Development and validation of a cavitation erosion model

A. Evangelisti[1, 2], G. Agati[1], D. Borello[1], P. Venturini[1], P. Capobianchi[2]

[1] San Pietro in Vincoli, Department of Mechanical and Aerospace Engineering, Sapienza University of Rome, Via Eudossiana, 18, Rome, 00184

[2] James Weir Fluids Laboratory, Department of Mechanical and Aerospace Engineering, University of Strathclyde, 75 Montrose Street, Glasgow, G1 1XJ

Introduction

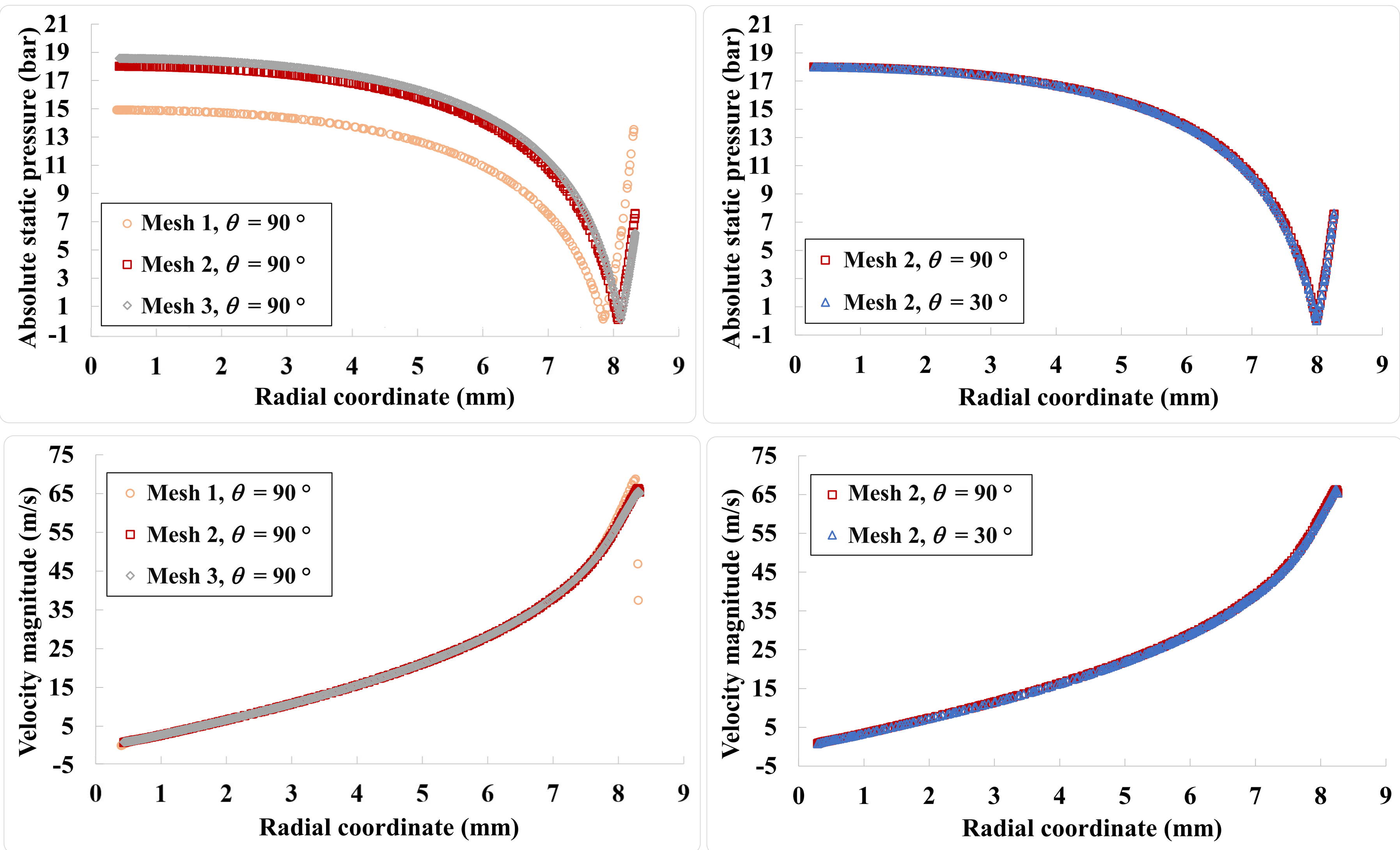
Cavitation erosion is a major issue in the working life of hydraulic turbomachines. Vapour formation and bubbles' implosions can lead to the damage of the solid boundaries, provoking performance losses and fatigue failure over time. In this work, cavitation is predicted combining the Full Cavitation Model of Singhal et al. [1] and the modified k-ω SST turbulence model [2]. Simulations, carried out in Ansys Fluent, reproduce a circular sector of the stainless-steel nozzle used in the experiment of J. P. Franc [3]. Results aim at validating a new, non-case sensitive, approach for erosion occurrence.

Computational domain

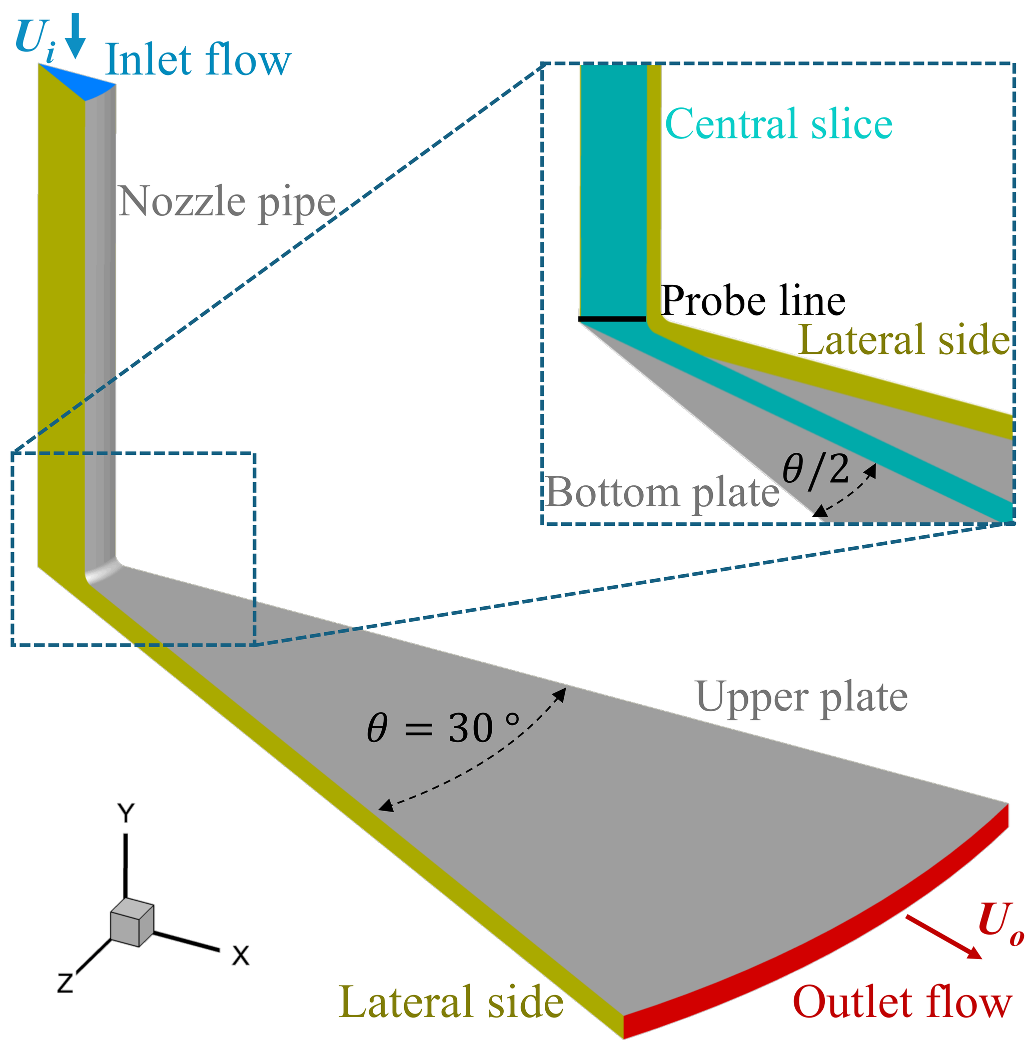
- Steady non-cavitating simulations of the nozzle are performed to evaluate the optimal compromise computational grid.
- 3D meshes differ for overall computational cells number and azimuthal amplitude, θ. Tridimensional grids are obtained extruding on the azimuthal direction ($\hat{\theta}$) the 2D mesh design.

2D case	3D case, θ (°)	Cells elements ($\times 10^6$)
Mesh 1	90	≅ 1.8
Mesh 2	90	≅ 4.5
Mesh 2	30	≅ 1.5
Mesh 3	90	≅ 18.5

- Absolute values of static pressure and velocity magnitude were compared on the central slice, on a probe line crossing the nozzle joint, for an angle equal to half the overall azimuthal amplitude.



- From grid independence analysis, Mesh 2, θ = 30 °, is chosen to predict cavitation erosion.



Turbulence modification and cavitation modelling

- Navier Stokes equations (mass and momentum) are discretized using a U-RANS mathematical approach, adopting the k-ω SST turbulence model.
- Standard U-RANS models overestimate the eddy viscosity when cavitation occurs. To manage this numerical issue the density function of Reboud et al. [2] was implemented to correct the CFD code:

$$f_R(\rho) = \rho_v + \frac{(\rho - \rho_v)^n}{(\rho_l - \rho_v)^{n-1}}, \quad n = 10 \text{ for water} \qquad \mu_{\tau,R} \propto \frac{k}{\omega} f_R(\rho)$$

- Cavitation prediction is realized by using the cavitation model of Singhal et al. [1]. The model equations are proportional to the bubble radius velocity term, computed as a first order approximation of the Rayleigh Plesset equation [4]:

$$\frac{dr_B}{dt} = \sqrt{\frac{2}{3}\frac{p_{v,\tau}(t) - p_\infty(t)}{\rho_l}}$$

- Evaporation and condensation terms, $\dot{m}_v$ and $\dot{m}_l$, respectively, are then enabled according to different pressure conditions:

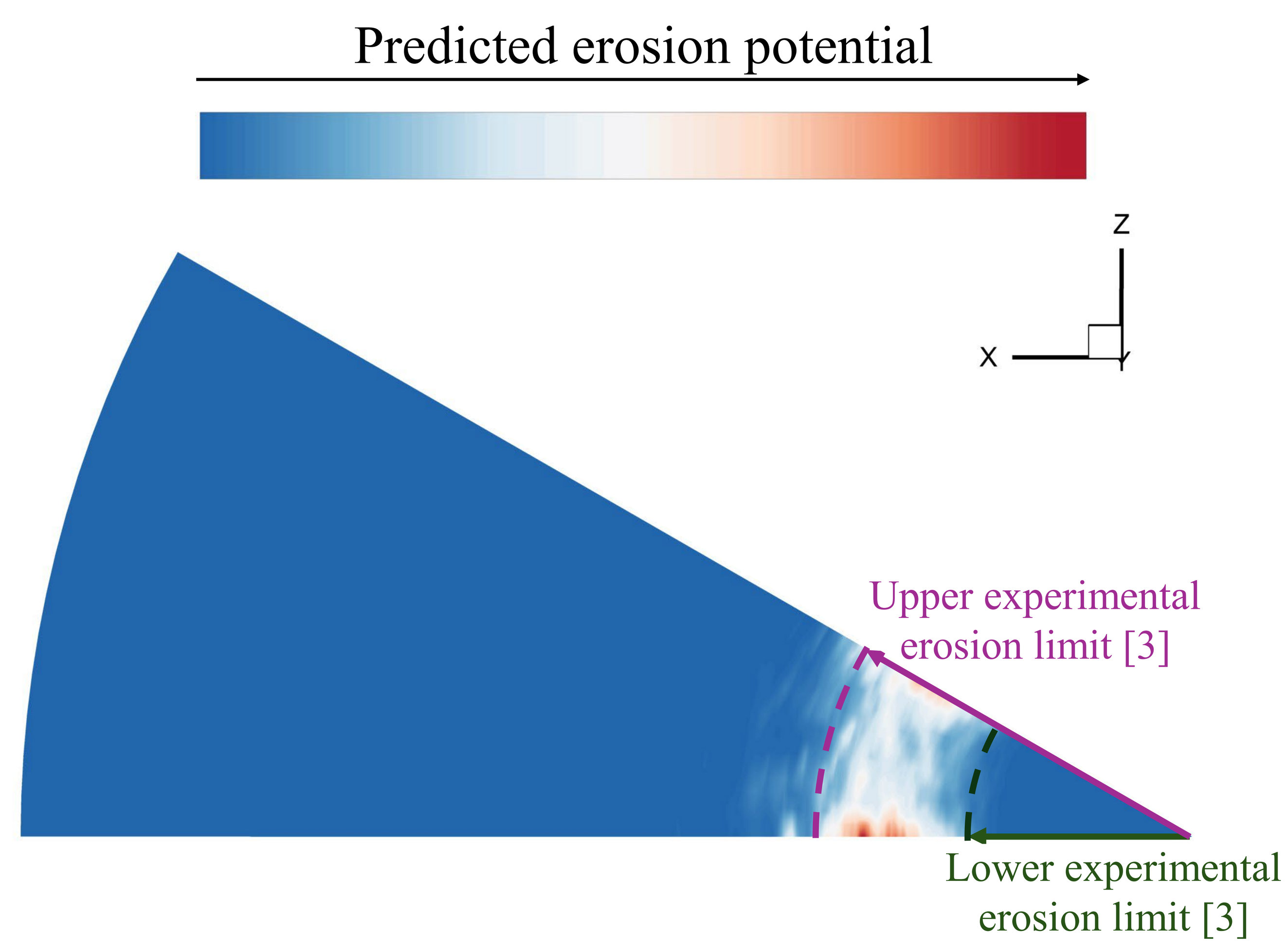
$$\text{if } p \le p_v \Rightarrow \dot{m}_v = C_v \frac{\max\left(1.0;\ \sqrt{k}\right)}{\sigma} \rho_l \rho_v \sqrt{\frac{2}{3}\frac{p_{v,\tau} - p_\infty}{\rho_l}} \left(1 - f_v - \sum_j f_j\right)$$

$$\text{if } p \ge p_v \Rightarrow \dot{m}_l = C_l \frac{\max\left(1.0;\ \sqrt{k}\right)}{\sigma} \rho_l^2 \sqrt{\frac{2}{3}\frac{p_\infty - p_{v,\tau}}{\rho_l}} f_v$$

- Numerical simulations are performed adopting the CFD software Ansys Fluent.

Numerical results vs experimental data

- From cavitating solutions, erosion is modelled implementing a new, non-case sensitive methodology.
- The erosion parameters, as for the cavitation model, are based on the bubble dynamics treatment. Bottom plate damage is caused by jets originated by cavitation clouds implosion.
- Results show the erosion potential due to jets – surface impacts over time. Location of the numerical predicted eroded area is displayed against experimental data produced by J. P. Franc [3]:



Conclusions and future works

- From the grid independence study on domains of azimuthal width equal to 90°, Mesh 2 is chosen as the best compromise computational grid, when static pressure and velocity magnitude curves are tested on the probe line.
- Reducing the computational domain azimuthal amplitude from θ = 90° to θ = 30° no relevant differences are detected for Mesh 2. This last condition identifies the chosen grid used to predict cavitation erosion.
- Cavitation erosion numerical results show good agreement in the reproduction of the experimental erosion mid-line.
- New steady simulations of finer planar meshes, extruded for θ = 30°, are being tested.
- New unsteady simulations will be dedicated to assess the effect of the time step sensitivity on the cavitation erosion formation.
- The influence of different cavitation models in the prediction of the area affected by erosion will be assessed.

References

[1] A. K. Singhal, M. M. Athavale, H. Li, and Y. Jiang, "Mathematical Basis and Validation of the Full Cavitation Model" , J Fluids Eng, vol. 124, No. 3, 2002, pp. 617-624.

[2] J. L. Reboud, B. Stutz, and O. Coutier, "Two-phase flow structure of cavitation: experiment and modelling of unsteady effects", in Third International Symposium on Cavitation, Grenoble, France, 1998, pp. 1–7.

[3] J.-P. Franc, "Incubation Time and Cavitation Erosion Rate of Work-Hardening Materials," J Fluids Eng, vol. 131, No. 2, 2009, pp. 021303 1-14.

[4] F. R. Young, Cavitation. England: Imperial Collage Press, 1999