Hydrodynamics of Cavitation Reactors based on Linear and Rotational Flows: Critical Comparison of Orifice, Venturi and Vortex Diodes

Alister Simpson, Vivek V. Ranade*

School of Chemistry & Chemical Engineering, Queen's University of Belfast, Belfast BT9 5AG

*Corresponding author: V.Ranade@qub.ac.uk

Highlights

- Investigated hydrodynamics of three cavitation reactors: orifice, venturi and vortex diode
- CFD models were developed and validated using the published experimental data
- Presented critical analysis of differences and similarities of three cavitation reactors
- Attempted to develop guidelines for possible tailoring of realized cavitation performance

1. Introduction

Hydrodynamic cavitation (HC) may be harnessed for variety of applications ranging from water disinfection, waste water treatment, oxidative desulfurization, enhancing biogas yield of anaerobic digestion [1]. Cavitation can be generated using two fundamental approaches; the most common method is to force the flow through a constriction using an orifice or venturi type device. The other approach involves accelerating the flow rotationally, typically using a rotor – stator device, such as a high-speed homogenizer. An alternative reactor type, based a vortex diode, can also be used to generate a cavitating rotational flow, while potentially overcoming limitations associated with other devices, such as erosion & clogging in orifice and venturi constrictions, and the high costs & energy requirements of rotor-stator type devices. Effective design of HC reactors relies on developing a thorough understanding of the reactor hydrodynamics, in terms of the cavitation inception and evolution across a wide range of operating conditions and geometric parameters. The objective of this work is to systematically investigate the hydrodynamic characteristics of three different cavitating devices; orifice, venturi and vortex diode (Figure 1).

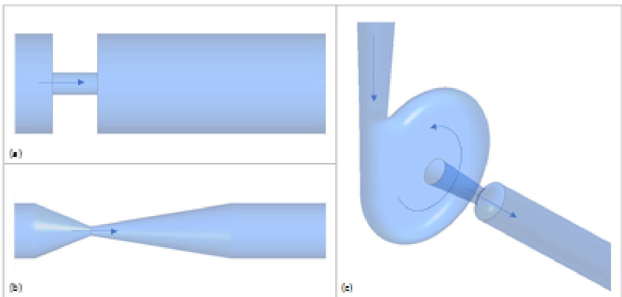

Figure 1. (a) Orifice, (b) Venturi and (c) Vortex diode

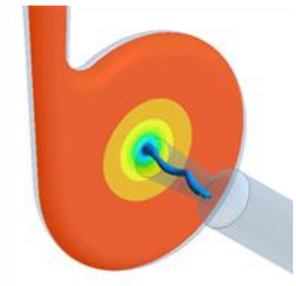

Figure 2. Cavitating vortex diode

The effect of varying key geometric parameters on the hydrodynamic behavior was evaluated for each device type, including the throat aspect ratio, throat inlet profile, and the venturi diffuser angle.

2. Computational Models

Cavitation is a complex mass transfer process spanning a wide range of time and length-scales; as such numerical approaches typically adopt simplified mass transfer relationships based on the predicted local pressure and vapour pressure, with necessary assumptions imposed to account for the presence of non-dissolved gases. In this study an Euler – Euler approach was adopted based on the Singhal cavitation model [], which employs mass source and sink terms based on a reduced form of the Raleigh-Plesset equation. As part of this work a thorough sensitivity study was performed to determine the influence of mesh density, turbulence model, choice of numerical procedure and also the choice of 2D vs 3D modelling approaches.

The Eulerian methods provide a means to quantify and compare the cavitational yield in different reactor designs in terms of the evolution of macro-scale vapour cavities (Figure 2). To investigate the behavior of individual sub-grid cavities, the Lagrangian calculations were performed on the solved Eulerian multi-phase flow fields to obtain quantitative predictions of the pressure – time history they experience, including local turbulence effects, in the cavitating regime.

3. Results and discussion

Figure 3 presents a sample comparison CFD predictions of a cavitating orifice against experimental data, showing good agreement. The effect of cavitation onset is captured, as well as the reduction in flow capacity caused by the formation the vapour cavity. The validated CFD models were then used to evaluate the cavitation effect produced in different orifice designs across a wide range of operating conditions, including the aspect ratio and orifice entry profile. Figure 4 shows a similar comparison of computed and measured characteristics for a venturi device, along with a comparison of the change in predicted cavitation effect with varying aspect ratio. Additional studies were carried out to identify the influence of diffuser exit angle.

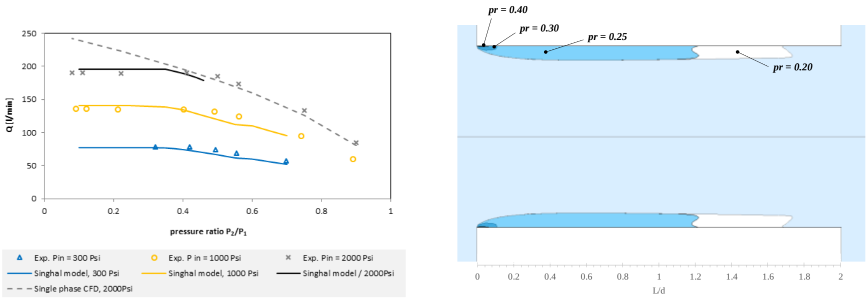

Figure 3. (a) Cavitating orifice; CFD vs test, (b) Comparison of predicted vapour cavity in an orifice with increasing pressure ratio

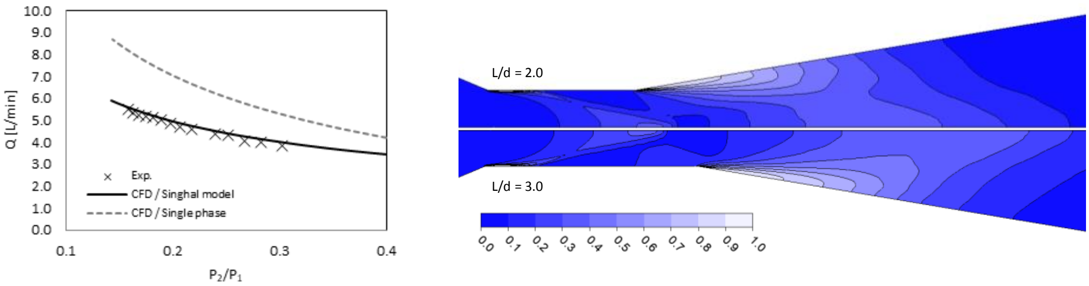

Figure 4. (a) Cavitating venturi; CFD vs test, (b) Comparison of predicted vapour cavity in a venturi for 2x throat aspect ratios

4. Conclusions

The developed Eulerian multi-phase CFD models showed good agreement with measured hydrodynamic characteristics of three cavitation reactors: orifice, venturi and vortex diode. Influence of key design and operating parameters was elucidated using comprehensive simulations. For orifice reactors, the aspect ratio and inlet radius has a pronounced effect on the resulting cavitation effect. An optimum aspect ratio of 2.0 was identified. Orifice inlet radii / throat radius ratios in excess of 5% were found to lead to significant delays in cavitation inception. For venturi designs, the diffuser angle was found to significantly affect the cavitation characteristics, with shallower angles found to lead to earlier onset. Predicted cavitation in vortex diode devices was found to be restricted to a central vortex core in the axial outlet port, fundamentally differing from the nature of cavitation in venturi / orifice designs.

References

[1] Ranade, V.V., Plenary Lecture at CAMURE10 & ISMR9 held at Qingdao China, July 2017

Keywords
Cavitation reactors, diode, orifice, venturi, CFD