Numerical investigation of sheet cavitation over a 3-D venturi configuration

Camille Gouin[1]*, Carlos Junqueira-Junior[1], Eric Goncalves Da Silva[2] and Jean-Christophe Robinet[1]

[1]Dynfluid laboratory, ENSAM, 151 Boulevard de l'Hôpital, 75013 Paris, France

[2]Institut Pprime, UPR 3346 CNRS, ISAE-ENSMA, 11 bd Marie et Pierre Curie, 86960 Chasseneuil , France

Abstract: Sheet cavitation appears in many hydraulic applications and can lead to technical issues. Numerical simulation is a pertinent way to study the phenomenon. A numerical tool based on 1-fluid compressible RANS equations with a cavitation model is used to compute a flow within a 3-D venturi geometry with a 4° divergent angle. In the present work, a detailed study of this cavitating flow, which presents a quasi-stable vapour pocket, is carried out using tools such as Power Spectral Densities or Spectral Proper Orthogonal Decompositions. An oblique oscillation of the cavity is then identified and discussed.

Keywords: sheet cavitation; four-equation model; 3-D venturi; SPOD; oblique mode.

1. Introduction

Cavitation is the formation of vapour cavities in a liquid due to a pressure drop. The phenomenon occurs in hydraulic systems or turbomachinery and can, eventually, cause structural damage, noise and degrade the performance of the apparatus. Such effects drive the study of the different types of cavitation, and their generating states are relevant concerns. The present work focuses on the 3-D simulation of cavitation pockets within a venturi geometry with a 4° divergent angle.

This flow configuration is characterized by the development of a quasi-stable cavity which presents only small shedding at its closure region with a relatively stable cavity length. Meanwhile, an upstream flow, called re-entrant jet, appears around the cavity closure. This type of cavity has been studied, experimentally and numerically, to describe the physical mechanism, the internal structure of cavities and the turbulence-cavitation interaction [1,2]. In the present work, a detailed study of the cavitating flow inside the selected geometry is carried out. First, computations are validated with experimental data. Then, the global behaviour of the flow is investigated using Power Spectral Densities (PSD) and Spectral Proper Orthogonal Decomposition (SPOD). A spanwise oscillation mechanism is highlighted and discussed in the last part.

2. Materials and Methods

The sheet cavitation is modelled using a 1-fluid compressible RANS system of equations combined with a transport equation of the void ratio α [2]. The phase change is triggered below a pressure limit and led by a sinusoidal equation of state to smooth the discontinuities of density along the interface. The turbulence part is solved using the Smith's k-l model [3]. The Reboud limiter [4] and the QCR correction [5] are added to control the amount of turbulent viscosity respectively in two-phase region and around corners. A two-layer wall function is applied on the wall. The inlet and outlet boundary conditions are set based on characteristic relations of Euler equations. Third-order central scheme with numerical dissipation, based on Jameson-Schmidt-Turkel scheme [6], is selected for this stiff problem. Moreover, due to the range of Mach number into the flow, a low Mach preconditioning method is developed using the strategy of

* Corresponding Author: Camille Gouin, camille.gouin@ensam.eu

Turkel [7] and applied on the dissipation term. A third-order Strong Stability Preserving Runge Kutta time integration [8] is used for the calculation of unsteady flow configurations.

3. Results

3.1. Numerical validation

Experimental data, extracted by Barre et al. (2009) [1], from the same venturi configuration are used to validate the numerical tool. This experiment provides measures of temporal means of velocity, void ratio, and wall pressure profiles at stations located in the midspan of the venturi. Probes positioning is calibrated to capture data adjacent to the vapour pocket. Numerical data of the 3-D simulation are in a good match with the experiment extractions. Moreover, a mean cavity length of L_c= 78.8 mm is calculated, in numerical results, with an α contour of 0.05. This dimension is consistent with a size between 70 and 85 mm observed in the experiment.

3.2. Global behaviour

A time-averaged analysis of numerical results is carried out. A symmetrical attached vapour pocket with a U-shape is developed downstream the venturi throat. A lower amount of void ratio suggests vapour release or pocket oscillation around the mid-width of the venturi. Then, the observation of the streamwise velocity component underlines the presence of a recirculating bubble around the vapour cavity closure corresponding to the re-entrant jet development. The jet topology is symmetric and is not present close to sidewalls.

The observation of the time dependent behaviour of the vapour pocket suggests the occurrence of a spanwise fluctuation of the vapour pocket. PSD analysis are performed along the longitudinal and the spanwise axis to study the flow dynamics. Figure 1 highlights PSD mappings for the void ratio and the spanwise velocity component along the longitudinal and spanwise axis. For the void ratio, a main Strouhal number of 0.075 is captured around the cavity closure and close to sidewalls with the emergence of firsts harmonics. For the spanwise velocity component, the same Strouhal number is observed but localised at the mid-width of the venturi and propagated downstream. It is worth to remark that no particular dynamics are detected by PSD mappings inside the attached vapour pocket. Then, a correlation study is investigated by extracting the flow variables over time close to both sidewalls to specify the cavity behaviour. The extractions are in opposition of phase, which can lead to a conclusion that the vapour pocket motion is assimilated to a periodic oscillation from one sidewall to another. Similarly, a dominant Strouhal number of 0.075 is extracted around the cavity closure and downstream from PSD mappings over the streamwise velocity component. The negative values of this velocity component, at the vapour pocket closure, mainly depicts the re-entrant jet position. Furthermore, by studying the time dependent behaviour of the vapour pocket and the re-entrant jet, it is remarked that both oscillate at the same frequency in opposition of phase. When the re-entrant jet is mostly located on one side, the vapour pocket is at the other one.

* Corresponding Author: Camille Gouin, camille.gouin@ensam.eu

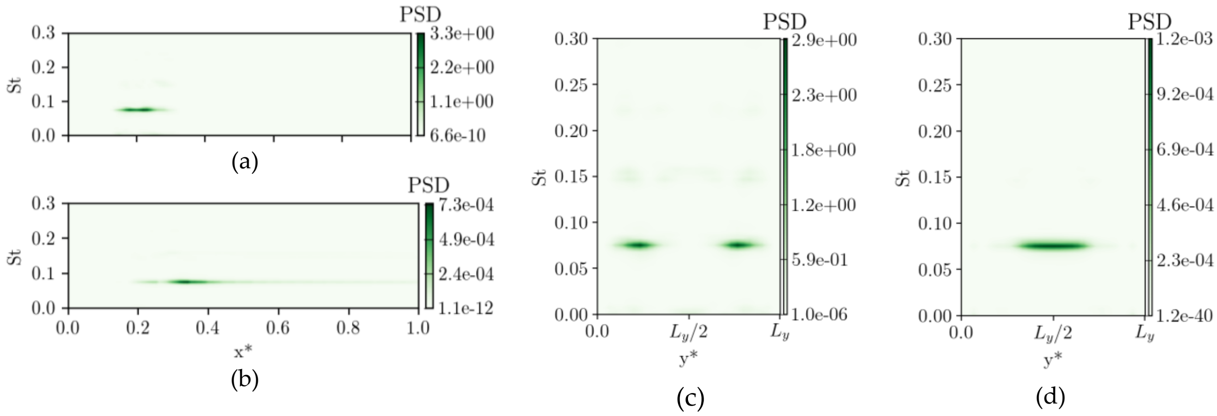

Figure 1. PSD mappings: (**a**) along longitudinal axis at the quarter-width for the void ratio; (**b**) along longitudinal axis at the quarter-width for spanwise velocity component; **(c)** along the spanwise axis around the cavity closure (x*=0.2) for the void ratio; **(d)** along the spanwise axis around the cavity closure (x*=0.2) for the spanwise velocity component**.**

3.3. Modal decomposition

Early results highlighted a dominant flow mechanism at the Strouhal number of 0.075 probably linked to a vapour pocket oscillation. Further investigations, such as a SPOD analysis, are performed from computational results to corroborate previous observations and identify spatiotemporal mechanisms. The choice of the SPOD is motivated by the extraction of spatiotemporal modes which is the most coherent method to study unsteady flow as presented by Towne et al. [9]. The SPOD methodology employed is based on the work of Schmidt and Colonius [10].

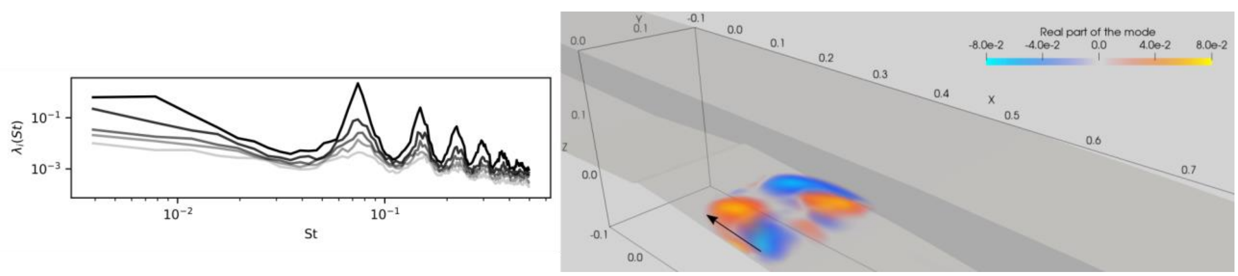

Figure 2. SPOD spectrum representing energy gain over frequency (left) and the dominant SPOD mode for the density(right).

The eigenspectrum of a SPOD computed with 900 snapshots, spaced by 0.0023 s, is shown in Figure 2. A dominant mode is captured at the Strouhal number of 0.075 as well as its harmonics. Hence, the dominant flow dynamic mechanism of the present case seems to be characterized by this Strouhal number. The dominant mode corresponds to a spanwise oscillation with an upstream motion around the cavity closure for the density. The same behaviour is observed for the velocity components, though, the oscillation is also propagated downstream the cavity closure.

3.4. Discussion

It has to be noticed that a similar behaviour of the cavitation pocket has been detected in computations with another turbulence model: the Spalart Allmaras's one. Moreover, oblique-shape behaviours of the vapour pocket have already been observed in the experiment of Timoshevskiy et al. [11] and could be

* Corresponding Author: Camille Gouin, camille.gouin@ensam.eu

suggested in the experiment of Dular et al. [12]. The spanwise oscillation is mainly described by the dominant mode dynamics for the spanwise velocity. This effect highlights an alternation of positive and negative values around the cavity closure. The study of this mode dynamics for velocity components underlines that the oscillation pattern is simultaneously propagated upstream, by the re-entrant jet, and downstream the flow. The downstream flow is thus highly influenced by the vapour pocket and the re-entry jet dynamics while the oscillation seems to be self-sustained by the upstream flow.

Fluctuations observed just downstream the vapour pocket in the modal decomposition analysis are reflected as an oscillation of magnitude for the vertical and the longitudinal velocity components. When the vapour pocket is longer near a sidewall, the magnitude of these two components increased on the same side and decreased on the other. This information involves that the longer the cavity is, the more accelerated the downward and the frontward speeds are. When the vapour pocket moves to the other side, the velocity effects are reversed. Hence, the vapour pocket interferes with the flow can be seen as a dynamic obstacle.

Acknowledgments: This research was supported by the French National Research Agency ANR (project 18-CE46-009). Computations have been performed on the HPC resources of GENCI (Grand Equipement National de Calcul Intensif) under allocation A0072A06362. The authors thank Jean-Christophe Loiseau for providing the SPOD code.

References

1. Barre S., Rolland J., Boitel G., Goncalves E., Patella R.F., Experiments and modeling of cavitating flows in venturi: attached sheet cavitation, *European Journal of Mechanics B/Fluids* 2009, Vol.28, pp 444-464.
2. Charrière B., Goncalves E., Numerical investigation of periodic cavitation shedding in a venturi. *International Journal of Heat and Fluid Flow* 2017, Vol.64, pp 41–54.
3. Smith, B., 1994. A near wall model for the k − l two equation turbulence model, in AIAA 94–2386, 25th Fluid Dynamics Conference – Colorado Springs, Colorado, 1994.
4. Reboud J.L., Stutz B., Coutier O., 1998. Two-phase flow structure of cavitation: experiment and modelling of unsteady effects, in: 3 rd International Symposium on Cavitation CAV1998, Grenoble, France.
5. Spalart P.R., Strategies for turbulence modelling and simulations, *International Journal of Heat and Fluid Flow* 2000, Vol.21, pp 252–263.
6. Jameson A., Schmidt W., Turkel E., Numerical solution of the Euler equations by finite volume methods using Runge-Kutta time stepping schemes, in: AIAA Paper 81–1259, 1981.
7. Turkel E., Preconditioned methods for solving the incompressible and low speed compressible equations. *Journal of Computational Physics* 1987, Vol.72, pp 277–298.
8. Spiteri R.J., Ruuth S.J., A new class of optimal high-order strong-stability-preserving time discretization methods. *SIAM Journal on Numerical Analysis* 2002, Vol.40, pp 469–491.
9. Towne A., Schmidt O.T., Colonius T., Spectral proper orthogonal decomposition and its relationship to dynamic mode decomposition and resolvent analysis. *Journal of Fluid Mechanics* 2018, Vol.847, pp 821–867.
10. Schmidt O.T., Colonius T., Guide to spectral proper orthogonal decomposition. *AIAA Journal* 2020, Vol. 58, No. 3, pp 1023-1033.
11. Timoshevskiy M.V., Churkin S.A., Kravtsova A.Y., Pervunin K.S., Markovich D.M., Hanjalic K., Cavitating flow around a scaled-down model of guide vanes of a high-pressure turbine. *International Journal of Multiphase Flow* 2016, Vol.78, pp 75–87.
12. Dular M., Khlifa I., Fuzier S., Maiga M.A., Coutier-Delgosha O., Scale effect on unsteady cloud cavitation. *Experiments in Fluids* 2012, Vol.53, pp 1233–1250.

* Corresponding Author: Camille Gouin, camille.gouin@ensam.eu