Numerical study of flow behavior in a cavitation tunnel using RANS with Scale-Adaptive Simulation (SAS) turbulence model in an OpenFOAM framework

V Hidalgo1,2,3, G Suárez2, J Erazo2, D Puga2,4, D Márquez2, I Benavides3, E Cando1, E Valencia1 and X Luo4

1 Departamento de Ingeniería Mecánica, Escuela Politécnica Nacional, Quito 170525, Ecuador
2 Laboratorio Informática-Mecánica, Escuela Politécnica Nacional, Quito 170525, Ecuador
3 Carrera de Ingeniería Automotriz, Facultad de Ingeniería y Ciencias Aplicadas, Universidad Técnica Norte, Ibarra 100105, Ecuador
4 State Key Laboratory of Hydro Science & Engineering, Tsinghua University, Beijing 100084, China

Correspondence: victor.hidalgo@epn.edu.ec; Tel.: +593-982-491-193

Abstract. The present paper focuses on the study of flow through a cavitation Venturi tunnel. Motivated by the results obtained in the studies of “Simultaneous observation of cavitation structures and cavitation erosion” and “Combined numerical and experimental investigation of the cavitation erosion process” by Dular et al. A structured mesh based on previous studies has been generated using the free software GMSH. The numerical study was performed using the Navier-Stokes equations with RANS approximation. For that, the k − ω − SST SAS turbulence model and the implemented Zwart-Gerber-Belamri have been applied in OpenFOAM. The results show that the phenomena in a vapor volume fraction where the formation, growth, detachment and collapse of the cavitation cloud can be appreciated. These results agree with the aforementioned studies. Furthermore, the results show the peak pressure formation corresponding to the detachment and collapse of the cloud during the cavitation cycle, which is the main reason for erosion. It is concluded that the model satisfactorily predicts the phenomena behavior on a Δt = 9 × 10^{-6}[s] being suitable to capture adverse pressure gradients.

1. Introduction

Cavitation causes multiple undesirable effects on hydraulic machinery, such as flow instability, reduced operating efficiency, increased noise, excessive vibrations and material surface damage [1]. The repetitive implosion of the cavities causes a build-up of stings in narrow areas, which erodes the material and leads to mass loss in the cavity [2]. Due to these reasons several studies of this phenomenon have been carried out from the beginning of the last century to the present day [3].

In 1917, Rayleigh first reported about this problem in the propellers of a ship, and from this evidence a considerable progress has been made to understand its behavior, such as the formation of the re-entry jet and the pressure shockwave [4]. The re-entry jet is the liquid that penetrates the cavity with an increase of velocity as the surrounding pressure is unbalanced [4]. Supponen O. et al. performed a qualitative and quantitative analysis of the microjet dynamics of a single cavitation bubble in a wide range of conditions [5]. Moreover, the cavity collapse in its final stage generates pressure shockwaves in several MPa order [6]. The damage caused by cavitation is related to pressure shockwaves, which is evident in the work of Dular M. et al [7][8]. These studies showed the cavitation effects by simultaneously observing cavitation structures and the surface of an aluminum foil in a Venturi section.
It was concluded that surface damage occurs during the separation of the cavitation cloud and in its collapse. In addition, the cloud size and its distance from the wall when collapsing do not influence the extent of the damage. In this context, flow simulation proves to be very useful for the understanding of cavity formation and the analysis of different parameters involved during its development.

The development of computational capacity and numerical simulation of computational fluid dynamics (CFD), has increased the accuracy of methods for cavitation flow prediction [4]. Hidalgo V. proposed a cavitation-erosion model based on energy released during cavity collapse and flow is assumed to be in a homogeneous mixture. The present study developed the model from the optimization of numerical CFD simulation of unsteady partial cavitation, considering asymmetric condensation and vaporization conditions. For the study of the aggressiveness of the water flow, the total affected region and the plastic deformation of the material, the Zwart-Gerber-Belamri Cavitation Model (ZGB) was applied. The ZGB cavitation model and a K-omega SST SAS turbulence model with ILES were implemented in OpenFOAM.

The results showed the development process of unsteady cavitation, cavity formation at the leading edge, re-entry jet, detachment and collapse of the cavity, which are consistent with the results in experimental studies [9]. On the other hand, Dular et al. conducted a CFD study of the fully compressible cavitation flow to solve the formation of shockwaves in cloud collapse. The ZGB cavitation model was used in ANSYS-Fluent and an RNG k-epsilon turbulence model with RANS. The results obtained were similar to the experimental visualization therefore it concludes that there are two pressure peaks in a cavitation cycle corresponding to the detachment and collapse of the cavity [4].

Due to the above, this work will continue the numerical study of the flow in a cavitation tunnel using RANS with the model proposed by Hidalgo V. [9] for the analysis of the cloud behavior. The results obtained from the numerical simulation will be compared with the experimental and numerical studies performed by Dular et al. [7].

2. Numerical Method Description

2.1. Mixture consideration

In the cavitating flow, the fluid is considered as a homogeneous mixture with phase transformation between vapor and water. Equation (1) represents the vapor fraction \( \alpha \), equation (2) represents the density \( \rho \), and equation (3) the dynamic viscosity \( \mu \) of the mixture [10].

\[
\alpha = \frac{V_v}{V} \tag{1}
\]

\[
\rho = \alpha \rho_v + (1 - \alpha) \rho_l \tag{2}
\]

\[
\mu = \alpha \mu_v + (1 - \alpha) \mu_l \tag{3}
\]

Where \( V \) is the total volume, the sub-indexes \( l \) and \( v \) correspond to liquid and vapor. For the phase transition, the mass transfer for unit volume \( \dot{m} \) is presented in equation (4).

\[
\dot{m} = \frac{\partial (\alpha \rho_v)}{\partial t} + \frac{\partial (\alpha \rho_v \bar{u}_i)}{\partial x_i} \tag{4}
\]

2.2. The Reynolds Averaged Navier-Stokes (RANS) equations

Turbulence model is used with the RANS equations, where equation (5) accounts for continuity and equation (6) for the moment [11].

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_i)}{\partial x_i} = 0 \tag{5}
\]

\[
\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = - \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \rho v \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] \tag{6}
\]
Where $\bar{u}$ is the mean (time averaged) velocity, $\bar{p}$ is the mean pressure, $t$ is time, $\rho$ and $\nu$ represent the flow density and the kinematic viscosity, respectively. The sub-indexes $i$ and $j$ represent the spatial coordinates [12].

2.3. Scale Adaptive Simulation (SAS) model
According to Menter, the SAS model is susceptible to fluctuations [13]. In SAS, the tensor $\tau_{ij}$ is solved using the length scale for turbulent viscosity [14], using equation (7).

$$L_{\nu\kappa} = \kappa \left| \frac{\bar{U}}{\bar{U}'} \right|$$ (7)

Where $\kappa$ is the von Kármán constant and has a value of 0.410 [12]. This equation represents a three-dimensional generalization of the definition of boundary layer considering a von Kármán length scale.

The terms $\bar{U}'$ and $\bar{U}''$ are auxiliary functions for the turbulence model $k$-$\omega$ [12].

In this context, the SAS term is an additional product term in the equation $\omega$ which increases when the flow equations start to have an unstable behaviour on the $k$ - $\omega$ SST SAS [12].

2.4. The Zwart-Gerber-Belamri (ZGB) cavitation model
This model is based on the mass transfer between interfaces $\dot{m}$, derived from the Rayleigh-Plesset equation [10]. The ZGB model is expressed in equation (8).

$$\dot{m} = \begin{cases} 
\dot{m}^+ = F_v \frac{3 \rho_{\text{nuc}} (1 - \alpha) \rho_v}{R_B} \sqrt{\frac{2}{3} \left( \frac{p_v - p}{\rho_t} \right)}, & \text{si } p < p_v \\
\dot{m}^- = -F_c \frac{3 \alpha \rho_v}{R_B} \sqrt{\frac{2}{3} \left( \frac{p_v - p}{\rho_t} \right)}, & \text{si } p > p_v 
\end{cases}$$ (8)

The values have been taken from Morgut et al. [15] for the calibration of the volume fraction values of the nucleation site $\rho_{\text{nuc}} = 5 \times 10^{-6}$ and of the vaporization constant $F_v = 300$. On the other hand, the predetermined values of the condensation constant are considered to be $F_c = 0.010$ [16]. Furthermore, the radius of the bubble is $R_B = 1.0 \times 10^{-6}$ [m] which is taken from the study performed by Zhang et al. [17].

3. Geometry and Solver

3.1. Geometric model
In order to model the Venturi section, the dimensions herein presented are based on the throttle length of the cavitation tunnel studied by Dular et al. [4] [8], which is considered as a characteristic dimension $L$ with a value of 10 [mm] as shown in Figure 1.

![Figure 1. Geometry dimensions of the Venturi section [mm].](image)

The converging profile used for this study is based on previous studies performed by Hidalgo et al. [12].
3.2. Computational Domain

The computational domain of the Venturi section is defined by the chord length “c” of 0.155 [m]. Distance increases at the entrance and at the flow exit at two and four times the chord length respectively [18]. The different faces are named as shown in figure 2 so the refinement zones and boundary conditions can be set. Additionally, “in” and “out” are assigned for the flow entrance and exit, “top” and “bot” for the upper and lower walls, “fab” for the frontal and posterior walls. A thickness of 10 [mm] has been set to identify the faces.

![Figure 2. Description of the computational domain.](image)

3.3. Mesh Generation

To generate the structured mesh with Gmsh, it was necessary to consider 27 points and 10 surfaces, as shown in figure 3. Mesh was generated avoiding high bias and sudden orthogonality change. For this, the lines that have same properties and divisions are identified.

![Figure 3. Points and sections of the Venturi domain.](image)
In order to obtain a good quality mesh and avoid results induction, a refinement is performed on the study section, figure 4. Thus, meshing focuses on the top and bottom ends and parameters of horizontal progression of each surface are varied to obtain a smooth transition.

Figure 4. Venturi section mesh.

3.4. Criteria for mesh quality
The proposed mesh is composed of 41 036 elements and the averaged y+ value is 7.38, which is within admissible range for this type of mesh according to previous studies [9].

The omega number (Ω) obtained is 0.492, this number describes the computational cost for the simulation. It relates the total number of elements with the total number of nodes [19].

The Courant number (Co) provides a measure for the speed at which information travels under influence of flow field during a time step through the cell size and it is a limiting factor for the performance of the numeric schemes. The Co is 0.166 which is within acceptable range of $0 < Co \leq 1$ [9].

3.5. OpenFOAM Solver
OpenFOAM version 4.1 is used in the operative system Ubuntu 16.04 LTS. Figure 5 shows the software development for this case study. The scheme depicts two parts, one corresponds for the simulation of the cavitating flow with the implementation of the solver vInterPhaseChangeFoam developed by Hidalgo V. [9]. In this implementation, the turbulence model $k - \omega$ SST SAS intervenes with the cavitation model Zwart-Gerber-Belamri (ZGB). The second part focuses on the results visualization.

Figure 5. Software development for the cavitation study.
3.6. Boundary conditions
The experimental study presented by Dular et al. [7] has a velocity of \( U = 24.7 \text{[m/s]} \), pressure of \( P = 454 \text{[KPa]} \), cavitation number of \( \sigma = 1.48 \) and Reynolds number of 247 000. These values are needed for the calculation of initial conditions of velocity and pressure. Table 1 shows the values for initial conditions.

Likewise, the turbulent kinetic energy, \( k \) (turbKE), and the specific dissipation rate of turbulence, \( \omega \) (turbulentOmega), of \( k - \omega \) SST SAS model [20].

Table 1. Initial conditions obtained for this case study

| Nombre                | Valor       |
|-----------------------|-------------|
| Velocity, \( U \) [m/s] | 6.175       |
| Pressure, \( P \) [KPa] | -107.084   |
| turbKE, \( k \) [m^2/s^2] | 0.083       |
| turbulentOmega, \( \omega \) [1/s] | 1.900       |

The boundary conditions for the computational domain at the entrance “in” and exit “out” are defined with the condition “patch”. The “fab” walls are defined as “empty” and the “top” and “bot” walls are defined as “wall”. Table 2 shows the boundary conditions based on the variables of the case study.

Table 2. Boundary conditions for each case variable.

| Boundary | in          | out         | fab 1, 2, 3 | top 1, 2, 3 | bot 1, 2, 3 |
|----------|-------------|-------------|-------------|-------------|-------------|
| alpha.water | fixedValue  | zeroGradient| empty       | zeroGradient| zeroGradient|
| k        | fixedValue  | zeroGradient| empty       | kqRWallFunction| kqRWallFunction|
| nut      | calculated  | calculated  | empty       | nutkWallFunction| nutkWallFunction|
| omega    | fixedValue  | inletOutlet | empty       | omegaWallFunction| omegaWallFunction|
| p_rgh    | zeroGradient| totalPressure| empty       | zeroGradient| zeroGradient|
| U        | fixedValue  | zeroGradient| empty       | noSlip      | slip        |

4. Results and discussion

4.1. Graphical comparison with previous studies
The qualitative results are presented in function of the water volume fraction, where, White color represents liquid water and black represents the vapor bubble. The numerical simulation is validated with the experimental and computational results obtained in the studies performed by Dular et al. [8] [11]. Figure 6 shows that the cavitation cycle partially agrees with previous studies because the unsteady behavior matches the ones presented by Dular et al., even though the slope is not identical. Moreover, it presents some differences, such as the time interval at which the phenomenon is captured and the cloud topology. This is a consequence of the turbulence model used in the aforementioned studies.

The \( k - \varepsilon \) RNG model was used, this is used for the study of external flows with small pressure gradients. Whereas for this study the \( k - \omega \) SST SAS and ZGB models implemented in OpenFOAM, help to capture the flow separation and it is suitable for adverse pressure gradients.

A time interval \( \Delta t = 9.000 \text{e}^{-6} \) [s] was considered in order to capture the phenomenon and an initial time of \( t_0 = 1.350 \text{e}^{-3} \) [s]. The detachment of the cavitating cloud was produced at the time \( t = t_0 + 80\Delta t \) and its collapse at \( t = t_0 + 175\Delta t \).
4.2. Absolute pressure analysis
For the analysis of the behavior between the cloud evolution and its induced damage, absolute pressure has been presented graphically in function of time on characteristic points of the section. Point A represents the cloud detachment and it is placed at 0.043 [m] from contraction start, point B represents its collapse and it is placed at 0.074 [m].

Figure 6. Graphical comparison of a cavitation cycle, a) results from the numerical simulation, b) experimental observation y c) simulation performed by Dular et al.

Figure 7. Absolute pressure at point A.
Figure 7 shows the pressure fluctuations at point A. There are six instances during the cavitation cycle where it can be observed that at time $t = 2.090 \times 10^{-3}$ [s] a peak pressure is produced $P = 4.800 \times 10^5$ [Pa], which corresponds to the cloud detachment.

At instant “i” the pressure has a value close to zero, which represents the cloud formation. At instant “ii” the cloud growth just before detachment is shown. At instant “iii” a peak local pressure is produced which corresponds to the instant of the “tail” detachment of the cloud with the boundary surface of attack. This high-pressure increment explains why the damage on the surface occurs also at different moments of the cloud collapse. Once detached, the instants “iv”, “v” and “vi” are shown. During these instants, pressure fluctuations are observed at the detachment point surroundings and how its value is still low next to the cloud according to its advance with the flow movement. Figure 9 depicts the changes in pressure on the B point corresponding to the cloud collapse, and similarly, six instants are set to capture the pressure fluctuations during the cloud collapse.

![Figure 8. Absolute pressure at point B.](image)

At instants “i” and “ii”, pressure tends to decrease according to the cloud evolution close to the surface. The cloud reaches a minimum pressure of $P = 7.000 \times 10^3$ [Pa], from there, instability at “iii” is observed. At instant “iv” the deformation caused by the re-entry jet can be seen. At “v” the cloud starts to collapse, pressure increases with a shockwave. Additionally, at “vi” a maximum peak pressure of $P = 2.000 \times 10^7$ [Pa] is observed which corresponds to the time $t = 2.961 \times 10^{-3}$ [s], due to the impact of the shockwave produced by the cloud collapse.

4.3. Re-entry jet formation
Also known as “micro-jet”, it is the responsible for the cavity separation with the surface when it reaches the attack border. In figure 7 (iii), vapor and water mixture in the “tail” of the cavity moves towards the attack border in the Venturi section. Under action of the high local pressure, a small vortex is generated with the re-entry jet. The vortex expands with the flow movement detaching the cavity of the attack border. The ZGB cavitation model can predict the structure of the cavity with high accuracy. However, this model cannot accurately simulate the re-entry jet formation which detaches from the surface [21]. This can be seen in figure 9 (a) where the velocity vectors corresponding at $v = 25.3$ [m/s] are shown during the detachment of the cloud at $t = 2.016 \times 10^{-3}$ [s].
On the other hand, the cavitation model evidences the shear action which generates the jet in the separated cavity. Figure 9 (b) shows the velocity vectors $v = 28.700$ [m/s] near the cavity before the cloud collapses at $t = 2.898 \times 10^{-3}$ [s]. The water flow that surrounds the cloud accelerates towards its interior, in opposite direction to the flow, forming a slit shape. At the cloud collapse, the jet that deforms the cavity maintains a perpendicular direction to the surface section. This provokes that the cloud collapse is not concentric in the nearby zone to the surface. [5].

5. Conclusions

It was possible to perform the numerical study of cavitating flow around a Venturi section by using RANS, PIMPLE algorithm, the $k$-$\omega$-SST SAS turbulence model and the implementation of the Zwart-Gerber-Belamri (ZGB) in OpenFOAM. Moreover, it is concluded that the use of the turbulence $k$-$\omega$-SST SAS model allows to capture the cavitation cycle in a time step of $9 \times 10^{-6}$ [s]. Even though the time step used in this study was smaller than the $1 \times 10^{-4}$ [s] used by Dular et al. in their numerical simulation using the $k$-$\epsilon$ RNG model and the experimental visualization. Additionally, the ZGB model fairly well predicts the cavitation cloud in a Venturi section.

Similarly, it was possible to simulate and describe the formation, growth, detachment and implosion of the cavitation cloud in the 2D Venturi model. The numerical approach used here was able to reproduce the re-entry jet during the collapse and the fluctuations on the surrounding pressure field. Furthermore, the study corroborated the peak pressures presented by Dular et al., obtaining a higher pressure emitted during the cloud collapse corresponding to $3 \times 10^7$ [Pa] and a lower of $4.8 \times 10^5$ [Pa] during the detachment. Similarly, the principal damage is caused by the cloud collapse mainly because to its topology and the re-entry jet formation.

Acknowledgments: The authors gratefully acknowledge the financial support provided by Escuela Politécnica Nacional for the development of this studio, which is part of the research projects PIJ 17-13 and PII-DIM-2019-06.

6. References

[1] Gupta A, Dhuri S, Chouhan D, Raskar S and Ingole Y 2018 Effects of cavitation in hydraulic machines IJRASET vol 6, pp 606–609
[2] Escaler X, Egusquiza E, Farhat M, Avellan F and Coussirat M 2006 Detection of cavitation in hydraulic turbines Mech. Syst. Signal Process vol 20 pp 983–1007
[3] Kozak J, Rudolf P, Hudec M, Stefan D and Forman M 2018 Numerical and experimental investigation of the cavitating flow within Venturi tube J. Fluids Eng. vol 141
[4] Jian W, Petkovšek M, Houlin L, Širok B and Dular M 2015 Combined numerical and experimental investigation of the cavitation erosion process J. Fluids Eng. vol 137, p. 051302
[5] Supponen O, Obreschkow D, Tinguely M, Kobel P, Dorsaz N and Farhat M 2016 Scaling laws for jets of single cavitation bubbles *J. Fluid Mech.* vol 802 pp 263–293

[6] Reisman G, Wang Y and Brennen C 1998 Observations of shock waves in cloud cavitation *Journal of Fluid Mechanics* pp. 255–283

[7] Petkovšek M and Dular M 2013 Simultaneous observation of cavitation structures and cavitation erosion *Wear* vol 300 pp 55–64

[8] Dular M and Petkovšek M 2015 On the mechanisms of cavitation erosion - coupling high speed videos to damage patterns *Exp. Therm. Fluid Sci.* vol 68 pp 359–370

[9] Hidalgo V 2016 Numerical Study on Unsteady Cavitating Flow and Erosion Based on Homogeneous Mixture Assumption (Tsinghua University).

[10] Hidalgo V, Luo X, Escaler X, Ji B, and Aguinaga A 2016 Implicit large eddy simulation of unsteady cloud cavitation around a plane-convex hydrofoil *J. Hydrodyn.* vol 27 pp 815–823

[11] Jian W, Petkovšek M, Houlin L, Širok B and Dular M 2015 Combined numerical and experimental investigation of the cavitation erosion process *J. Fluids Eng.* vol 137 pp 1–9

[12] Hidalgo V, Escaler X, Valencia E, Peng X, Erazo J, Puga D and Luo X 2019 Scale-adaptive simulation of unsteady cavitation around a Naca66 hydrofoil *Appl. Sci.* vol 9 18

[13] Davidson L, 2006 Evaluation of the sst-sas model: channel flow, asymmetric diffuser and axi-symmetric Hill *ECCOMAS* pp 1–20

[14] Hidalgo V, Luo X, Escaler X, Valencia E and Cruz P 2019 Numerical simulation of the cavitation micro-jet velocity and erosion on a plane-convex hydrofoil with semicylindrical obstacle *IOP Conf. Ser. Earth Environ. Sci.* vol 240

[15] Morgut M, Nobile E and Biluš I 2011 Comparison of mass transfer models for the numerical prediction of sheet cavitation around a hydrofoil *Int. J. Multiph. Flow* vol 37 pp 620–626

[16] Zwart P, Gerber A and Belamri T 2004 A two-phase flow model for predicting cavitation dynamics *Int. Conf. Multiph. Flow* p 152

[17] Zhang G, Shi W, Zhou L, and Zhang D 2015 Effect of the Maximum Density Ratio Between Liquid and Vapor on Cavitating Simulation *American Journal of Engineering and Applied Science* vol 8 pp 119-126

[18] Li D, Grekula M and Lindell P 2010 Towards numerical prediction of unsteady sheet cavitation on hydrofoils *J. Hydrodyn* vol 22 pp 741–746

[19] Hidalgo V, Luo X, Yu A and Soto R 2014 Cavitating flow simulation with mesh development using salome open source software *ICHD* p 6

[20] Menter F and Egorov Y 2010 The scale-adaptive - simulation method for unsteady turbulent flow predictions *Flow Turbl. Combust* pp 113-138

[21] Yu A, Tang Q and Zhou D 2019 Cavitation evolution around a NACA0015 hydrofoil with different cavitation models based on level set method *Appl. Sci.* vol 9 p 758