Flow characteristics of vortex based cavitation devices: Computational investigation on influence of operating parameters and scale

Simpson, A., & Ranade, V. V. (2019). Flow characteristics of vortex based cavitation devices: Computational investigation on influence of operating parameters and scale. *AIChE Journal*, [e16675].
<https://doi.org/10.1002/aic.16675>

Published in:
AIChE Journal

Document Version:
Peer reviewed version

Queen's University Belfast - Research Portal:
[Link to publication record in Queen's University Belfast Research Portal](#)

Publisher rights
Copyright 2019 Wiley. This work is made available online in accordance with the publisher's policies. Please refer to any applicable terms of use of the publisher.

General rights
Copyright for the publications made accessible via the Queen's University Belfast Research Portal is retained by the author(s) and / or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy
The Research Portal is Queen's institutional repository that provides access to Queen's research output. Every effort has been made to ensure that content in the Research Portal does not infringe any person's rights, or applicable UK laws. If you discover content in the Research Portal that you believe breaches copyright or violates any law, please contact openaccess@qub.ac.uk.

Flow Characteristics of Vortex Based Cavitation Devices

Computational Investigation on Influence of Operating Parameters and Scale

Alistar Simpson and Vivek V. Ranade*

School of Chemistry and Chemical Engineering

Queen's University Belfast, Belfast BT9 5AG, UK

*Email: V.Ranade@qub.ac.uk

Abstract

Vortex based hydrodynamic cavitation devices offer various advantages over conventional linear flow devices, such as early inception, low erosion risk and higher cavitation yield. Despite several promising applications, the key underlying flow characteristics are not yet adequately understood. This paper presents results of a computational investigation into cavitation in vortex devices. Multiphase computational fluid dynamics (CFD) results are presented and compared with experimental data on pressure drop over a range of flow rates. The results highlight the unique hydrodynamic characteristics of this type of device in relation to conventional linear flow reactors; cavitation inception occurs in the liquid bulk away from solid surfaces, and rapid pressure recovery rates are achieved. The models were used to simulate detailed time-pressure histories for individual vapour cavities, including turbulent fluctuations. The developed approach, models and results will provide a sound and useful basis for comprehensive multi-scale modelling of vortex-based devices for hydrodynamic cavitation.

Key words: Vortex cavitation, CFD, multiphase, turbulence models, scale - up

1 Introduction

Vortex based flow devices have found extensive use in a wide variety of industrial applications, most notably in cyclonic separation systems ^{1,2}. Recent novel examples of vortex flow devices in chemical reaction engineering include rotating fluidized bed reactors ³⁻⁵, in which particle bed fluidization is achieved by generating highly swirling flow within a fixed geometry reactor. Another promising industrial application for vortex-based reactors is in hydrodynamic cavitation applications, where the intense collapse of vapour cavities in a flowing liquid medium can be utilized to facilitate a range of industrial processes. The focus of the present study is a novel type of vortex cavitation reactor, based on a vortex diode. Vortex diodes are bi-directional fluid flow devices, characterised by a high flow resistance in one direction relative to the other. Fluidic diodes of different construction have been in use since the early part of the 20th Century ^{6,7}, and the absence of moving parts and low maintenance overhead makes them particularly suited to operation in harsh environments. This has led to their application in the nuclear industry as a type of non-return valve ⁸, and as part of pumping systems ⁹. The basic construction of a vortex diode features a tangential inlet, swirl chamber and an axial outlet port (**Error! Reference source not found.**).

In “reverse” operation, flow enters through the tangential inlet, which quickly establishes a vortex in the main swirl chamber with an associated radial reduction in static pressure. With sufficiently high tangential velocities, the local pressure at the core of the vortex reaches the vapour pressure of the working fluid. Vapour cavities formed at these low pressures may experience oscillatory growth, eventually collapsing to generate spots of extreme temperatures and pressures (5000°K and 1000 atm respectively ^{10,11}). These collapse events are accompanied by localised shock waves and extremely high shear rates, and as a result cavitation can be famously damaging in pressurised fluid handling equipment. When carefully harnessed however, these effects can be utilized to produce intense reactions; at the point of collapse, the contents of the cavity dissociate and release highly oxidising radical species, including hydroxyl radicals (OH), which can be utilized in advanced oxidation processes.

The successful use of hydrodynamic cavitation (HC) has been reported for a wide range of industrial applications; for wastewater in particular it has been identified as a promising method for the removal of

micropollutants, and numerous studies have highlighted its potential to degrade a range of contaminants recalcitrant to conventional treatment methods. Comprehensive summaries can be found in the reviews compiled by Rajoriya et al.¹², Dular et al.¹³ and Ranade and Bhandari¹⁴. To highlight some specific examples, Kalumuck and Chahine¹⁵ conducted experiments to degrade p-nitrophenol (pNP) using a recirculating loop cavitation system, and reported reductions in concentration of the order of 90%. Comparison with ultrasonic cavitation showed hydrodynamic cavitation to yield an increase in energy efficiency equal to two orders of magnitude. Capocelli et al.¹⁶ have reported similar success in degrading pNP using HC, with removal rates of over 40% achieved using a venturi loop reactor. Successful use of vortex cavitation reactors has recently been reported in degrading industrial effluents; Suryawanshi et al.¹⁷ have demonstrated the effectiveness of vortex diode reactors in degrading industrial solvents, achieving 80% toluene removal. Beyond wastewater applications, the successful adoption of HC technology has been reported in a wide range of industrial processes, including microbial disinfection¹⁸, fuel desulfurization¹⁹, biodiesel synthesis^{20,21}, bio-mass pre-treatment^{22,23}, and in food and beverage production^{24,25} to name just a few.

The published literature on HC applications features a range of non-optimised reactor designs, most typically of venturi or orifice type construction. By accelerating fluid through a restriction, cavitation in such devices initiates and evolves around the surface of the restriction itself^{26,27}. This imposes some inherent limitations; firstly the reactor surfaces are exposed to potential cavitation erosion, which can greatly reduce the operational lifetime and performance of the device. Secondly the restriction itself creates a clogging risk for applications involving solids loading, such as bio-mass pre-treatment. Vortex based devices can potentially overcome these specific limitations, as cavitation occurs in the core of the fluid bulk as opposed to the surface of the restriction, as occurs in orifice and venturi type devices^{26,27}. As such, with the correct design approach there is the potential to harness the benefits of cavitation whilst eliminating the risks of clogging and erosion. The flow in such devices is however highly complex; Pandare and Ranade²⁸ presented a detailed description of the key single phase flow features and complexities in vortex diodes in reverse operation. They described the formation of an inherently unsteady, precessing vortex core (PVC) along the device axis, predicted to oscillate at frequencies of the order of 60Hz. This precessing vortex behaviour has also been widely reported in vortex separation units^{2,29}. Along the axis of the vortex unit, Pandare and Ranade also reported a significant core of reverse flow in the axial port, which persists for up to 30 diameters into the downstream pipework. Similar

1 observations of reverse flow in vortex units have been reported by Niyogi et al.⁵, who presented a detailed
2 description of secondary flow patterns in gas vortex devices of similar geometry. Using 2D axi-symmetric CFD
3 modelling, Niyogi et al. found a backflow region to develop in the axial port at relatively low Reynolds number
4 ($Re = 14$), which was found to further develop with increasing Reynolds number, and persist until the observed
5 extent of the downstream port at $Re = 28$. In the turbulent regime, this backflow region was shown to fill half
6 of the cross-sectional area at the maximum Reynolds number of 13,000. All of these aforementioned numerical
7 simulations consider single phase flow only, and to date no detailed description of multiphase flows with phase
8 change have been presented for vortex reactors in the open literature.

9
10 Vortex cavitation itself has been the subject of numerous studies, primarily in the context of tip vortices
11 formed at the trailing edge of hydrofoils. This specific form of vortex formation is understood to be a major
12 contributor to erosion in hydraulic machinery, as well as noise and vibration. Arndt et al. have highlighted some
13 of the complexities of such cavitating vortices, which exhibit a range of broadband pressure fluctuations and
14 complex collapse characteristics^{30,31}. Recent computational modelling efforts have highlighted novel ways to
15 predict the formation and evolution of cavitating line vortices, such as the combined lagrangian-vof approach
16 developed by Ma et al.³², and more recently a modelling approach has been presented by Chen et al. which
17 offers a method to predict tip vortex cavitation inception by accounting for water quality³³. Confined cavitating
18 vortex flows, as found in vortex diodes, have received relatively little attention on the other hand, and advances
19 in device design depend on developing an improved understanding of the key hydrodynamic features in terms
20 of the two phase flow fields and turbulent fluctuations. The present work aims to address these gaps in current
21 understanding, using a series of numerical investigations to highlight the unique features of the device in terms
22 of the flow patterns, pressure gradients and turbulence quantities. Following the 2D axi-symmetric approach
23 by Niyogi et al.⁵, we present a detailed description of multiphase flow with phase change for cavitating vortex
24 devices across a range of operating conditions, and how these characteristics vary with device scale up.

Mathematical models

The following section discusses the physical models chosen to simulate the two-phase flow (pure liquid water and water vapour), turbulence characteristics and mass transfer in a cavitating vortex device.

Flow and turbulence

The working medium is treated as a single fluid, comprised of a homogeneous mixture of two phases. The continuity equation for the mixture flow is written as:

$$\frac{\partial}{\partial t}(\rho_m) + \nabla \cdot (\rho_m \vec{u}_m) = 0 \quad (1)$$

Where ρ_m is the mixture density, and $\vec{u}_m$ is the mass-averaged mixture velocity. The corresponding momentum equation for the mixture flow, assuming that both phases share the same velocity field, is written as:

$$\frac{\partial}{\partial t}(\rho_m \vec{u}_m) + \nabla \cdot (\rho_m \vec{u}_m \vec{u}_m) = -\nabla p + \nabla \cdot [\mu_m (\nabla \vec{u}_m + \nabla \vec{u}_m^T)] + \rho_m \vec{g} + \vec{F} \quad (2)$$

Where μ_m is the mixture viscosity, $\rho_m \vec{g}$ is the gravitational body force, and the term $\vec{F}$ accounts for additional external body forces applied to the fluid volume (i.e. that may arise from interaction with dispersed phases).

Applying a Reynolds Averaging approach, the velocity terms in Equations (1) and (2) are replaced by the sum of their mean ($\bar{u}$) and instantaneous (u') components, $u = \bar{u} + u'$, and an ensemble average taken. Additional terms are then introduced, known as the Reynolds stresses, each having the general form $\partial/\partial x_j(-\rho \bar{u}_i \bar{u}_j)$. In order to close the momentum equation, additional physical models are required to approximate the Reynolds stresses. One approach is to use a Reynolds Stress Model (RSM), which involves solving separate transport equations for each of the additional Reynolds stresses (6 in total for 3D cases). Full details of this approach can be found in Wilcox³⁴. More typical in RANS approaches is to employ the Boussinesq hypothesis, which approximately relates the Reynolds stresses to the mean velocity gradients in the flow as follows:

$$-\rho \bar{u}_i \bar{u}_j = \mu_t \left(\frac{\partial u_i}{\partial j} + \frac{\partial u_j}{\partial i} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (3)$$

This expression introduces the concept of the turbulent viscosity, μ_t ; a scalar quantity which has a value proportional to the local turbulence properties. This quantity is typically modelled through additional transport

equations for the turbulent kinetic energy (k) and turbulent dissipation rate (ε), or the specific dissipation rate (ω). In the present work, the Shear Stress Transport (SST) k - ω model of Menter ³⁵ has been selected for use, in which the turbulent viscosity, μ_t is defined as:

$$\mu_t = \alpha \frac{\rho k}{\omega} \quad (4)$$

Where the pre-multiplier α is typically computed from a number of further sub-functions and empirical constants, presented in detail in ^{35,36}. The transport equations for k and ω are then written as follows:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \quad (5)$$

and:

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + S_\omega \quad (6)$$

G represents the generation, Γ the effective diffusivity, and Y is the dissipation due to turbulence ³⁷. The subscripts k and ω denote turbulence production and dissipation respectively, and S_k and S_ω are user-defined source terms.

Standard two-equation turbulence models are known to suffer from excessive predictions of turbulence kinetic energy, G_k , in the vicinity of stagnation points, which is particularly relevant to swirling flows as the swirl-axis itself represents a stagnation condition. Limiting formulations for the production term, G_k , have therefore been proposed by Menter ³⁶, and also Kato and Launder ³⁸. The formulation proposed by Menter is as follows:

$$G_k = \min[G_k, C_{lim} \rho k] \quad (7)$$

Where the limiting coefficient, C_{lim} has a default value of 0.9. Kato and Launder proposed an additional limiting correction on the production term, based on the observation that high turbulence kinetic energy production in stagnation regions is generally caused by the calculation of excessive shear strain rate, γ . As such, this is corrected by the local vorticity rate, Ω , which tends to low values in stagnation regions:

$$G_k = \mu_t \gamma \Omega \quad (8)$$

Therefore two mechanisms exist by which the turbulence kinetic energy production term can be adjusted to mitigate against excessive predictions in axially symmetric swirling flow cases; firstly through adjusting the μ_t term via Eq. (4), and secondly by applying limiters to the production term, G_k , via equations (7) and (8). The sensitivity of the results to the choice of closure model is presented in full detail in the subsequent dedicated section on turbulence modelling closure sensitivity.

Cavitation model

The two-phase flow field in this work is modelled by assuming a homogeneous mixture of water liquid and vapour, with the vapour mass fraction, f , computed locally using a separate transport equation:

$$\frac{\partial}{\partial t}(\rho_m f) + \nabla \cdot (\rho_m \vec{v}_m f) = \nabla \cdot (\Gamma \nabla f) + R_e - R_c \quad (9)$$

Here ρ_m , ρ_v and ρ_l refer to the densities of the mixture, vapor and liquid respectively. An additional pair of mass source and sink terms are now introduced for the evaporation (R_e) and condensation (R_c) of the vapour phase. Numerous modelled source and sink terms have been proposed to account for cavitation mass transfer in Eq. (9), with the majority of approaches deriving these terms from a reduced form of the Rayleigh-Plesset equation, commonly shortened to the “R-P” equation:

$$\rho \left[R_B \frac{d^2 R_B}{dt^2} + \frac{3}{2} \left(\frac{dR_B}{dt} \right)^2 \right] = p_B - p - \frac{2\sigma}{R_B} - \frac{4\mu}{R_B} \frac{dR_B}{dt} \quad (10)$$

In this expression, R_B is the bubble radius, p_B refers to bubble surface pressure, and p is the local liquid phase pressure. Full derivation of the R-P equation is presented in ³⁹, and numerous examples of solutions of the R-P equation exist in literature, for example Alehossein and Qin ⁴⁰. Commonly, when modelling cavitation mass transfer mechanisms in CFD codes, the R-P equation for bubble growth is used to approximate void propagation. To derive mass transfer terms compatible with the Eulerian-Eulerian multiphase approach, the surface tension,

viscous damping and higher order acceleration terms in Eq.(10) are neglected to produce a mass transfer rate term of the following general form:

$$\frac{dR_B}{dt} \cong (-1)^n \sqrt{\frac{2}{3}} \frac{(p_B - p)}{\rho_l} \quad (11)$$

Where $n=1$ during bubble expansion / evaporation, and $n=2$ during the condensation phase. There exists in open literature a variety of source term approximations of this general form, for example those proposed by Schnerr and Sauer ⁴¹, Zwart et al. ⁴² and Singhal et al. ⁴³. The Singhal model was chosen for use in the present study, given below in Eqs. (12) and (13).

$$R_e = C_1 \frac{\sqrt{k}}{\sigma} \rho_v \rho_l \left[\frac{2}{3} \left(\frac{p_v - p}{\rho_l} \right) \right]^{1/2} (1 - f_v - f_g) \quad (12)$$

$$R_c = C_2 \frac{\sqrt{k}}{\sigma} \rho_v \rho_l \left[\frac{2}{3} \left(\frac{p - p_v}{\rho_l} \right) \right]^{1/2} f_v \quad (13)$$

Real engineering liquid systems typically contain a small quantity of non-condensable gas (NCG), present in a dissolved state and potentially as bubbles introduced by aeration. Aeration may be intentionally introduced, or unintentionally through system leaks or other means of entrainment. The presence of non-condensable gas is accounted for through the additional mass rate term, f_g , in Eq.(12). The precise amount of non-condensable gas is typically an unknown in most engineering applications however, as direct measurements are extremely difficult. Solubility data for air in water gives an equilibrium concentration of 8 g/m³ (8ppm) at typical laboratory room conditions of 1 atm and 22°C, however the total gas content can vary according to a wide range of additional factors, such as prior liquid processing and storage steps, for example; degassing, pressurization, and the length of time the system has been exposed to air. Liquid vortex units in particular are known to feature a distinct core of non-condensable gas above the vapour limit of the working fluid ^{2,29,33,44,45}; as such, the Singhal model was chosen in order to take account of this. For the purposes of the present investigation, it is assumed that the liquid system free surface is exposed to atmosphere for a sufficient length of time for it to reach equilibrium with air at atmospheric pressure and room temperature, and that the system has also had sufficient time to settle such that any entrained macroscopic bubbles rise to the free surface and escape. As such, the fixed fraction of NCG has been set at 8g/m³ (8ppm) throughout.

The evaporation and condensation source terms in Eqs. (12) and (13) also feature a pair of pre-multipliers, C_1 and C_2 . Empirical pre-multipliers are also incorporated into other cavitation models of this general type, which highlights some fundamental features of mass transfer terms based on this general “reduced Rayleigh Plesset” approach described by Eq. (11); firstly that the bubble growth rate term does not directly correlate with the magnitudes of cavitation mass transfer rates observed in reality, and secondly that the basic approach implies equivalent evaporation and condensation rates, which requires additional correction. In validating their cavitation calculations, Singhal et al.⁴³ compared predictions against available published empirical data sets for a range of orifice, hydrofoil, blunt body and impeller flows, and stated that hundreds of permutations were considered in order to determine suitable values for this pair of constants in order to reliably and repeatedly match experimental values of flow rates, discharge coefficients and cavitating flow patterns. The values of the empirical constants C_1 and C_2 were set as 0.02 and 0.01 respectively in the original publication, and these recommended values have been adopted throughout the present study without modification.

Discrete Phase Model

Using the two-phase flow field predictions as a basis, the trajectories of individual cavities were simulated using the Lagrangian approach. Since the two-phase flow field is already computed using the models described above, one-way coupling was assumed between the discrete cavities and the continuous mixture while simulating individual cavity trajectories. The particle trajectories are computed by integrating the force balance for a discrete particle of a series of discrete time steps; the force balance is given as:

$$\frac{d\vec{u}_p}{dx} = \frac{\vec{u} - \vec{u}_p}{\tau_r} + \frac{\vec{g}(\rho_p - \rho)}{\rho_p} + \vec{F} \quad (14)$$

This equates the particle inertia with the forces acting on the particle. The first term on the right-hand side of the equation is the drag force per unit mass of the particle, where τ_r is the particle relaxation time. The second term is the force due to gravity, and the final term, $\vec{F}$, is an additional acceleration term. The presence of a strong radial pressure gradient, and large density differences between the discrete phase and continuous phase in this case also necessitate the inclusion of the virtual mass force and pressure gradient terms, presented below in equations (15) and (16).

$$\vec{F} = 0.5m_p \frac{\rho}{\rho_p} \vec{u}_p \nabla \vec{u} - \frac{d\vec{u}_p}{dt} \quad (15)$$

$$\vec{F} = m_p \frac{\rho}{\rho_p} \vec{u}_p \nabla \vec{u} \quad (16)$$

The influence of continuous phase turbulence on the tracked particles is accounted for by separating the velocity, u , into mean and instantaneous components:

$$u = \bar{u} + u' \quad (17)$$

In the work presented here, the discrete random walk model, or “eddy lifetime” model is used to include the effects of turbulence on the discrete cavity trajectories. In this approach, each discrete particle is considered to interact with a succession of discrete turbulent eddies which modify their instantaneous velocities. This involves introducing two modelled terms; firstly, the random fluctuating component of velocity is calculated as a function of the local turbulent kinetic energy value:

$$u' = \zeta \sqrt{2k/3} \quad (18)$$

Where ζ is a normally distributed random number. Secondly, the concept of a particle eddy lifetime, T_L , is introduced to define the time intervals over which this random fluctuating component is updated. This “eddy lifetime” is approximated as a function of the local turbulence frequency:

$$T_L \approx 0.15 \frac{k}{\varepsilon} \quad (19)$$

Additional limits can be placed on the maximum time step size; in this study a minimum of 5 time steps is also imposed across any given computational cell. Using this approach in combination with a two-equation turbulence model provides an efficient means to couple individual cavity pressure histories with realistic turbulent multiphase hydrodynamic behaviour.

Results & discussion

Diode Configurations

Error! Reference source not found. presents the construction and dimensions of the vortex devices considered in the present study. Four geometrically similar units were studied of different scales, the smallest having a 6mm throat diameter (d_t), and largest having a 48mm throat diameter.

Numerics and convergence strategy

The model equations described in the preceding section were all solved using commercial CFD code, Ansys Fluent (v17). Following similar approaches adopted by Niyogi et al. ⁵, the present study primarily makes use of a 2D axi-symmetric approach. This simplified 2D approach necessarily misses some of the detail revealed by a time-resolved 3D calculation, and with this in mind an initial series of transient 3D CFD calculations were performed in order to derive appropriate boundary conditions for the 2D models. This reduced 2D approach subsequently allowed a larger number of operating conditions and parameters to be explored within feasible computational timescales. A detailed comparison of the 2D and 3D approaches is presented and discussed in the subsequent sections. Although the 2D simulation results presented here do not exhibit prominent unsteady behaviour in the predicted velocity and pressure fields, an unsteady solver was required in order to ensure convergence across all of the cases studied. Time steps of the order of $1e-4$ to $1e-6$ were used across the range of operating conditions set out in Figure 2. The SIMPLE algorithm was used for pressure velocity coupling, with the PRESTO! discretisation scheme applied for pressure. For the momentum and turbulent quantities 2nd order discretization was applied in each instance.

Mesh and boundary conditions

At the outset of the 2D simulations, a comprehensive grid sensitivity study was performed to establish a suitable levels of mesh refinement in the regions of particular interest; **Error! Reference source not found.** illustrates the mesh topology and refinement levels. The boundary conditions are also shown; at the inlet, velocity profile data extracted from 3D simulation results was applied. The outlet of the device in practices discharges freely to a holding tank, and this was represented by setting a fixed pressure equal to 1 atmosphere at the outlet

boundary, positioned 40x diameters downstream of the axial port exit. The computed gradients of swirl velocity and turbulent kinetic energy through the mid-section of the smaller device chamber (device A) are compared for 4 different mesh sizes in **Error! Reference source not found.b** & **Error! Reference source not found.c** respectively. Negligible differences were found in the predictions above an overall mesh count of 120,000 cells, which was therefore set as the grid count for the remainder of the present study for device A. This featured a cell size of 0.08mm ($= d_t/75$) through the centre of the device and axial port, and a near wall initial cell height of 0.003mm, ensuring y^+ values of unity even at the maximum simulated flow rate. For the larger geometries B, C and D, the near wall sizing was held constant, and the mesh spacing adjusted so as to result in final mesh sizes which scaled approximately with device scale. The highest cell count for the largest 48mm device was 840,000 elements.

Turbulence closure model sensitivity

The interaction between cavitation and turbulent flow structures is extremely complex, and the detailed mechanisms are not yet fully understood. The difficulties in turbulence modelling in cavitating flows and proposed approaches have been well documented in open literature^{46–49}; for detached flows in particular, such as those produced over hydrofoils, there is strong interaction between the eddy vortices formed and the generated vapor phase, creating strong pressure fluctuations^{50–52}. These complex turbulent features present significant modelling challenges; In studying a cavitating venturi, Reboud et al.⁴⁶ made the argument that the standard k- ϵ model tends to produce high turbulent viscosity (μ_t) predictions in the separation region, which acts to dampen unsteady effects, leading to the prediction of a stable, fixed separation bubble. To properly capture unsteady cavity shedding, the authors found it necessary to apply an essentially arbitrary pre-multiplier to limit the turbulent viscosity. A number of examples exist in open literature of favourable correlations being obtained between models applying this type of empirical limiter with experimental data for unsteady cavitation^{53–57}. For detached venturi flows Charriere et al.^{48,49} have successfully demonstrated that the use of a Reboud type eddy-viscosity limiter in RANS models can reliably predict the re-entrant jet phenomenon for 2D venturi configurations⁴⁸. The arbitrary nature of the applied correction factors however means that it is likely to be dependent on geometry, and in lieu of sufficient experimental data to “tune” the turbulence model for the present flow case no such additional limiters have been applied directly to the turbulent viscosity term in

1 this study. More recent studies on hydrofoils highlight the highly three dimensional, complex interactions
2 between detached eddy vortices and the secondary gas phase, and in such cases some degree of scale resolving
3 is required to obtain reliable results^{58,59}.

4
5 In the case of a confined, single vortex of the type studied in the present work cavitation is initiated and
6 develops along the swirl axis, and the interactions between turbulence and the secondary phase in this scenario
7 are not well understood. At the axis of the vortex chamber there is no separation, or eddy vortex shedding,
8 however a stagnation region of a different nature is present; tangential velocities go to zero at the axis, and the
9 axial velocities also tend to low / near zero values. As such, as is the case for cases with detached eddy shedding,
10 these stagnation regions require careful consideration when imposing a turbulence closure model so as to avoid
11 overprediction of the production term. Additionally, a recent study by Zaman et al.⁶⁰ has highlighted the fact
12 that constrained highly swirling flows can lead to re-laminarization at nominally turbulent Reynolds numbers; as
13 a consequence it could be anticipated that standard two-equation approaches such as the $k-\epsilon$ model would also
14 give rise to significant discrepancies in single vortex flows. In the present study it is particularly crucial that the
15 choice of turbulence closure model leads to realistic tangential velocity profiles, as this tangential acceleration
16 produces the low-pressure regions along the axis of the device that drive cavitation mass transfer. Previously
17 Pandare and Ranade²⁸ presented results for the present vortex unit geometry using the $k-\omega$ SST, re-normalised
18 group (RNG) $k-\epsilon$ as well as a Reynolds Stress Model (RSM). The $k-\epsilon$ model was found to underpredict the swirl
19 velocity magnitudes, while the $k-\omega$ SST and RSM models showed close agreement. Other studies of strong
20 swirling flows, such as that performed by Niyogi et al.⁵ have found that omega-based Reynolds stress models
21 produce more realistic results than eddy-viscosity approaches. In order to determine an appropriate choice of
22 closure model in the present case, a number of different modelling approaches were evaluated and compared
23 with full 3D predictions. Particular focus was paid to the prediction of the tangential velocity profiles to evaluate
24 the tendency of the closure model to overpredict the turbulence production term at the vortex core. The models
25 evaluated include an ω -based RSM³⁴, a laminar calculation, the RNG $k-\epsilon$ as well as two different variations of the
26 $k-\omega$ SST model. The two SST $k-\omega$ variants comprise the unmodified SST- $k-\omega$, and an SST- $k-\omega$ model incorporating
27 both turbulence production limiting functions of Menter, and Kato and Launder presented previously in
28 Equations (7) & (8). These relationships can be combined to produce the following limiting function:

$$G_k = \min[\mu_t \gamma \Omega, C_{lim} \rho k] \quad (20)$$

Where the default value of the limiting coefficient, $C_{lim} = 0.9$, has been retained from the original reference³⁶.

Comparisons of the swirl velocity profiles using each approach are presented in **Error! Reference source not found..** The results indicate that the k- ε model underpredicts the swirl magnitudes in the current case, owing to an over-prediction of turbulence kinetic energy. This result is somewhat expected owing to the widely reported deficiencies of ε -based models in similar flow scenarios^{60,61}; perhaps less expected however is the difference in the unmodified SST k- ω model predictions in 2D & 3D. The 2D SST k- ω models show significantly lower tangential velocities, which correspondingly mean that the minimum pressure, and thus cavitation inception, at the axis of the swirl chamber is similarly under-predicted. This suggests that the imposition of zero velocities in the radial and swirl directions at the axis leads to an over-prediction of turbulence production in 2D, which necessitates the introduction of additional production limiting terms. Imposing the combined limiting formulation given in equation (20) to the production term in the SST k- ω model was found to be successful in achieving much closer agreement, producing results in line with the RSM model. This approach was then subsequently adopted across the full range of operating conditions, and produced good agreement with experimental data across the investigated range, and at different scales (See experimental comparison shown in **Error! Reference source not found.** in the following section).

Comparison of 2D and 3D simulations

To judge the capability of the 2D approach to capture the key hydrodynamic characteristics of the device, at the outset of the present study an initial set of 3D, time-resolved CFD calculations were performed. 3D calculations were performed on a hexahedral grid of 4.1 million elements, which resulted from a separate mesh independency study. The 3D results were initially used to derive appropriate boundary conditions for the 2D models. Velocities at the inlet plane of the 2D models were sampled from the time averaged 3D results at radius $r = r_i$, (**Error! Reference source not found.**), and non-dimensional radial & tangential velocity profiles then extracted which could be imposed as boundary conditions for 2D axi-symmetric models. These inlet profiles were then scaled accordingly to simulate at a range of operating conditions. One prominent feature of the flow conditions illustrated in **Error! Reference source not found.** is the near-zero radially inward mass flow across a

significant portion of the span at the sample radius / 2D inlet radius, r_i , with the majority of the mass flux concentrated towards the outer walls of the swirl chamber.

The predicted swirl strength through the midplane of the swirl chamber is compared for the 3D & 2D approaches in **Error! Reference source not found.a**. One notable omission from the 2D predictions is vortex precession, which is a common feature of vortex flows^{62,63}. Previous studies of single-phase flow in vortex diodes has reported the presence of a precessing vortex core (PVC), with a frequency of the order of 60 Hz²⁸. The 3D results in **Error! Reference source not found.a** are assembled from a time average, and despite ignoring precession effects close alignment is found with the 2D axisymmetric approach in both the maximum swirl ratio and gradient towards the centre of the device. **Error! Reference source not found.b** presents a similar comparison of the corresponding predictions of static pressure through the chamber mid-plane, again showing close agreement between the two modelling approaches. The comparison plots highlight a slight eccentricity of the minimum pressure core found by the 3D approach; the tangential inlet in this instance generates a circumferential bias in mass flux, which pushes the low-pressure core slightly off-axis. Despite ignoring these circumferential non-uniformities, the 2D approach shows good general agreement between the axisymmetric model and the full 3D CFD approach in terms of the magnitude of pressure reduction at the vortex core, which is crucial in the study of cavitation inception and evolution within the vortex device.

Comparison with experimental data

Error! Reference source not found. presents the predicted pressure drop, Δp , across devices A & B with increasing throat velocity, u_t , alongside experimentally measured values. The 2D predictions provide close agreement at both device scales. In addition to results from 2D multi-phase simulations, results from corresponding single phase calculations are also included in **Error! Reference source not found.** for device A, showing the characteristic curve to overlap the multi-phase predictions. This is in contrast to linear flow devices such as orifice and venturi, where the onset of cavitation is accompanied by a reduction in flow rate at a given pressure ratio in comparison to single phase predictions^{26,64,65}. In such linear flow devices, cavitation initiates and grows at the restriction. The gas phase therefore presents an effective reduction in liquid flow area, which acts to reduce the flow rate in the cavitating regime. In a vortex diode, cavitation initiates along the axis of the

chamber, rather than at the device throat, and as the results in Figure 6a imply the gas phase therefore does not limit the flow rate over the range of conditions studied here. Predictions indicate that the cavitating region does not extend to fill the axial port even at the highest flow rates studied here, and it may be possible that higher flow rates could lead to a deviation in this trend. For the present case, cavitation inception is indicated in **Error! Reference source not found.**, which shows both the minimum pressure (dotted line) and cavitation number, σ (solid line), as a function of throat velocity. In this case the cavitation number is defined by Eq.(21), with the maximum tangential velocity applied as the characteristic velocity. Using this definition, cavitation numbers of 1 were found to correlate to the minimum pressure in the device reaching the vapour pressure, p_v . Cavitation inception points at both scales are similar, with inception for the larger device B predicted slightly earlier at throat velocities of 2.5 m/s, whereas inception is indicated at 2.7 m/s for device A.

$$\sigma = \frac{p_2 - p_v}{\frac{1}{2}\rho u_{\theta max}^2} \quad (21)$$

In addition to comparing computed and measured flow rate and pressure losses, ideally it would be useful to compare predicted velocity and turbulent kinetic energy profiles, however no such experimental data currently exists in open literature for multi-phase vortex units. Further experimental investigations are planned to carry out detailed flow visualization and particle image velocimetry (PIV) measurements of cavitating flows in vortex devices. In order to inform the design of an appropriate experimental facility and measurement strategy, it is first important to understand the key flow features, and the subsequent sections therefore describe numerical studies which span a wide parameter space (geometric as well as operational) to elucidate these key flow features. These results are also qualitatively compared with published studies on the key flow features of geometrically similar gas vortex units ⁵, and any observed similarities and differences highlighted. The results are presented and discussed in the following section.

Predicted flow fields

The following section presents an examination of the key features of the 2-phase flow behaviour in the liquid vortex unit. To judge the performance of the unit as a cavitation reactor it is initially important to understand the swirl generation, and thus the minimum pressures attained, and subsequently to determine the

location and extent of the cavitating vortex core. **Error! Reference source not found.a** & **Error! Reference source not found.b** present in plane streamlines and velocity vectors to illustrate the flow through the device chamber and axial port at two conditions: a throat velocity, u_t of 2.3 m/s, corresponding to a Reynolds number of 13,775, and at $u_t = 3.5$ m/s, $Re = 20,960$, which is well into the cavitating regime. The velocity vectors have been normalised to better depict the local flow directions; it should be noted that this results in small magnitudes of reverse flow near the inlet plane being represented by vectors which project out of the domain; this exaggerates the fact that radial velocities are near zero across most of the inlet span, with locally high inward velocities along the outer chamber walls. **Error! Reference source not found.c** presents results for the larger scale device B at a throat velocity of 3.5 m/s, $Re = 41,925$. The presence of two counter-rotating secondary flow vortices is shown in the chamber, generated by the high inward radial velocities along the chamber walls. This observation is similar to that found for gas vortex units by Niyogi et al. ⁵, who also described the formation of a counterflow region in their vortex chamber. Also highlighted by Niyogi et al. was the formation of a backflow region in the axial port; this is also indicated in **Error! Reference source not found.**, with pockets of negative axial velocity predicted along the axis just after the diverging section, persisting up to 7x diameters downstream. A smaller region of reverse flow is also shown along the axis towards the back wall of the chamber. The geometry studied by Niyogi et al. featured a constant cross section axial port, and as such the backflow region was found to extend from the chamber back wall through to the observed exit; in the present case the variation in cross sectional area in the axial port is predicted to interrupt the formation of a backflow region. Comparing **Error! Reference source not found.a** & **Error! Reference source not found.b**, the flow pattern predictions show little change in general behaviour with increasing throat velocity. The non-dimensional numerical values remain consistent as throat velocity increases, reflecting a linear increase in the absolute swirl velocity with increasing throat velocity. Similarly, the absolute strength of the reverse flow core increases linearly with increasing throat velocity. The non-dimensional values shown in Figure 9c at the larger scale factor of 2 are also consistent with the smaller device scale, suggesting that the general flow structure, in terms of the strength of the swirl component and reverse flow core, remains a linear function of the characteristic throat velocity as the device scale increases.

The normalized tangential velocity profiles through the midplane of the chamber are presented in **Error! Reference source not found.**a at each device scale. The predictions show the smallest device (A) to exhibit the lowest maximum tangential velocities, with a relatively large increase in maximum swirl ratio predicted with a doubling of device scale to device B. As further highlighted in **Error! Reference source not found.**b, the maximum values are then indicated to plateau with increasing scale beyond device C ($d_t = 24$, scaling factor = 4).

Device pressure recovery

The evolution of the absolute static pressure along the axis of the small vortex device A is illustrated in Figure 11. As throat velocity increases, the minimum pressure is predicted to reach the saturated vapour pressure at a throat velocity of 2.5 m/s. This identifies the cavitation inception point, beyond which the low pressure vapour core continues to extend along the axis with increasing flow rate. Above throat velocities of 3 m/s, the extent of the low pressure core is found to be limited by the increase in cross section, and the majority of the pressure recovery in the axial direction is found to occur over a relatively short distance either side of this expansion point (between $z = 3 \times$ chamber height, H and $z = 4 \times H$). The influence of device scale is also included in Figure 11b, with the predictions showing a marginal increase in the axial extent of the low pressure region for device B (2x scale); this can be attributed to a rise in overall pressure loss with increasing device scale, as determined experimentally between devices A & B (**Error! Reference source not found.**).

Unlike linear flow devices, such as venturi and orifice type cavitation devices, the axial pressure recovery is not the dominant pressure gradient; Figure 12a presents the radial pressure distribution at a series of axial locations through the device, with the corresponding axial pressure gradient provided in Figure 12b for comparison.

The radial pressure gradient is found to remain relatively constant up until an axial distance equal to $3 \times H$, again just prior to the increase in cross section. Comparing this to the maximum axial pressure gradient shows the radial gradient to be higher by a factor of 3. Smaller cavities will tend to be transported radially outwards from the low vapour core to the outer wall, and may therefore experience a sharp rise in static pressure in the axial port of up to $30 \times p_v$, which equates to $2 \times$ the outlet / fully recovered pressure. Contrastingly, in linear flow

devices, the maximum pressure gradient experienced by individual cavities is limited to the difference in vapour pressure and the device outlet pressure. Other research in this field has found that increasing the rate of pressure recovery has a positive effect on device performance in terms of increasing the intensity of cavitation bubble collapse. Particularly relevant is a study by Soyama et al.⁶⁶, who found that increasing the back pressure in a venturi cavitation reactor by a factor of 2 resulted in a recorded increase in acoustic power of two orders of magnitude. Capocelli et al.⁶⁷ have also studied the effect of increasing back pressure in a venturi cavitation reactor on the degradation of p-Nitrophenol in water. For the same overall device pressure drop, an increase in back pressure of the order of 0.4bar was reported to deliver an appreciable improvement in degradation performance, with recorded concentration ratio found to reduce by 15%. The rapid pressure recovery attained in the vortex unit axial port will therefore in principle contribute to achieving more intense cavity collapse, and by utilizing the radial pressure gradient the performance of the vortex device may be optimized for practical applications. The effect of these predicted pressure gradients on the pressure histories experienced by individual cavities is explored further in the “Discrete cavity trajectories” section.

Phase change predictions

The corresponding predicted vapour volumes for device A are given for two different flow rates in **Error! Reference source not found.a**, showing the evolution of the gas & vapour in the vortex core of the unit. The key feature of the two-phase flow in the vortex device is the cavitating vapour core, which is constrained to the centre of the device, away from the outer walls of the axial port. The extent of the vapour core is restricted to the portion of the axial chamber between the rear wall and the increase in cross section area, owing to the rapid recovery in pressure at this location. **Error! Reference source not found.b** shows the gas hold up, ε_g , as a function of throat velocity at each device scale, where ε_g is the ratio of gas volume to total reactor volume. The predictions indicate a plateau beyond throat velocities of 4.0 m/s, which corresponds to the cavitating vapour core reaching the area change in the axial port. With increasing size, corresponding to the slight increase in extent of the low pressure region, there is an accompanying marginal increase in gas hold up. In general, the results suggest that the cavitation generation rate is a simple function of device flow rate, which in this reactor geometry is predicted to reach an asymptote at throat velocities around 4 m/s.

Predicted turbulent fluctuations

Individual generated cavities experience oscillatory expansion and compression as a result of the turbulent fluctuations in pressure experienced over their lifetimes. This turbulent pressure history dictates the final conditions in terms of peak temperatures and pressures at the point of collapse; as such it is vital to understand the magnitudes of the turbulence properties in and around the vortex core region. The turbulence kinetic energy in device A is given in **Error! Reference source not found.a**, normalized by the square of throat velocity, with the corresponding normalized turbulence eddy frequencies plotted in **Error! Reference source not found.b**. Turbulence kinetic energy magnitudes are highest in the axial port at the point of increase in cross sectional area. At this location there are high gradients of both swirl velocity and axial velocity, with a strong reverse flow along the central axis, all of which combine to create a highly turbulent region. As throat velocity is increased, the turbulent kinetic energy magnitudes show a proportional increase as expected, with the highest magnitudes observed at the change in cross section in the axial port. **Error! Reference source not found.b** shows the corresponding turbulence eddy frequency distributions, which indicate high values at the rear wall of the swirl chamber; for Device A at a throat velocity of 3 m/s, the absolute values at this location are of the order of 500 kHz. Immediately downstream of the change in cross section in the axial port, the absolute values of eddy frequency correspond to 5 kHz. The influence of device scale on turbulence properties is presented in **Error! Reference source not found..** As the contour plots indicate, some influence of device scale is evident; turbulent kinetic energy increases with throat velocity, with an additional increase in peak values of the order of 40% due to increasing size between the smallest and largest geometries studied. The trend in eddy frequency magnitudes with on the other hand shows an inversely proportional relationship to device scale. At the largest scale with throat diameter, d_t , of 48mm, the absolute values of eddy frequencies in the axial port reduce to 0.6 kHz. In terms of individual cavity dynamics behaviour, these turbulent fluctuations remain orders of magnitudes higher than other anticipated transient mechanisms, namely vortex precession, which as reported elsewhere is expected to be of the order of 60 Hz²⁸. In examining the transient, turbulent pressure fluctuations experienced by individual cavities, the computed 2D flow fields therefore form a suitable basis on which to perform more detailed discrete phase calculations; these are discussed in detail in the following section.

Discrete cavity trajectories

The overall reactor performance is governed by complex interactions of numerous phenomena, including the cavity generation rate, cavity size distributions, the turbulent pressures experience by the cavities, as well as cavity-cavity interactions. The predicted flow field results therefore form just one part of a very complex picture. Final cavity collapse conditions are currently determined via direct numerical simulation of the Rayleigh-Plesset equation (see for example ^{16,40}), which is beyond the scope of the present investigation. The presented CFD results do however offer a means to link cavity dynamics predictions at the micro-scale to realistic, turbulent flow data at reactor scale. This offers a route to optimize reactor designs with a view to creating the ideal conditions for maximum cavitation yield and cavity collapse intensity. It is instructive therefore to understand the typical instantaneous pressure histories of the generated cavities. In order to study individual cavity trajectories, a series of lagrangian calculations were performed on the solved continuous flow fields for each device. The instantaneous pressures experienced by the tracked cavities can be related to the mean flow & turbulence quantities; the amplitude of the pressure fluctuations, and the instantaneous pressure, p' can be described by the following relationship:

$$p' = p + \rho k \quad (22)$$

The frequency of oscillation can then be directly related to the turbulence eddy frequency, ω , using an expression of the following form:

$$p' = p + \rho k \sin(\omega t) \quad (23)$$

The trajectories were treated as point masses with density equal to the water vapour density (0.55 kg/m^3), assumed from a reference pressure of 800Pa and a temperature of 295°K.

Complex interactions take place between individual cavities, or groups of cavities, and the central cavitating core of the device. Owing to the large density difference and the presence of a radial pressure gradient, larger cavities will tend to travel towards the axis of the device, where they may accumulate and coalesce in the central core. Smaller generated cavities on the other hand will tend to approach the path of the liquid phase, and will thus

orbit the vortex core. At the point of area increase in the axial port, the high turbulence predictions, coupled with relatively strong reverse flow, will lead to unsteady break-up of the vortex core, producing highly complex interactions between the gas phase and local turbulent eddies. In the present work therefore some simplifying assumptions are required in order to set the starting point from which to initiate single cavity calculations. In this instance, using the solved eulerian multiphase results, the cavities were tracked from a starting point on surface of constant volume fraction equal to 1, considered to be the centre of the gas core. **Error! Reference source not found.** shows typical cavity trajectories for bubble sizes of 10, 100 & 1000 μm , tracked up to a maximum residence time of 0.01s. The predictions show cavity size to have a negligible influence on the computed trajectories over this range, and as such the median size of 100 μm was applied for subsequent calculations. An initial observation from **Error! Reference source not found.** is that the cavities tend to be entrained around the central vortex core, continuously flowing around the low-pressure core of the device up until the point of rapid pressure recovery in the axial port. Limited investigations of larger sizes showed that for bubbles of 500 μm and above, the cavities become trapped within the reverse flow central core, resulting in incomplete trajectory calculations.

The resulting instantaneous pressure histories are presented in **Error! Reference source not found.** for device A at different throat velocities. The graphs in each case show a selected sample of 10 trajectories in total, with the initial starting positions of each trajectory evenly spaced in the axial direction along the surface of the gas core, and each terminating after a total residence time of 0.01 s. The trajectory time-pressure histories show two oscillation mechanisms; a large amplitude fluctuation which exceeds the difference in outlet pressure (p_2) and vapour pressure (p_v), and superimposed on these large scale fluctuations a higher frequency, lower amplitude oscillation due to the local turbulence field. Due to the high radial pressure gradients around the vortex core (illustrated in the contour plots in **Error! Reference source not found.**), as the cavities swirl around the core they are continuously moving radially from the low pressure core at p_v into high pressure regions (of the order of $20 \times p_v$) as they are advected through the axial port. At higher throat velocities, the plots highlight a shift in frequency of the fluctuations in instantaneous pressure; both the large amplitude fluctuations, and also the lower amplitude, turbulence fluctuations show a clear increase with increasing throat velocities.

The instantaneous pressure fluctuations are presented at different device scales in **Error! Reference source not found.**, each at a common throat velocity, $u_t = 3.0$ m/s. The key trend shown in the predicted results is that the frequency of both large scale and small-scale turbulence oscillations show an obvious decrease as the device scale increases, reflecting the reduction in turbulence eddy frequency with increasing device scale. Also indicated by the time-pressure plots is an apparent reduction in amplitude of the instantaneous pressures with increasing scale (from approx. $20 \times p_v$ for device B, down to $12 \times p_v$ for device D).

The results presented here provide useful insights into key flow characteristics of cavitating flow in vortex devices, and highlight the unique steep radial pressure gradients generated by such high swirl flow devices. The results will also be useful to inform the design of systematic experiments to generate empirical data on the velocity and turbulence fields to act as a basis for further rigorous validation of computational models. Future work will focus in particular on the complex interactions between the central gas core and individual cavitating bubbles, or groups of bubbles. The simulation results presented here highlight a highly turbulent region where the fixed vortex core terminates in the axial port, and detailed flow visualization experiments are planned to build our understanding of the precise nature of the two-phase flow in this region. These experiments are being carried out and will be published separately. The developed 2D approach and models will also provide a sound and useful basis for extending the models to full 3D, time-resolved simulations of cavitating flows in vortex devices.

Conclusions

Multiphase computational fluid dynamics models of vortex cavitation devices were developed which successfully reproduce experimental trends in flow rate vs pressure drop. The resulting models were subsequently used to investigate the cavitation inception conditions, and thereafter the development of the two-phase flow and turbulence fields at successively increasing flow rates. This particular vortex device design exhibits two prominent flow features; firstly the strong swirling flow sets up a strong radial pressure gradient, creating a cavitating vortex core in the axial port. Secondly, this strong vortex is accompanied by a region of reversed flow along the axis of the device, which persists through the full length of the axial port. The vortex device thus differs from linear flow devices in two fundamental respects; unlike orifice or venturi devices,

cavitation is initiated in the fluid bulk as opposed to the solid surface of the restriction. As such, through the use of swirling flows it is possible to eliminate the damaging effects of clogging and erosion. Secondly, the strong radial pressure gradient in the cavitating region leads to individual cavities experiencing pressure rises of over 2x the device back pressure, whereas in linear flow devices the maximum pressure recovery experienced by cavities is limited to the device pressure recovery. Discrete cavity trajectory calculations highlight the large amplitudes in pressure recovery experienced by individual cavities in the micron size range. The results also highlight the key differences in hydrodynamic behaviour with device scale; principally that the turbulence fluctuation frequencies reduce as the device size is increased. This may have an effect on the resulting cavity dynamics, and therefore the collapse intensities achieved. This warrants further investigation, which will form the basis of future work in this area. There is therefore promising potential to harness the fundamental hydraulic features of vortex devices revealed in this work to produce optimised device designs which can realise the full potential of hydrodynamic cavitation for use in full scale industrial processes. The key findings of the present study are summarised as follows:

- The strong swirling flow is shown to generate a cavitating vortex core, which is restricted to the fluid bulk. This uniquely differs from cavitation in linear flow devices commonly adopted in the available literature, which tends to initiate and evolve on the surface of the flow restriction.
- The radial pressure gradient in the cavitating region is shown to dominate over the axial pressure gradient. Smaller cavities (of the order of 10 μm) therefore experience pressure recovery rates in excess of 2x the overall device pressure recovery.
- Cavity trajectories are predicted to flow around the vortex core as they are advected through the axial port, continuously moving from low pressure (p_v) to high pressure (up to 20 x p_v) due to the large local pressure gradients. These large amplitude fluctuations are combined with lower amplitude fluctuations due to local turbulence quantities.
- Turbulence kinetic energy in the cavitating region is predicted to be a linear function of velocity at all device scales (scaling factors up to 8). Turbulent eddy frequencies were found to be inversely proportional to device scale.

References

1. Hreiz R, Gentric C, Midoux N, Lainé R, Fünfschilling D. Hydrodynamics and velocity measurements in gas-liquid swirling flows in cylindrical cyclones. *Chem Eng Res Des.* 2014;92(11):2231-2246. doi:10.1016/j.cherd.2014.02.029.
2. Hreiz R, Gentric C, Midoux N. Numerical investigation of swirling flow in cylindrical cyclones. *Chem Eng Res Des.* 2011;89(12):2521-2539. doi:10.1016/j.cherd.2011.05.001.
3. De Wilde J, Broqueville, De A. Rotating Fluidized Beds in a Static Geometry: Experimental Proof of Concept. *AIChE J.* 2007;53(4):793-810. doi:10.1002/aic.
4. Volchkov EP, Dvornikov V, Lukashov, V.V., Abdrakhmanov RK. Investigation of the flow in the cortex chamber with centrifugal fluidizing bed with and without combustion. *Thermophys Aeromechanics.* 2013;20(6):663-668.
5. Niyogi K, Torregrosa MM, Marin GB, Heynderickx GJ, Shtern VN. On the mechanisms of secondary flows in a gas vortex unit. *AIChE J.* 2018;00(00). doi:10.1002/aic.16087.
6. Tesla N. Valvular Conduit. 1920.
7. Zobel R. Experiments on a Hydraulic Reversing Throat. *Mitt Hydraul Munich.* 1930;8(19):1-47.
8. Elkassabgi Y, Leon G De, Fetterly C, Ramos J, Texas A, Cunningham RB. *Vortex Diode Analysis and Testing for Fluoride Salt-Cooled High-Temperature Reactors.*; 2011.
9. Tippetts JR, Priestman GH. The Flow States of Liquid Through Two Series-Connected Diffusers Subject to

- 1 Cavitation. *ASME J Fluids Eng.* 1983;105(82):350-355.
- 2 10. Moholkar VS, Senthil Kumar P, Pandit AB. Hydrodynamic cavitation for sonochemical effects. *Ultrason*
3 *Sonochem.* 1999;6(1-2):53-65. doi:10.1016/S1350-4177(98)00030-3.
- 4 11. Suslick KS, Mdleleni MM, Ries JT. Chemistry induced by hydrodynamic cavitation. *J Am Chem Soc.*
5 1997;119(39):9303-9304. doi:10.1021/ja972171i.
- 6 12. Rajoriya S, Carpenter J, Saharan VK, Pandit AB. Hydrodynamic cavitation: An advanced oxidation process
7 for the degradation of bio-refractory pollutants. *Rev Chem Eng.* 2016;32(4):379-411. doi:10.1515/revce-
8 2015-0075.
- 9 13. Dular M, Griessler-Bulc T, Gutierrez-Aguirre I, et al. Use of hydrodynamic cavitation in (waste)water
10 treatment. *Ultrason Sonochem.* 2016;29:577-588. doi:10.1016/j.ultsonch.2015.10.010.
- 11 14. Ranade V V., Bhandari VM. *Industrial Wastewater Treatment, Recycling, and Reuse-Past, Present and*
12 *Future.* Elsevier Ltd.; 2014. doi:10.1016/B978-0-08-099968-5.00014-3.
- 13 15. Kalumuck KM, Chahine GL. the Use of Cavitating Jets To Oxidize Organic Compounds in Water. *Proc*
14 *FJIDSM ASME Fluids Eng Div Summer Meet June -, Washington, DC .* 1998;122(September 2000):1-10.
15 doi:10.1115/1.1286993.
- 16 16. Capocelli M, Prisciandaro M, Lancia A, Musmarra D. Hydrodynamic cavitation of p-nitrophenol: A
17 theoretical and experimental insight. *Chem Eng J.* 2014;254:1-8. doi:10.1016/j.cej.2014.05.102.
- 18 17. Suryawanshi PG, Bhandari VM, Sorokhaibam G, Ruparelia JP, Ranade V V. Solvent Degradation Studies
19 Using Hydrodynamic Cavitation. 2018;37(1). doi:10.1002/ep.12674.
- 20 18. Gaikwad V, Ranade VV. Disinfection of Water Using Vortex Diode as Hydrodynamic Cavitation Reactor.
21 *Asian J Chem.* 2016;28(8):1867-1870.
- 22 19. Suryawanshi NB, Bhandari VM, Sorokhaibam LG, Ranade V V. A Non-catalytic Deep Desulphurization
23 Process using Hydrodynamic Cavitation. *Sci Rep.* 2016;6(September):1-8. doi:10.1038/srep33021.
- 24 20. Maddikeri GL, Gogate PR, Pandit AB. Intensified synthesis of biodiesel using hydrodynamic cavitation
25 reactors based on the interesterification of waste cooking oil. *Fuel.* 2014;137:285-292.
26 doi:10.1016/j.fuel.2014.08.013.
- 27 21. Kelkar MA, Gogate PR, Pandit AB. Intensification of esterification of acids for synthesis of biodiesel using
28 acoustic and hydrodynamic cavitation. *Ultrason Sonochem.* 2008;15(3):188-194.
29 doi:10.1016/j.ultsonch.2007.04.003.

22. Terán Hilaes R, Ramos L, da Silva SS, Dragone G, Mussatto SI, Santos JC dos. Hydrodynamic cavitation as a strategy to enhance the efficiency of lignocellulosic biomass pretreatment. *Crit Rev Biotechnol*. 2017;0(0):1-11. doi:10.1080/07388551.2017.1369932.
23. Nakashima K, Ebi Y, Shibasaki-Kitakawa N, Soyama H, Yonemoto T. Hydrodynamic Cavitation Reactor for Efficient Pretreatment of Lignocellulosic Biomass. *Ind Eng Chem Res*. 2016;55(7):1866-1871. doi:10.1021/acs.iecr.5b04375.
24. Albanese L, Ciriminna R, Meneguzzo F, Pagliaro M. Beer-brewing powered by controlled hydrodynamic cavitation: Theory and real-scale experiments. *J Clean Prod*. 2017;142:1457-1470. doi:10.1016/j.jclepro.2016.11.162.
25. Carpenter J, George S, Saharan VK. Low pressure hydrodynamic cavitating device for producing highly stable oil in water emulsion: Effect of geometry and cavitation number. *Chem Eng Process Process Intensif*. 2017;116(2016):97-104. doi:10.1016/j.cep.2017.02.013.
26. Simpson A, Ranade V V. Modelling of Hydrodynamic Cavitation with Orifice: Influence of different orifice designs. *Chem Eng Res Des*. 2018. doi:10.1016/j.cherd.2018.06.014.
27. Simpson A, Ranade V V. Modeling Hydrodynamic Cavitation in Venturi: Influence of Venturi Configuration on Inception and Extent of Cavitation. *AIChE J*. 2019;65(1):421-433. doi:10.1002/aic.16411.
28. Pandare A, Ranade V V. Flow in vortex diodes. *Chem Eng Res Des*. 2015;102:274-285.
29. Yin J, Li J, Ma Y, Li H, Liu W, Wang D. Study on the Air Core Formation of a Gas–Liquid Separator. *J Fluids Eng*. 2015;137(9):091301. doi:10.1115/1.4030198.
30. Arndt REA, Arakeri VH, Higuchi H. Some observations of tip-vortex cavitation. *J Fluid Mech*. 1991;229:269-289. doi:10.1017/S0022112091003026.
31. Arndt R, Pennings P, Bosschers J, van Terwisga T. The singing vortex. *Interface Focus*. 2015;5(5):1-11. doi:10.1098/rsfs.2015.0025.
32. Ma J, Hsiao C-T, Chahine GL. A physics based multiscale modeling of cavitating flows. *Comput Fluids*. 2017;145:68-84. doi:10.1016/j.compfluid.2016.12.010.
33. Chen L, Zhang L, Peng X, Shao X. Influence of water quality on the tip vortex cavitation inception. *Phys Fluids*. 2019;31(2):023303. doi:10.1063/1.5053930.
34. Wilcox DC. *Turbulence Modeling for CFD*. DCW Industries Inc., La Canada, California; 1998.

35. Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J.* 1994;32.
36. Menter FR, Kuntz M, Langtry R. Ten Years of Industrial Experience with the SST Turbulence Model.pdf. *Turbul Heat Mass Transf.* 2003;4.
37. Ansys. Ansys Inc, FLUENT 17.0 Theory Guide. 2016.
38. Kato M, Launder B. The modeling of turbulent flow around stationary and vibrating square cylinders. *Ninth Symp Turbul Shear Flows.* 1993;(January 1993):10.4.1-10.4.6. doi:10.1007/s13398-014-0173-7.2.
39. Brennan CE. *Cavitation and Bubble Dynamics*. Oxford University Press: New York; 1995.
40. Alehossein H, Qin Z. Numerical analysis of Rayleigh-Plesset equation for cavitating water jets. *Int J Numer Meth Engng.* 2007;72:780-807.
41. Sauer J, Schnerr GH. Unsteady cavitating flow: a new cavitation model based on a modified front capturing method and bubble dynamics. *FEDSM'00, ASME Fluids Eng Summer Conf.* 2000:11-15.
42. Zwart PJ, Gerber AG, Belamri T. A Two-Phase Flow Model for Predicting Cavitation Dynamics. In: *Fifteenth International Conference on Multiphase Flow, Yokohama, Japan.* ; 2004.
43. Singhal AK, Athavale MM, Li H, Jiang Y. Mathematical Basis and Validation of the Full Cavitation Model. *J Fluids Eng.* 2002;124(3):617. doi:10.1115/1.1486223.
44. Arndt REA, Keller AP. Water Quality Effects on Cavitation Inception in a Trailing Vortex. *J Fluids Eng.* 1992;114(3):430-438. doi:10.1115/1.2910049.
45. Jiao L, Zhang PP, Chen CN, Yin JL, Wang LQ. Experimental study on the cavitation of vortex diode based on CFD. *IOP Conf Ser Earth Environ Sci.* 2012;15(PART 6). doi:10.1088/1755-1315/15/6/062058.
46. Reboud J-L, Stutz B, Coutier O. Two-phase flow structure of cavitation: Experiment and modelling of unsteady effects. *Thirteenth Int Symp cavitation.* 1998;2017(July 2014):203-208.
47. Coutier-Delgosha O, Fortes-Patella R, Reboud JL. Evaluation of the Turbulence Model Influence on the Numerical Simulations of Unsteady Cavitation. *J Fluids Eng.* 2003;125(1):38. doi:10.1115/1.1524584.
48. Charriere B, Decaix J, Goncalves E. A comparative study of cavitation models in a Venturi flow. *Eur J Mech B/Fluids.* 2015;49(PA):287-297. doi:10.1016/j.euromechflu.2014.10.003.
49. Charrière B, Goncalves E. Numerical investigation of periodic cavitation shedding in a Venturi. *Int J Heat Fluid Flow.* 2017;64:41-54. doi:10.1016/j.ijheatfluidflow.2017.01.011.
50. Ji B, Luo X, Wu Y, Peng X, Duan Y. Numerical analysis of unsteady cavitating turbulent flow and shedding

- horse-shoe vortex structure around a twisted hydrofoil. *Int J Multiph Flow*. 2013;51:33-43. doi:10.1016/j.ijmultiphaseflow.2012.11.008.
51. Zhang DS, Wang HY, Shi WD, Zhang GJ, Van Esch BPM. Numerical analysis of the unsteady behavior of cloud cavitation around a hydrofoil based on an improved filter-based model. *J Hydrodyn*. 2015;27(5):795-808. doi:10.1016/S1001-6058(15)60541-8.
52. Long X, Cheng H, Ji B, Arndt REA. Numerical investigation of attached cavitation shedding dynamics around the Clark-Y hydrofoil with the FBDCM and an integral method. *Ocean Eng*. 2017;137(December 2016):247-261. doi:10.1016/j.oceaneng.2017.03.054.
53. Coutier-Delgosha O, Fortes-Patella R, Reboud JL. Evaluation of the Turbulence Model Influence on the Numerical Simulations of Unsteady Cavitation. *J Fluids Eng*. 2003;125(1):38. doi:10.1115/1.1524584.
54. Chen Y, Lu CJ, Wu L. Modelling and computation of unsteady turbulent cavitation flows. *J Hydrodyn*. 2006;18(5):559-566. doi:10.1016/S1001-6058(06)60135-2.
55. Li D-Q, Grekula M, Lindell P. A modified SST k- ω turbulence model to predict the steady and unsteady sheet cavitation on 2D and 3D hydrofoils. *Int Symp Cavitation*. 2009;(107):1-13.
56. Goncalvs E. Numerical study of unsteady turbulent cavitating flows. *Eur J Mech B/Fluids*. 2011;30(1):26-40. doi:10.1016/j.euromechflu.2010.08.002.
57. Decaix J, Goncalves E. Time-dependent simulation of cavitating flow with k-l turbulence models. *Int J Numer Methods Fluids*. 2011;68:1053-1072. doi:10.1002/fld.2601.
58. Ji B, Luo X, Arndt REA, Wu Y. Numerical simulation of three dimensional cavitation shedding dynamics with special emphasis on cavitation – vortex interaction. *Ocean Eng*. 2014;87:64-77. doi:10.1016/j.oceaneng.2014.05.005.
59. Long X, Cheng H, Ji B, Arndt REA, Peng X. Large eddy simulation and Euler–Lagrangian coupling investigation of the transient cavitating turbulent flow around a twisted hydrofoil. *Int J Multiph Flow*. 2018;100:41-56. doi:10.1016/j.ijmultiphaseflow.2017.12.002.
60. Zaman E, Vakil A, Martinez M, Olson J. An Integral Criterion for Turbulence Suppression in Swirling Flows. *Canad*. 2018. doi:10.1002/cjce.23145.
61. Bardina, J.E., Huang, P.G., Coakley TJ. Turbulence modelling validation. In: *AIAA-1997-2121*. ; 1997.
62. Derksen JJ, Akker HEA Van Den. Simulation of Vortex Core Precession in a Reverse-Flow Cyclone. 2000;46(7):1317-1331.

63. Dinesh KKJR, Kirkpatrick MP. Computers & Fluids Study of jet precession , recirculation and vortex breakdown in turbulent swirling jets using LES. *Comput Fluids*. 2009;38(6):1232-1242. doi:10.1016/j.compfluid.2008.11.015.
64. Brinkhorst S, Lavante E von, Wendt G. Experimental and numerical investigation of the cavitation-induced choked flow in a herschel venturi-tube. *Flow Meas Instrum*. 2017;54(November 2016):56-67. doi:10.1016/j.flowmeasinst.2016.12.006.
65. Ashrafizadeh SM, Ghassemi H. Experimental and numerical investigation on the performance of small-sized cavitating venturis. *Flow Meas Instrum*. 2015;42:6-15. doi:10.1016/j.flowmeasinst.2014.12.007.
66. Soyama H, Hoshino J. Enhancing the aggressive intensity of hydrodynamic cavitation through a Venturi tube by increasing the pressure in the region where the bubbles collapse. *AIP Adv*. 2016;6(4). doi:10.1063/1.4947572.
67. Capocelli M, Prisciandaro M, Lancia A, Musmarra D. Cavitation reactor for advanced treatment of contaminated water: the effect of recovery pressure. *Desalin Water Treat*. 2015;55(12):3172-3177. doi:10.1080/19443994.2014.947779.

1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28

List of Figures

Figure 1: Vortex diode schematic..... 32

Figure 2: Vortex device geometry and operating conditions 33

Figure 3a: Computational grid & boundary condition recipe..... 34

Figure 3b: Mesh sensitivity study; swirl velocity 34

Figure 3c: Mesh sensitivity study; turbulent kinetic energy..... 34

Figure 4: Swirl velocity profiles in chamber mid-plane – turbulence model comparison 35

Figure 5: Two-dimensional approximation of vortex unit: velocity profiles at the inlet..... 36

Figure 6a: Predicted swirl ratio in chamber mid-plane; 2D vs 3D model 37

Figure 6b: Predicted static pressure in chamber mid-plane; 2D vs 3D model 37

Figure 7: Measured & predicted Δp with increasing throat velocity, u_t 38

Figure 8: Predicted cavitation inception versus throat velocity, u_t 39

Figure 9a: Device A | $u_t = 2.3$ m/s | $Re = 13,775$ 40

Figure 9b: Device A | $u_t = 3.5$ m/s | $Re = 20,960$ 40

Figure 9c: Device B | $u_t = 3.5$ m/s | $Re = 41,925$ 40

Figure 10a: Chamber swirl velocity profiles at different device scale 42

Figure 10b: Maximum swirl velocities vs Reynolds number at different device scales..... 42

Figure 11a: Static pressure contours, Diode A 43

1	Figure 11b: Static pressure distributions along device axis.....	43
2	Figure 12: Predicted static pressure distributions (Diode B, $u_t = 3.3$ m/s).....	44
3	Figure 13a: Computed volume fractions, Device A.....	45
4	Figure 13b: Gas hold up, ϵ_g	45
5	Figure 14a: Turbulent kinetic energy, device A.....	46
6	Figure 14b: Turbulence eddy frequencies, device A.....	46
7	Figure 15a: Turbulent kinetic energy, Scale effect.....	47
8	Figure 15b: Turbulence eddy frequency, Scale effect.....	47
9	Figure 16: Cavity size influence on predicted trajectories, Device D.....	48
10	Figure 17: Predicted cavity trajectories, device A.....	49
11	Figure 18: Predicted cavity trajectories, devices B, C & D.....	50
12		
13		
14		
15		
16		
17		
18		
19		
20		
21		
22		
23		
24		
25		
26		
27

Figure 1: Vortex diode schematic

	Scaling factor	d_t [mm]	Q [L/min]	Re ($\rho u_t H / \mu$)
Device A	1	6.0	2.4 - 6.0	8,600 – 20,962
Device B	2	12.0	10.0 - 23.8	17,967 – 41,924
Device C	4	24.0	62.0 – 98.0	55,090 – 86,227
Device D	8	48.0	250 - 391	110,180 – 172,454

Figure 2: Vortex device geometry and operating conditions

Figure 3a: Computational grid & boundary condition recipe

Error! Reference source not found.b: Mesh sensitivity study; swirl velocity **Error! Reference source not found.**c: Mesh sensitivity study; turbulent kinetic energy

Figure 4

Figure 5: Swirl velocity profiles in chamber mid-plane – turbulence model comparison

Figure 6: Two-dimensional approximation of vortex unit: velocity profiles at the inlet

Figure 7a: Predicted swirl ratio in chamber mid-plane; 2D vs 3D model

Figure 7b: Predicted static pressure in chamber mid-plane; 2D vs 3D model

Figure 8: Measured & predicted Δp with increasing throat velocity, u_t

Figure 9: Predicted cavitation inception versus throat velocity, u_t

Figure 10a: Device A | $u_t = 2.3 \text{ m/s}$ | $Re = 13,775$

Error! Reference source not found.b: Device A | $u_t = 3.5 \text{ m/s}$ | $Re = 20,960$

Error! Reference source not found.c: **Device B** | $u_t = 3.5 \text{ m/s}$ | $Re = 41,925$

Figure 11a: Chamber swirl velocity profiles at different device scale

Error! Reference source not found.b: Maximum swirl velocities vs Reynolds number at different device scales

Figure 12a: Static pressure contours, Diode A

Error! Reference source not found.b: Static pressure distributions along device axis

1

2

(a) Static pressure ratio, p/p_v

3

(b) Distance along axis, z [mm]

4

Figure 13: Predicted static pressure distributions (Diode B, $u_t = 3.3$ m/s)

6

7

8

9

10

11

12

13

14

1

(a) Computed volume fractions, Device A

(b) Gas hold up, ϵ_g

Figure 14: Predicted gas hold-up

Figure 15a: Turbulent kinetic energy, device A

Error! Reference source not found.b: Turbulence eddy frequencies, device A

Figure 16a: Turbulent kinetic energy, Scale effect

Error! Reference source not found.b: Turbulence eddy frequency, scale effect

1

2

3

Figure 17: Cavity size influence on predicted trajectories, Device D

4

5

6

7

8

9

10

11

12

13

14

15

16

17

18

19

20

21

22

1

Figure 18: Predicted cavity trajectories, device A

1

Figure 19: Predicted cavity trajectories, devices B, C & D

1
2