**PAPER • OPEN ACCESS**

# Identification of the wave speed and the second viscosity in cavitating flow with 2D RANS computations - Part II

To cite this article: S Alligné *et al* 2015 *J. Phys.: Conf. Ser.* **656** 012057

View the article online for updates and enhancements.

# Identification of the wave speed and the second viscosity in cavitating flow with 2D RANS computations – Part II

**S Alligné[1,4], J Decaix[2], C Nicolet[1], F Avellan[3] and C Münch[2]**

[1] Power Vision Engineering Sàrl, Chemin des Champs Courbes 1, 1024 Ecublens, Switzerland
[2] HES SO Valais-Wallis, Route du Rawyl 47, 1950 Sion 2, Switzerland
[3] EPFL Laboratory for Hydraulic Machines, Avenue de Cour 33 bis, 1007 Lausanne, Switzerland

E-mail: sebastien.alligne@powervision-eng.ch

**Abstract.** The 1D modelling of cavitation vortex rope dynamics in Francis turbine draft tube is decisive for prediction of pressure fluctuations in the system. However, models are defined with parameters which values must be quantified either experimentally or numerically. In this paper a methodology based on CFD simulations is setup to identify these parameters by exciting the flow through outlet boundary condition. A simplified test case is considered to assess if 1D cavitation model parameters can be identified from CFD simulations. It is shown that a low wave speed and a second viscosity due to the cavitating flow can be identified.

## 1. Introduction

Francis turbines develop a cavitation vortex rope at the runner outlet which induces pressure fluctuations propagating in the whole hydraulic system. Interaction between this excitation source and the system may lead to resonance phenomena. To investigate such problematic, a one-dimensional (1D) model of the system is setup including a model of the cavitation volume dynamics which is decisive to predict potential interactions. In the framework of the European FP7 research project Hyperbole, experimental investigations [1] identified the wave speed and the second viscosity parameters of the cavitation model. This paper is aiming to identify them numerically from two phase flow CFD simulations. The developed methodology is applied to a simplified test case: a venturi geometry [2].

## 2. Methodology for identification of cavitation dynamics parameters

The case study is a venturi geometry [2] featuring a divergent angle of 4 degrees downstream the throat. Due to the increase of velocity in the throat region and the downstream divergent, a cavitation sheet develops for low cavitation numbers. A CFD model [3] and a one-dimensional (1D) compressible model of the venturi geometry are setup. The 1D model is based on continuity and momentum equations including the convective terms and the divergent geometry [4]. In the throat region where cavitation develops a 1D cavitation model is used which parameters are the following:

- the local wave speed $a$ defined implicitly by the cavitation compliance $C_c = \left( gAL \right) / a^2$ ;

---

- the second viscosity $\mu''$ induced by the phase change during cavitation volume fluctuations.

In the cavitation free region, the wave speed value is set to 1000 m.s$^{-1}$. The evolution of the dimensionless static pressure head $h^* = h/\left(C_{in}^2/(2g)\right)$ along the venturi is in agreement with the CFD model. To achieve this matching between the two models, the Darcy-Weisbach value is set to $\lambda = 0.0085$. In figure 1, the methodology to identify the two cavitation parameters $a$ and $\mu''$ is presented. The system response to a fluctuating outlet static pressure of the CFD model and the 1D model are compared. The set of parameters of the 1D model is optimized to get the best fitting between the two system responses.
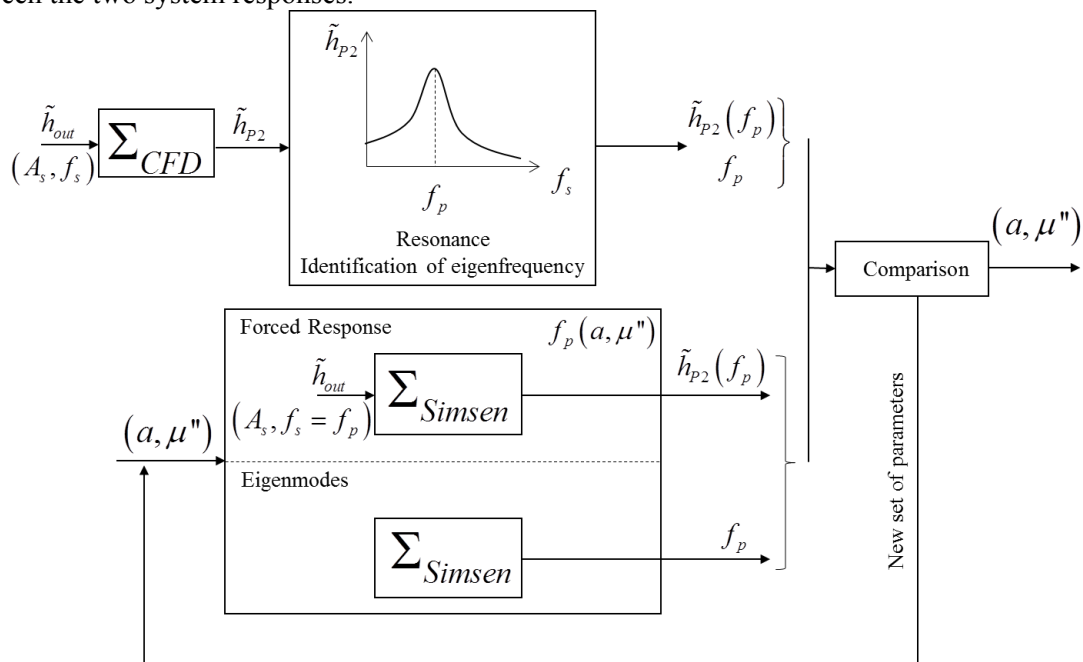


**Figure 1.** Methodology for identification of 1D cavitation model parameters.

On the one hand, the CFD model is excited in a given range of frequency to find a resonance of the cavitation volume fluctuations. The resonance found, the amplitude of pressure fluctuations in Plane2 (see the plane definition in [3]) $\tilde{h}_{P2}$ and the related resonance frequency $f_p$ are the two objectives for the optimization process. On the other hand with the 1D SIMSEN model, the set of cavitation parameters is found by minimizing the error for the two objectives. For the amplitude of pressure fluctuations in Plane2 a forced response analysis is performed in the frequency domain at the frequency $f_p$ and for the resonance frequency, the eigenfrequency of the 1D model is computed.

## 3. Results and discussions

The CFD model features a resonance frequency value of $f_p = 2$ Hz where maximum of pressure and cavitation volume fluctuations are experienced [3]. Time history of dimensionless pressure fluctuations in Plane2 and in Plane4 are represented and compared to the excitation outlet pressure set to a frequency value of $f_s = 6$ Hz in figure 2 and to the resonance frequency $f_s = 2$ Hz in figure 3.

For out of resonance conditions, the amplitude of pressure fluctuations decreases from the outlet excitation source to the cavitation sheet which behaves like a pressure node for the system. However, at resonance conditions the amplitude near the cavitation sheet is higher than the outlet excitation source. Moreover, pressure fluctuations do not feature sinusoidal shape anymore. These typical shapes can be observed in Francis turbine draft tubes at resonance conditions due to system non-linearities.
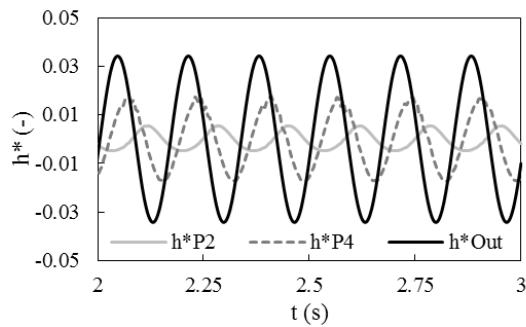
**Figure 2.** Pressure fluctuations for out of resonance conditions ( $f_s = 6Hz$ ).
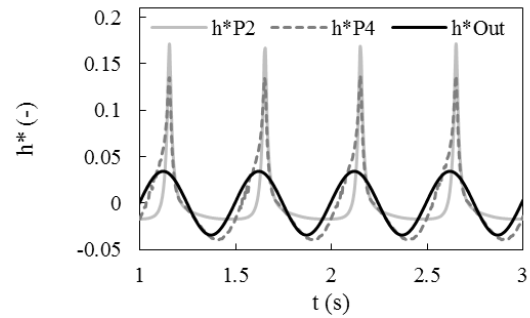


**Figure 3.** Pressure fluctuations for resonance conditions ( $f_s = 2Hz$ ).

In the methodology, the eigenmodes are computed with the 1D SIMSEN model as function of the set of cavitation parameters. Since the two parameters have both influence on damping and frequency of the first eigenmode, they must be considered simultaneously in the optimization process as presented in figure 1. In figure 4 and figure 5, the forced response of the system is computed respectively for two second viscosity values and the resulting pressure fluctuations along the venturi abscissa is plotted. The black solid line corresponds to the outlet excitation frequency set at the first eigenfrequency of the 1D system. The two remaining grey lines represent an excitation frequency out of resonance conditions. The behavior observed with the CFD model is reproduced with the 1D model. Indeed, for excitation frequency different to the eigenfrequency, the cavitation volume behaves like a pressure node for the system. However, with an excitation at the first eigenfrequency, the amplitude near the cavitation volume is higher than the outlet excitation source. Moreover, it is shown that the second viscosity value influences the amplitude of pressure fluctuations near the cavitation volume. Finally, by running the optimization process, the set of 1D cavitation model parameters leading to a matching of the eigenfrequency and the amplitude of pressure fluctuations in Plane2 is given in table 1. Moreover, the dimensionless cavitation compliance $C_c^* = C_c \cdot h_{in} / V_{throat}^3$ is computed.
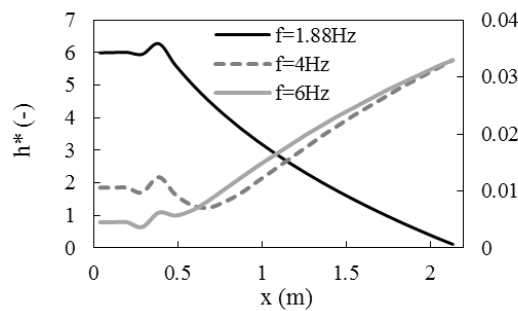


**Figure 4.** Forced response of the system with the set of cavitation parameters $a = 4$ m.s$^{-1}$ and $\mu'' = 280$ Pa.s .
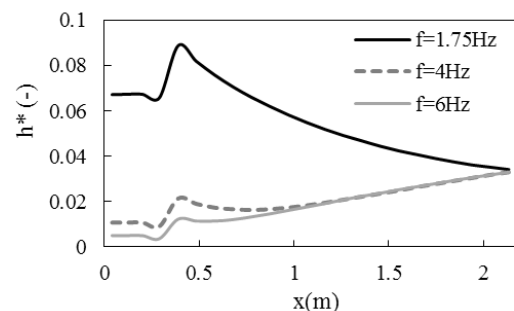


**Figure 5.** Forced response of the system with the set of cavitation parameters $a = 4$ m.s$^{-1}$ and $\mu'' = 1000$ Pa.s .

**Table 1.** Set of cavitation 1D model parameters.

| a | $C_c^*$ | μ" |
|---|---|---|
| (m/s) | (−) | (Pa.s) |
| 4.3 | 3.1 | 790 |

In the methodology, the 1D cavitation model parameters are considered as constant. However, the typical shape of pressure fluctuations observed in resonance conditions in figure 3 suggests non-linear

parameters depending on pressure. By considering constant parameters the simulated system response features sinusoidal pressure and discharge fluctuations as shown in figure 6 and in figure 7.
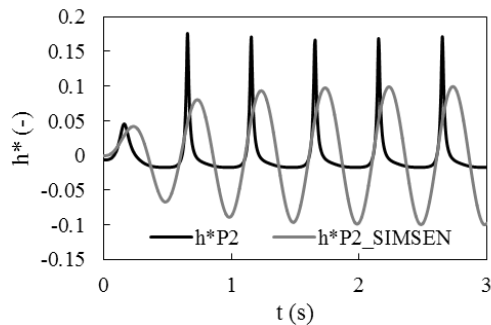


**Figure 6.** Comparison of the simulated pressure fluctuations in Plane2 between the CFD model and the 1D model.
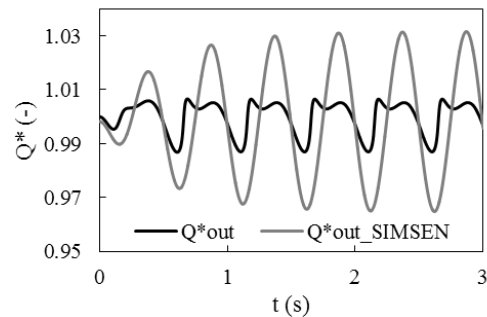
**Figure 7.** Comparison of the simulated downstream discharge fluctuations between the CFD model and the 1D model.

## 4. Conclusions

A methodology based on CFD simulations is setup to identify 1D cavitation model parameters. A venturi geometry is considered as case study and it is shown that a low wave speed and a second viscosity due to the cavitating flow can be identified. Moreover, the 1D model and the CFD model have the same behaviour to an outlet excitation source depending on its frequency. However, parameters are considered as constants and therefore the simulated system response does not reproduce the non-linear CFD simulation results. Further investigations will be carried out by taking into account non-linear parameters depending on pressure. Then, the methodology will be applied to identify parameters of the cavitation vortex rope modelling in the draft tube which involves one additional parameter: the mass flow gain factor. Results will be compared to an identification performed experimentally.

## 5. References

[1]    Landry C, Favrel A, Müller A, Nicolet C and Avellan F 2014 Experimental identification of the local wave speed and the second viscosity in cavitating draft tube flow *J. of Hydraulic Research* In reviewing process.
[2]    S. Barre J, Rolland G, Boitel E, Goncalves and Patella RF 2009 Experiments and modeling of cavitating flows in venturi: attached sheet cavitation *Eur. J. Mech. - B/Fluids* **28** (3).
[3]    Decaix J, Alligné S, Nicolet C, Avellan F and Münch C 2015 Identification of the wave speed and the bulk viscosity of cavitation flows with 2D RANS computations – Part I. In *Proc. of the 9th International Symposium on Cavitation,* Lausanne, Switzerland.
[4]    Alligné S, Nicolet C, Tsujimoto Y and Avellan F 2014 Cavitation surge modelling in Francis turbine draft tube *J. of Hydraulic Research.* **52** (3).