

OpenFOAM investigations of cavitation in a flushed water-jet inlet

A Gattorochieri¹ and R Bensow²

¹ Termomeccanica Pompe, La Spezia, Italy

² Department of Shipping and Marine Technology, Chalmers University of Technology, Gothenburg, Sweden

E-mail: a.gattorochieri@gmail.com

Abstract. The cavitation on the lip of a flushed water-jet inlet has been simulated with a transient RANS model and the results has been validated against experiments. The $k-\omega$ SST turbulence model has been adopted together with the cavitation correction proposed by Reboud. The defined setup shows promising results and the vortex shedding has been qualitatively predicted. Moreover, the importance of the sufficient spatial resolution to capture the cavity closure and its extension has been studied and proved to be crucial.

1. Introduction

In the 20th century and even more in the last two decades, water-jets have been widely used for high speed crafts propulsion. The typical speed range determines relatively low free stream cavitation numbers which may easily lead to cavitation problems in different parts of the system. Beside the pump cavitation, which has been deeply investigated [1] [2], the cavitation in the inlet duct has been pointed out by various authors [3], [4]. The water-jet inlet is fundamental for the overall propulsion performance and for the pump operative life, therefore the study of the cavitation that may occur in different parts of the duct, namely the ramp, the bend and the inlet lip, is of particular interest. Previous authors pointed out that, for a given geometry, the cavitation at the inlet lip is considerably influenced by the waterjet inlet velocity ratio [4], [5] and by the hull boundary layer thickness [5]. Moreover, Bulten [4] highlights that cavitation tunnel measurements of the cavitation inception may be considerably influenced by tunnel walls. Consequently, reliable numerical simulations for the prediction of the cavitation on the inlet lip would be a valuable tool in the design process.

The here reported study is part of an intense simulation campaign aimed at the understanding of the capability of the open source CFD code OpenFOAM to predict the flow features in a flushed water-jet inlet through the comparison of the numerical results with a well-documented experimental campaign [5]. In the first part of this research, good results in simulation of non-cavitating conditions have been obtained, considering the minor uncertainties on the prediction of the influence of boundary layer thickness on the flow separation occurring along the ramp at the highest inlet velocity ratio (IVR) that have been pointed out [6]. The following presents the preliminary results obtained with a setup modified in order to predict the cavitation at the inlet lip documented in the same experimental campaign.

2. Experimental setup

The experiments have been carried out in the AMC Tom Fink Cavitation Tunnel, a closed-circuit variable-pressure water tunnel, shown in Figure 1. Detailed information on the tunnel and water-jet inlet



test loop are given in Brandner and Walker [7]. The experiments have been carried out with the natural test section boundary layer and with a thickened boundary layer of 20 and 50 mm respectively, maintaining a turbulence and velocity distribution similar to those generated on a flat plate. The model has been instrumented to measure the principal flow features: boundary layer development, ramp pressure distribution, lip incidence and the pressure and velocity distributions at the pump face. Details of the instrumentation, not reported here for the sake of shortness, are available in the reference paper. The cavitation in this area is strongly influenced by the value of the IVR which determines the incidence on the lip and consequently the low pressure zone. For IVR smaller than that corresponding to the ideal incidence, the stagnation point moves towards the hull bottom, while for higher value it moves inside the duct, thus the IVR determines if cavitation may occur above or below the lip. The authors pointed out that also the boundary layer thickness influences the cavitation inception, as shown in Figure 2, where the inception cavitation number is reported as a function of the IVR and boundary layer thickness. The cavitation number has been defined as:

$$\sigma_{\infty} = \frac{p_{\infty L} - p_v}{\frac{1}{2} \rho U^2}$$

where U is the free stream velocity when the cavitation occurs below the lip and the mean exit velocity in the other condition; $p_{\infty L}$ is the free stream static pressure at lip elevation and p_v is the vapour pressure. Furthermore, the reference article reports four picture of the cavitation occurring on the outside of the lip at different IVR and Reynolds number in the thickened boundary layer configuration.

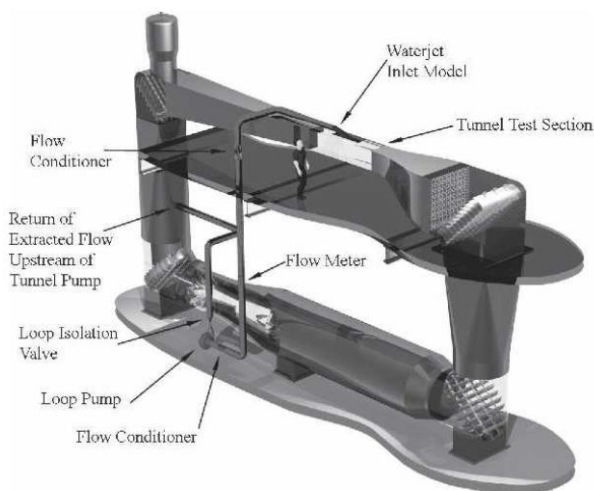


Figure 1. Water-jet test loop.

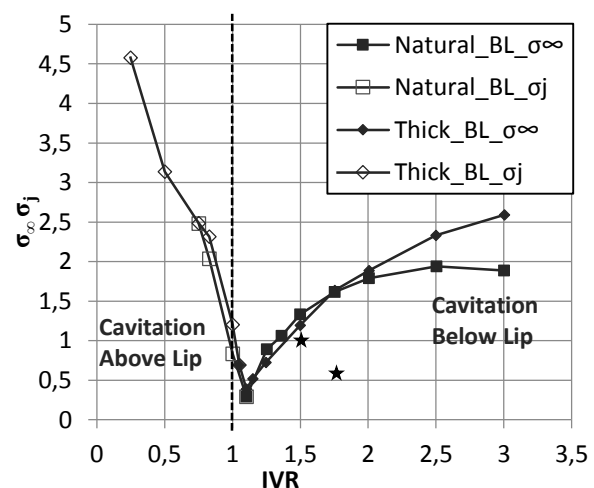


Figure 2. Lip cavitation inception and analysed operative condition.

3. Numerical model

Consistently with the first part of this research, [6], a computational domain coincident with the test section has been chosen for the simulations and it has been discretised using hexahedral elements in the sweepable parts and tetrahedral in the remaining. The prism layer has been defined in order to satisfy the requirements of the low-Reynolds wall treatment. The grid obtained from the mesh dependency study carried out for the wet-flow simulations has been refined several times in correspondence of the of the lip, varying the average edge sizes from about 3 mm used in the original grid to 0.6 mm used for the results presented in the following. This modification increases the total number of elements from about 1.3 to 5 millions. A mesh dependency study is currently being performed with a further level of refinement. No more details about the grids are reported here due to the short paper format.

The two phases coexisting in the cavitation have been modelled as a mixture of two incompressible fluids using the transient solver *interPhaseChangeFoam*, introducing a transport equation for the vapour volume fraction and the widely used Schnerr-Sauer mass transfer model [8]. The effects of the

turbulence closures on the simulations of unsteady sheet cavitation have been pointed out in previous work [9] [10]. These publications highlight that the implicit LES model is capable in predicting the unsteady nature of this type of cavitating flows. However, in order to limit the computational time and to be consistent with the previous simulations of the waterjet inlet duct, the RANS approach has been adopted. The $k-\omega$ SST turbulence model available in OpenFOAM has been modified in order to include the Reboud correction [11]. This ad hoc correction reduces the turbulent viscosity in the mixture which is responsible for the inability of the RANS models to produce the unsteady cavitation behaviour, as it has been stressed in several publications [11] [12]. The time step has been fixed in order to ensure the maximum Courant number equal one, necessary due to the explicit implementation of the solver in the OpenFOAM version used.

4. Results

As mentioned before, the reference paper documents the cavitation under the inlet lip in four different operative conditions. Two of these, identified by the “star” marker in Figure 2, have been chosen to verify if the numerical simulations are able to correctly predict the cavitation behaviour.

In the first condition, characterized by $Rn = 1 * 10^6$, $IVR=1.5$ and cavitation number $\sigma_\infty = 1$, a sheet cavity with highly unstable closure and cavitating vortex filaments in the turbulent shear layer is documented in the reference paper. Figure 3 shows the picture taken in the model test in comparison with two isosurfaces computed from the numerical solution with different threshold of vapour volume fraction. The shape and extent of the sheet cavitation are well captured in proximity of the leading edge and in the centre of the duct, while its closure in the external part is visible only with the higher iso-value of the vapour volume fraction. The cloudy structure shed from the previous sheet and the cavitating vortex filaments visible in the experiments have not been observed in the numerical solution.

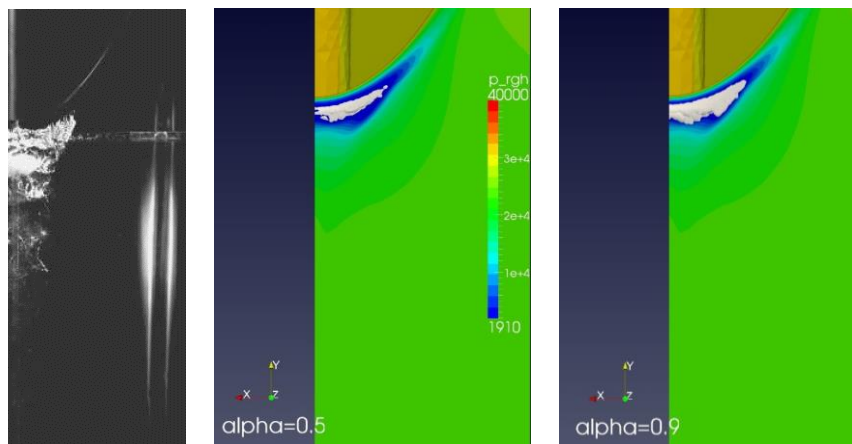


Figure 3. External lip cavitation detected in the experiments and visualized in the numerical simulation with $\sigma_\infty = 1$ and $IVR = 1.5$.

In the second simulated condition, characterized by the same Reynolds number, $IVR=1.75$ and cavitation number $\sigma_\infty = 0.6$, the cavity grows in base width and length and a more coherent closure mechanism cause the shedding of horseshoe vortices. The comparison between the model test and the numerical results, reported in Figure 4, confirms what was been pointed out for the first operative condition: the initial part of the cavity is visible with both threshold values while, the downstream cloudy structure is not predicted by the numerical solution; however, alpha equal to 0.9 highlights the shedding of cavitating vortices. It has been shown through experimental measurements that the vapour volume fraction could well be around 10 percent in relatively large regions in this type of cloudy cavitation [13]. Nevertheless, the difference between the iso-surface computed with the two threshold value indicate that further grid refinements are still necessary. It should be noted that the discrepancy between the numerical results and experimental data in prediction of the cavity size is also dependent on the accuracy

of capturing the separated vortex. Having appropriate mesh resolution is vital for preserving the vortex core strength while transporting it downstream. Lack of spatial resolution will lead to smearing the pressure gradients especially at the core of the separated vortex, and therefore over prediction of the pressure in the vortex region. Consequently over prediction of the pressure will lead to over prediction of vapour to liquid mass transfer.

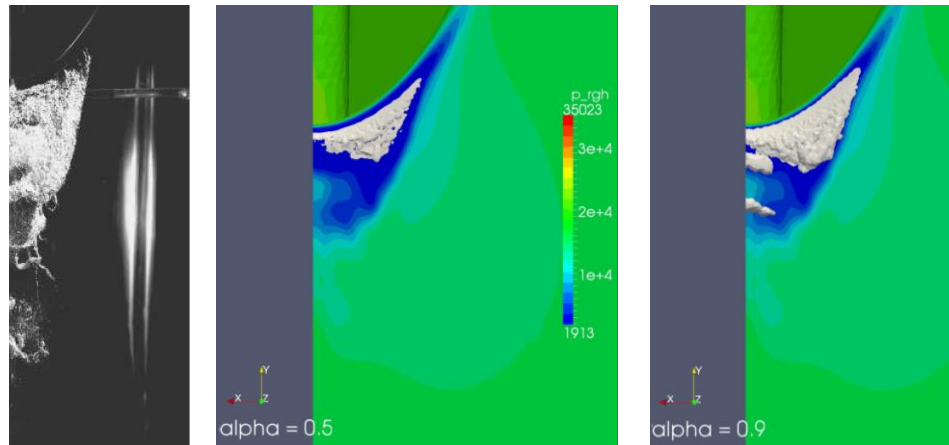


Figure 4. External lip cavitation detected in the experiments and visualized in the numerical simulation with $\sigma_\infty = 0.6$ and $IVR = 1.75$.

5. Conclusions

The defined setup provides promising results, the vortex shedding has been qualitatively predicted and the importance of the spatial resolution to capture the cavity closure has been pointed out. Further analysis are still required to refine the simulation setup; for this purpose more information about the experiments would be valuable for the validation of the numerical model.

References

- [1] Dang J, Liu R and Pouw C 2013 Waterjet System *Performance and Cavitation Test Procedures Third Int. Symp. on Marine Propulsors smp'13* Tasmania Australia
- [2] Gulich J F 2010. *Centrifugal Pumps* Springer
- [3] Faltinsen O M 2005 *Hydrodynamics of High-Speed Marine Vehicles* 70-73
- [4] Bulten N W 2006 *Numerical Analysis of a Waterjet Propulsion System* PhD Thesis 101-105
- [5] Brandner P A and Walker G J 2007 An Experimental Investigation Into the Performance of a Flush Water-Jet Inlet *J. of ship research*
- [6] Gattorochieri A and Cravero C 2014 OpenFOAM investigations of a flushed water-jet inlet performance *proc. NuTTS'14* Marstrand Sweden
- [7] Brandner P A and Walker G J 2001 A waterjet test loop for the Tom Fink Cavitation Tunnel *proc. of Waterjet Propulsion Conference III, RINA*, February 20 Gothenburg Sweden
- [8] Schnerr G H and Sauer J 2001. Physical and Numerical Modeling of Unsteady Cavitation Dynamics. *Proc. 4th Int. Conf. on Multiphase Flow* New Orleans U.S.A.
- [9] Bensow R 2011. Simulation of the unsteady cavitation on the Delft Twist11 foil using RANS, DES and LES. Sec. Int. Symp. on Marine Propulsors. Hamburg Germany
- [10] Asnaghi A, Feymark A and Bensow R 2013 Effect of Turbulence Closure on the Simulation of the Cavitating Flow on the Delft Twist11 Foil. *NuTTS13*
- [11] Reboud J L, Stutz B and Coutier O 1998 Two-Phase Flow Structure of Cavitation: Experiment and Modeling of Unsteady Effects. *3rd Int. Symp. on cavitation. Grenoble France*
- [12] Coutier-Delgosha O et al 2003 Evaluation of the Turbulence Model Influence on the Numerical Simulations of Unsteady Cavitation
- [13] Stutz B and Reboud J L 1997 Two-phase Flow Structure of Sheet Cavitation *Phys. Fluids* 9 (12)