Sixth HRLM Symposium, 26-28 September 2016, Strasbourg University, France

Session: Comparative studies of hybrid RANS-LES and/or other turbulence-resolving simulations

# Numerical study of 3D turbulent cavitating flows

**E. Goncalves[a], J. Decaix[b], B. Charrière[a]**

**Institut Pprime, ENSMA, CNRS UPR 3346, Poitiers, France**

**University of Applied Sciences, Sion, Switzerland**

Cavitating flows are characterized by important variations of the local speed of sound, non equilibrium thermodynamic states and small-scale turbulence interactions. The interplay between turbulence and cavitation regarding the unsteadiness and structure of the flow is complex and not well understood. This constitutes a determinant point to accurately simulate the dynamic of sheet cavities.

Sheet cavitations that appear on solid bodies are characterized by a closure region which always fluctuates, with the presence of a re-entrant jet. Observations on hydrofoils with high-speed motion pictures put in evidence the three-dimensional structures of the phenomena. To distinguish between various directions of the re-entrant flow, the term side-entrant jet was introduced. This term refers to the part of the jet that has a strong spanwise velocity component directed into the cavity originating from the sides. The shape of the closure region of the cavity sheet governs the direction of the re-entrant and side-entrant jets [1].

Numerical simulations are carried out using two different softs solving the one-fluid hybrid RANS/LES system: the in-house density-based code CAVIFLOW and the open source code OpenFOAM. Both codes use a cavitation model based on a transport equation for the void ratio. The first one is a compressible code based on a sinusoidal equation of state for the mixture. The mass transfer term appearing explicitly in the source term is closed assuming its proportionnality to the divergence of the velocity [2]. The tubulence modelling is based on the Smith k-l transport-equation model and its Scale-Adaptive (SAS) version [3].

For the second soft, cavitation is modelled using the model proposed by Kunz [4]. The mass and momentum conservation equations are solved in a segregated manner using a velocity pressure algorithm. Turbulence is taken into account using the k-ω SST SAS model of Menter [5].

Numerical results are given for a Venturi geometry and comparisons are made with experimental data and previous computations using a barotropic model [6]. A qualitative view of the cavitation sheet is illustrated plotting the density gradients near the Venturi throat. Simulations provide a cavitation sheet with a U-shape form (Fig 1) characterized by a stable part from the throat of the Venturi to x = 0.25 meter, then followed by an unsteady region were the re-entrant jet is well captured in comparison with the experimental data available only in the mid span section. Other comparisons show a good agreement concerning the void fraction and the longitudinal velocity.

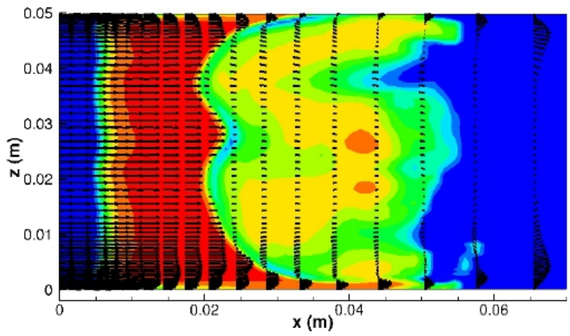


**Fig. 1** Gradient of density at the throat.

## References

[1] Foeth, E.J., van Doorne, C.W., van Terwisga, T., "Time-resolved PIV and flow visualization of 3D sheet cavitation", *Exp. In Fluids*, vol. 40, p503-513, 2006.

[2] Goncalves, E., Charriere, B. "Modelling for isothermal cavitation with a four-equation model", *Int. J. of Multiphase Flow*, vol. 59, p54-72, 2014.

[3] Decaix, J., Goncalves, E, "Time-dependent simulation of cavitating flow with k-l turbulence model" *Int. J. Numer. Meth. Fluids*, vol 68, p1053-1072, 2012.

[4] Kunz, R., Boger, D., Stinebring, T., Lindau, J., Gibeling, H., Venkateswaran, S., "A preconditioned Navier–Stokes method for two-phase flows with application to cavitation prediction", *Comput. & Fluids*, Vol. 29 (8), p 849–875, 2000.

[4] Menter, F.R., Egorov, Y. "The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 1: Theory and Model Description", *Flow, Turbulence & Combustion*, vol. 85, p113-138, 2010.

[5] Decaix, J., Goncalves, E, "Investigation of three-dimensional effects on a cavitating Venturi flow" *Int. J.of Heat and Fluid Flow*, vol 44, p576-595, 2013.