyed the ideal gas law and the liquid density variation was described via Tait equation:

\[
\rho_i = \rho_{\text{ref}} \sqrt{\frac{p + B}{p_{\text{ref}} + B}} \tag{11}
\]

where \( \rho_{\text{ref}} \) and \( p_{\text{ref}} \) denote the reference liquid density and pressure 200mm upstream of the Venturi section. As for constant \( B \) and \( n \), they are 300MPa and 7 for water, respectively.

3. NUMERICAL MODEL AND SIMULATION SETUP

To get a better accuracy and convergence behavior, the structured hexahedral grid was generated to model the fluid computational domain. The grid independence test was conducted on the basis of three kinds of mesh density. Considering the calculation time and accuracy, the grid with the total number of the elements about 0.5 million was applied. [13].

4. RESULTS

The numerical simulation results are presented in Figures 5-7. The left columns show isosurfaces of 10% vapour volume fractions while absolute pressure ratio \( p/p_{\text{max}} \) is shown in right columns. The value of \( p_{\text{max}} \) present the maximum pressure in each sequence analysed. The typical cavitation structure cycles are presented in time series with \( \Delta t=1.6 \times 10^{-6} \)s.

Figure 5 show the mechanism of stagnation point formation at closure of attached cavity. It is evident, that cavitation structure (left column) remains relatively stable. From pressure field (right column) it is evident, that elevated pressure region match the stagnation point created by flow, directed towards solid face. Following the fact that surface area where pressure peaks are present is relatively small, it is assumed that small scale cavitation structures (bubbles) collapse in this region. According to stable nature of attached cavity, it is expected the damage on surface to occur gradually. On the other hand, the damage process is limited by re-entrant jet development since it blocks the shedding of small scale structures.

The sequence shown on Figure 6 capture the cavitation cloud collapse process. Vapour fraction isosurface at \( t_0 \) show the cloud of bubbles separated from vapour structure at Venturi throat. It is evident from the pictures sequence that cavitation cloud volume reduces during movement through the region of pressure recovery. At \( t_r+3.3t \) pressure wave which is a result of violent collapse of vapour cloud reaches the surface directly beneath the position of cloud implosion. The distribution of high pressure area at Venturi wall which indicates the region of pressure peak confirms that pressure wave is a result of collapse of homogenous cloud cavity. According to this, the cluster of erosion pits is expected in the region of cloud structure collapse.

The process of cavitation cloud separation and collapse of horseshoe shaped cloud is shown at Figure 7 sequence. Vapour fraction isosurface at \( t_0 \) shows the re-entrant jet under the attached cloud. Between \( t_0 \) and \( t_r+3t \) the re-entrant jet lose momentum, turn upward and separate the attached cloud structure. At \( t_r+2t \) the gap between separated cloud and attached cavity is filled with liquid. The momentum change in the vicinity of the wall cause pressure wave and pit formation in the region.

The separated cavity which is present in the region at \( t_0 \) forms into a horseshoe shaped structure at the \( t_r+3t \) and \( t_r+2t \). The reason for this are local vortices caused by roll-up of separated structure during re-entrant jet propagation. The pressure recovery process causes the implosion of horseshoe shaped cloud at \( t_r+3.3t \). The phenomenon is connected to the pressure wave evident at pressure field at same moment. The erosion damage is expected in the “high pressure region” and in the region where the vortex leg touches the surface.
Figure 5. Numerical simulation results – closure of attached cavity.
Figure 6. Numerical simulation results – collapse of cavitation cloud.
Figure 7. Numerical simulation results – cavitation cloud separation and horseshoe structure collapse.
5. CONCLUSION

In this paper, fully compressible cavitating flow simulations were performed to resolve the formation of the shock waves during cavitation collapse phase in a Venturi section. At the beginning, the results were validated against visualization experiments performed on the cavitation tunnel [9]. The simulation results, obtained with a time step of $1.6 \times 10^{-4}$ s, are in a good agreement with the visualization experiments. The analysis confirmed that cavitation cloud shedding causes extreme conditions which are connected to different typical hydrodynamic mechanisms causing pressure waves in analyzed domain. On the other hand, the simulation results obtained with a time step of $1.6 \times 10^{-6}$ s showed the periodic nature of pressure instabilities during the collapse of four different cavitation structures types. The pressure peak areas are predicted by homogeneous model which corresponds to that by collapse of tiny bubbles in larger vapour structures and which considerably contributes to the erosive energy potential of cavitating flow in the pressure recovery region of Venturi section. The numerical model used and results obtained will be useful for further numerical investigation and development of cavitation erosion prediction models.

REFERENCES


