



## City Research Online

### City, University of London Institutional Repository

---

**Citation:** Koukouvinis, P., Mitroglou, N., Gavaises, M., Lorenzi, M. & Santini, M. (2017). Quantitative predictions of cavitation presence and erosion-prone locations in a high-pressure cavitation test rig. *Journal of Fluid Mechanics*, 819, pp. 21-57. doi: 10.1017/jfm.2017.156

This is the accepted version of the paper.

This version of the publication may differ from the final published version.

---

**Permanent repository link:** <https://openaccess.city.ac.uk/id/eprint/18411/>

**Link to published version:** <https://doi.org/10.1017/jfm.2017.156>

**Copyright:** City Research Online aims to make research outputs of City, University of London available to a wider audience. Copyright and Moral Rights remain with the author(s) and/or copyright holders. URLs from City Research Online may be freely distributed and linked to.

**Reuse:** Copies of full items can be used for personal research or study, educational, or not-for-profit purposes without prior permission or charge. Provided that the authors, title and full bibliographic details are credited, a hyperlink and/or URL is given for the original metadata page and the content is not changed in any way.

---

City Research Online:

<http://openaccess.city.ac.uk/>

[publications@city.ac.uk](mailto:publications@city.ac.uk)

---

# Quantitative predictions of cavitation presence and erosion-prone locations in a high pressure cavitation test rig

Phoevos Koukouvinis<sup>†,1</sup>, Nicholas Mitroglou<sup>1</sup>, Manolis Gavaises<sup>1</sup>, Massimo Lorenzi<sup>1</sup>, Maurizio Santini<sup>2</sup>

<sup>1</sup>School of Mathematics Computer Science and Engineering, City University London, London, EC1V 0HB, UK.

<sup>2</sup>Department of Engineering and Applied Sciences, University of Bergamo, Bergamo, 24129, Italy.

Experiments and numerical simulations of cavitating flow inside a single-orifice nozzle are presented. The orifice is part of a closed flow circuit with Diesel fuel as working fluid, designed to replicate the main flow pattern observed in high pressure Diesel injector nozzles. The focus of the present investigation is on cavitation structures appearing inside the orifice, their interaction with turbulence and the induced material erosion. Experimental investigations include high-speed shadowgraphy visualisation, X-ray micro-computed tomography (micro-CT) of time-averaged volumetric cavitation distribution inside the orifice, as well as pressure and flow rate measurements. The highly transient flow features that are taking place, such as cavity shedding, collapse and vortex cavitation (also known as "string cavitation"), have become evident from high-speed images. Additionally, micro-CT enabled the reconstruction of the orifice surface, which provided locations of cavitation erosion sites developed after sufficient operation time. The measurements are used to validate the presented numerical model, which is based on the numerical solution of the Navier-Stokes equation, taking into account compressibility of both the liquid and liquid-vapour mixture. Phase change is accounted with a newly developed mass transfer rate model, capable of accurately predicting the collapse of vaporous structures. Turbulence is modelled using Detached Eddy Simulation and unsteady features like cavitating vortices and cavity shedding are observed and discussed. The numerical results show agreement within validation uncertainty against the obtained measurements.

## 1. Introduction

Cavitation is the phenomenon of vaporous pocket formation inside liquids, due to a drop in the local static pressure (Brennen 1995); it commonly appears in hydraulic pumps, turbines, propellers, rudders (Li 2000), in high pressure fuel injection systems (Egler, Giersch et al. 2010) and even heart valves (Kini, Bachmann et al. 2000). Cavitation causes flow blockage and choking, while the collapse of cavitation structures may lead to cavitation erosion damage with detrimental consequences on the reliability and maintenance of relevant devices. Apart from the aforementioned effects, in the field of fuel injection systems, cavitation plays a detrimental role in jet formation, stability and atomization (Sou, Hosokawa et al. 2007), affecting the combustion process and finally the performance and emissions of modern engines. For all the previous reasons, significant effort has been put in the investigation of cavitating flows and prediction/quantification of its related effects both with experimental and numerical/simulation techniques, in order to prevent negative aspects or harness any positive potential (if applicable).

Cavitating flows have been extensively examined utilising high-speed shadowgraphy. An excellent summary of optical visualization techniques, involving Schlieren, shadowgraphy and interferometry in the field of cavitation is discussed in the work of Mauger, Méès et al. (2012) in a simplified, high pressure 2D channel. In the field of Diesel injection systems, Mitroglou, McLorn et al. (2014) studied the flow in a real size Diesel injector, whose metallic tip was removed and replaced with a transparent one, to conduct visualization studies. The study involved the realistic operation of the injector, under a pressure pulse up to 600 bar, while observing cavitation formation in the sac and nozzles of the

48 injector. The authors focused on unsteady cavitating features and especially the formation of  
49 cavitating vortices (which are also termed as "string cavitation" in the fuel injection industry, due to  
50 their rope-like appearance), known to increase spray cone angles (Zigan, Schmitz et al. 2012).  
51 Similarly, Hult, Simmank et al. (2016) studied the cavitating flow in a marine diesel injector using  
52 high-speed shadowgraphy, among other techniques, for understanding the near nozzle jet behaviour  
53 and how this is affected by rolling vortex streets, formed at the edge of the orifice entrance. In  
54 general, such optical techniques are not limited in Diesel injectors or fuel injection systems; for  
55 example, Pennings, Bosschers et al. (2015) used shadowgraphy for studying the formation,  
56 development and oscillation of a cavitating vortex emerging from the tip of a hydrofoil. Žnidarčič,  
57 Mettin et al. (2014) employed shadowgraphy to determine the extents of cavitation presence in the  
58 vicinity of an ultrasonic horn and compared the results with numerical simulations. At fundamental  
59 bubble dynamics, Lindau & Lauterborn (2003) studied the behaviour of laser bubble generation,  
60 growth and collapse in the vicinity of wall surfaces with high-speed shadowgraphy.

61 Despite the undoubted significance of optical methods as experimental techniques in  
62 understanding cavitation structures, their main drawback is that cavitation clouds, as observed from  
63 elastic scattering, do not correlate with the actual density of the liquid/vapour mixture. Additionally,  
64 cavitation may obstruct the optical path, preventing further observation of flow features. For this  
65 reason, quantitative experimental techniques have been developed and employed to study and gain  
66 insight in the density distribution of cavitating flows. Such techniques rely on the attenuation of  
67 powerful photon or particle beams due to the presence of sample material along their path. Notable  
68 examples are neutron imaging (IAEA 2008) and X-ray imaging methods, which will be discussed  
69 later on.

70 Two dimensional X-ray radiography of cavitating flows in Diesel injector orifices has been  
71 reported in simplified or even more complicated geometries. Duke, Kastengren et al. (2014) studied a  
72 simple, axis-symmetric nozzle of 500 $\mu\text{m}$  diameter, using a powerful synchrotron X-ray source and  
73 compared the 2D radiography projection of the vapour distribution with simulation results. Battistoni,  
74 Duke et al. (2015) employed a similar simplified geometry, but studied the effect of non-condensable  
75 gases (like dissolved air in fuel) and their interaction with cavitation. Moon, Liu et al. (2010)  
76 performed ultrafast X-ray phase contrast imaging for the identification of cavitation and spray  
77 boundaries emerging from a realistic multi-hole Diesel injector. Further notable examples of work of  
78 similar nature, but on different application fields, involve the study of cavitating flow around a  
79 NACA009 hydrofoil (Ganesh, Mākiharju et al. 2016) using high-speed X-ray radiography, where  
80 compressibility effects during the collapse of attached cavitation clouds (condensation shocks) have  
81 been identified. Such effects are difficult, if not impossible, to be identified with traditional optical  
82 measurements since the density field is not directly reproducible. In the same spirit, Sun, Ganesh et al.  
83 (2015) studied the cavitating flow emerging from vortex shedding in the wake of a cylinder at  
84 conditions ranging from cavitation inception to supercavitation regimes. X-ray imaging can provide  
85 valuable information in more industrial cases as well; Duplaa, Coutier-Delgosha et al. (2013)  
86 examined cavitation formation around the impeller of a pump during fast start-up operation and  
87 correlated the density variations to pressure-time evolution.

88 While undeniably 2D X-ray radiography can shed light on the instantaneous density distribution of  
89 the flow, it cannot provide information on the exact 3D cavity shape. For this reason, Bauer, Chaves  
90 et al. (2012) examined the flow inside an axis-symmetric nozzle at cavitation numbers ranging from  
91 inception to supercavitation with X-ray Computed Tomography (CT). In their investigation the  
92 working fluid was water and a medical scanner was utilized for the measurements. The CT scans  
93 enabled a 3D volumetric reconstruction of the vapour cloud inside the orifice, which has been used for  
94 validation of CFD codes that have been previously developed (Giannadakis, Gavaises et al. 2008).

95 Simulation of cavitating flows is of high complexity due to the large density ratios involved and

96 flow unsteadiness. Andriotis, Gavaises et al. (2008) showed that turbulence can play a detrimental  
97 role in the flow development. Cavitating vortices, or "string cavitation", were identified to affect the  
98 flow inside injector nozzle holes and the emerging jet. In more recent works, the effect of accurate  
99 turbulence description is highlighted in the work of Edelbauer, Struel et al. (2016). The authors  
100 focused on fundamental investigation in microchannel flows at high pressures, showing the situational  
101 applicability of RANS/URANS models in describing cavitation effects at cavitation inception  
102 conditions. More recently Örley, Hickel et al. (2016) have achieved an impressive simulation of a  
103 complete 9-hole Diesel injector, including the injection in air environment, aiming to study the  
104 influence of cavitation to the emerging jets. The work of Koukouvinis, Gavaises et al. (2016) focused  
105 instead on the prediction of cavitation inside industrial Diesel injector designs operating at similar  
106 conditions, while tracking the collapse of cavitation structures for correlation with the erosion  
107 development that was found from X-ray topographies of used injectors.

108 Despite the numerous examples of numerical work, validation of relevant simulation models is  
109 not straightforward. Commonly it is done by comparing macroscopic quantities, like flow rate, or  
110 examining the unsteady patterns of cavitation structures in comparison to high-speed imaging. The  
111 present investigation aims to examine the cavitating flow in Diesel injector-like geometries on a  
112 fundamental level. Similar to the work of Bauer, Chaves et al. (2012), the cavitating flow inside a  
113 cylindrical nozzle is investigated. However, in the present work the orifice is placed asymmetrically  
114 in respect to upstream elements (e.g. the throttling needle), forcing the flow to be non axis-symmetric,  
115 forming a configuration that resembles closer the flow path of a Diesel injector. Despite the apparent  
116 simplicity of the experimental configuration and the stationary needle, notable flow effects occurring  
117 in real-world fuel injection systems, such as the formation of cavitating vortices and cavitation  
118 induced surface erosion, have been observed. Understanding these effects is crucial for the  
119 performance of a fuel injection system, since unsteady cavitating vortices affect jet atomization and  
120 spray patterns (Andriotis, Gavaises et al. 2008; Giannadakis, Gavaises et al. 2008) and consequently  
121 engine performance, whereas material erosion can lead to performance deterioration, very high  
122 maintenance costs and reliability issues. It is highlighted that these effects are not limited in fuel  
123 systems, but have a general interest from a broad fluid dynamics perspective. The dynamics of  
124 cavitating vortices are discussed extensively in Franc & Michel (2005), who showed that circulation  
125 leads to a rebounding behaviour of a cavitating vortex, even in the absence of non-condensable gases,  
126 due to conservation of angular momentum. In real life, cavitating vortices occur in a wide range of  
127 engineering applications, such as at the tips of hydrofoils (Arndt, Arakeri et al. 1991; Pennings,  
128 Bosschers et al. 2015) or propeller blades (Duttweiler & Brennen 2002) and contribute to the erosion  
129 of ship propellers and rudders (Carlton 2012) or the operation of hydro-turbines (Decaix, Balarac et  
130 al. 2015). The aforementioned non-exhaustive list of references indicate the importance and need for  
131 an accurate and validated numerical model, capable of predicting cavitating vortex and cavitation  
132 erosion effects in a quantitative way.

133 The focus of the present paper is on the unsteady cavitating phenomena occurring inside the  
134 orifice, with emphasis on the formation of cavitating vortices, interaction of cavitation with turbulence  
135 and the mechanism that causes erosion in affected regions. The flow is studied experimentally and  
136 numerically. From the experimental side, high-speed shadowgraphy and time-averaged X-ray micro-  
137 CT scanning are employed. Numerical simulations complement the experimental results, providing  
138 additional insight, due to the complexity of the flow field which inevitably obstructs detailed  
139 observations in the whole flow passage. The objective of the present study is to obtain quantitative  
140 experimental data on both cavitation distribution and cavitation erosion sites, for a given orifice  
141 geometry (see also supplementary material) and operating conditions. These data may be further used  
142 for quantitative comparisons with numerical simulations, hence serving as a benchmark/validation  
143 case. Moreover, a cavitation model is discussed and validated against the aforementioned data,

144 showing the main flow mechanisms and the outcome of the modelling approach. The novelty of the  
145 current work is the combined presentation of new experimental results, including 3-D density and  
146 cavitation erosion measurement, accompanied by numerical simulations of the relevant phenomena  
147 occurring in this flow orifice at low and high cavitation numbers. In particular, X-ray micro-CT  
148 cavitation volume fraction measurements, high-speed flow visualisation and cavitation erosion areas  
149 are used for the detailed quantitative validation of the proposed cavitation model. The latter has been  
150 used in recent studies in predicting erosion locations in high pressure fuel injectors, see Koukouvinis,  
151 Gavaises et al. (2016).

152 The paper is organized as follows: initially, the experimental test rig is described, along with  
153 details of the actual orifice geometry and operating conditions. Next, the numerical models used for  
154 the simulations, fluid properties and simulation set-up are discussed. Then, flow details from  
155 simulations are provided, such as visualization of cavitation, flow field and vortical structures. Next,  
156 the experimental and numerical results are compared, both in qualitative (e.g. vapour fraction  
157 isosurface shape) and quantitative terms (e.g. mass flow rate, vapour cavity extents, etc.). Finally, a  
158 short discussion of the results is made and main conclusions are summarized, highlighting the most  
159 important findings.

## 160 **2. Experiment and simulation set-up**

161 The geometry employed in the present study resembles the features of a Diesel injector orifice and  
162 more specifically, it was designed to resemble the asymmetric flow of Valve Covered Orifice (VCO)  
163 or mini-sac type nozzles, inspired from the work of Reid, Hargrave et al. (2010).

### 164 *2.1. The experimental test rig*

165 The experimental test rig consists of a closed loop hydraulic circuit (see also Figure 1), comprised  
166 of a high pressure hydraulic unit that circulates the flow, valves and pressure regulators to regulate  
167 flow rate and upstream/downstream pressure, the orifice test section and a shell and tube water-cooled  
168 heat exchanger to maintain the Diesel fuel temperature constant. The hydraulic circuit also includes  
169 filters for controlling Diesel quality, a fuel tank where fuel rests at low velocities, allowing any  
170 bubbles to escape, and the necessary piping. Pressure, temperature and flow rate are measured with  
171 appropriate transducers, connected to a computer via an Analog to Digital Converter (ADC) card for  
172 data logging.

173 A view of the cavitation test section is shown in Figure 2 (a-c). Following the flow from the high  
174 pressure inlet towards the low pressure outlet, the test section consists of a high pressure inlet socket,  
175 which is split to three feed lines, Figure 2 (a). This arrangement was used to accommodate adjustable  
176 needle lift for controlling the flow rate with higher precision. The feed lines converge to a funnel  
177 shaped passage that further downstream becomes annular; inside this passage the needle is located.  
178 Note that, even though the needle is cylindrical upstream the funnel, it becomes asymmetrical near the  
179 orifice entrance, due to a bevel cut at its tip and the support structure that is in contact with the  
180 passage walls, blocking part of the flow. The orifice is placed off-axis, to further promote flow  
181 asymmetry, on an interchangeable part that includes also the collector volume. Downstream the  
182 collector are four low-pressure outlet pipes, that direct the flow to the return of the hydraulic unit.

183 The interchangeable part, that includes the orifice itself, allows for different types of flow  
184 investigations to be conducted. For visualization studies, the orifice was made of a transparent acrylic  
185 resin piece with rectangular collector, see Figure 2 (b), whereas for the X-ray investigation the orifice  
186 was drilled on a PolyEther Ether Ketone (PEEK) piece with cylindrical collector (not shown here).  
187 The idealized orifice examined here is a cylinder of 9.5mm length and 3mm diameter.

190  
191

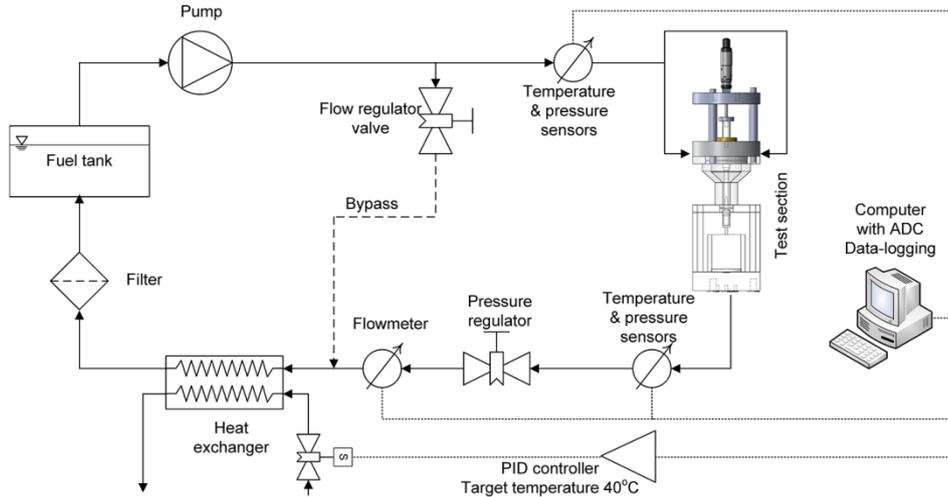


FIGURE 1. Single-line diagram of the hydraulic rig circuit.

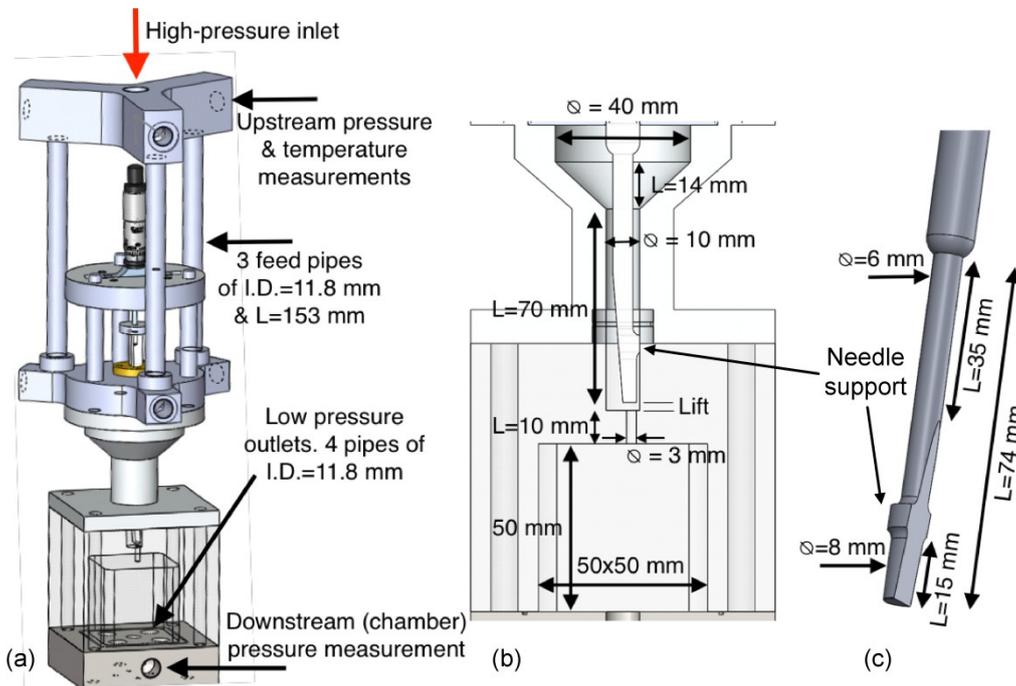


FIGURE 2. (a) Drawing of the experimental test section (b) Focused view in the area of interest with main dimensions (c) drawing of the needle and relevant dimensions.

192  
193  
194  
195

The two cases that have been examined experimentally and numerically are outlined in table 1; two common cavitation number definitions are provided according to (2.1):

198

$$Cn = \frac{1}{\sigma} = \frac{p_{up} - p_{down}}{p_{down} - p_v} \quad (2.1)$$

199  
200

where  $p_{up}$  is the upstream pressure,  $p_{down}$  the downstream pressure and  $p_v$  the vapour (or saturation) pressure. Reynolds number is defined as:

201

$$Re = \frac{u_{avg} D \rho_l}{\mu_l} \quad (2.2)$$

202  
203

where  $u_{avg}$  is the average velocity in the orifice cross-section, assuming it is fully occupied by liquid,  $D$  is the orifice diameter,  $\rho_l$  is the liquid density and  $\mu_l$  liquid viscosity.

204  
205

Both cases correspond to the same needle lift of 1mm from the fully closed position and to two different cavitation numbers, at  $Cn = 2.18$  and  $Cn = 1.5$ , that will be referred to as high and low

206 cavitation intensity, respectively. For the whole duration of the experiments, fuel temperature was  
 207 controlled at  $40^{\circ}\pm 0.5^{\circ}$  C. Errors reported in table 1 are based on systematic and random measurement  
 208 errors of the pressure and flow transducers and error propagation analysis for the derived values (e.g.  
 209 cavitation number, Reynolds number), see Bevington & Robinson (2003). The accuracy of the  
 210 pressure transducers is 0.4 bar upstream, 0.17 bar downstream and their temporal resolution is 1ms.  
 211 The accuracy of the flow meter is 0.1 lt/min.

	Upstream pressure	Downstream pressure	Cavitation number		Reynolds number	Average flow velocity	Volume flow rate
Case	$p_{up}$ (bar)	$p_{down}$ (bar)	$Cn$ (-)	$\sigma$ (-)	$Re$ (-)	$u_{av}$ (m/s)	$\dot{Q}$ (lt/s)
1	$43.1 \pm 0.4$	$17.3 \pm 0.2$	$1.50 \pm 0.04$	$0.67 \pm 0.02$	$66000 \pm 4500$	$56 \pm 3.4$	$0.397 \pm 0.002$
2	$55.0 \pm 0.4$	$17.5 \pm 0.2$	$2.18 \pm 0.04$	$0.46 \pm 0.01$	$77000 \pm 5300$	$65 \pm 4.0$	$0.460 \pm 0.002$

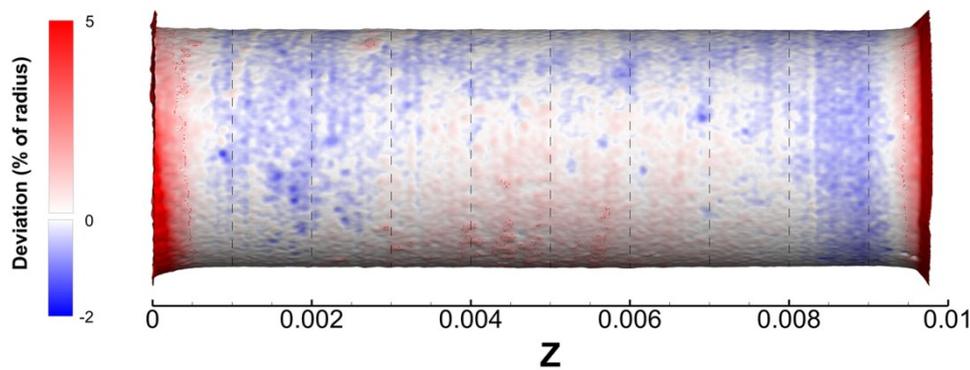
TABLE 1. Operating conditions for two of the cases examined with experiments and simulations.  
 All cases are at 1mm needle lift. Average velocity is estimated inside the orifice cross-section,  
 assuming the whole cross-section is occupied by liquid.

212  
 213 Shadowgraphy was employed as a visualization technique for capturing still images of the  
 214 transient cavitation structures emerging inside the orifice. The experimental setup of the image  
 215 acquisition methodology is comprised of a halogen floodlight providing white band light, which was  
 216 subsequently focused by a Fresnel lens on the area of interest. The placement of a CCD camera at the  
 217 opposite side of the device allowed for side-view images of the arising vapour cloud to be taken. A  
 218 high-speed Photron SA-X camera was employed, adjusted to capture 80000 frames per second (fps) at  
 219 a resolution of  $896 \times 264$  pixels. The shutter speed was set to  $0.37 \mu s$  to increase image clarity and  
 220 “freeze” the high velocity flow phenomena inside the orifice. Since the overall length of the  
 221 visualized orifice region is equal to 10 mm and 646 out of the total 896 pixels are employed for its  
 222 discretization, the visualization resolution is equal to  $15.47 \mu m/pixel$ . The resolution of the  
 223 visualization technique is in the order of the manufacturing uncertainty associated with the orifice  
 224 dimensions, while the size of cloud cavitation and coherent cavitating vortices is much larger and  
 225 comparable to the orifice diameter ( $\sim 3.0$  mm). Hence, the spatial resolution is sufficient for properly  
 226 illustrating the topology of the two-phase flow in the area of interest.

227 The prototype micro-computed tomography facility (located at University of Bergamo) employed  
 228 for cavitation volume fraction measurements and orifice surface deformation consists of a 160 kV (at  
 229  $400 \mu A$ ) open type cone-beam x-ray source, an air-bearing direct drive rotary stage and a  $1944 \times 1536$   
 230 pixels flat panel CMOS detector with a custom matched scintillator plate. The spatial resolution was  
 231 adapted to provide an isotropic voxel size of  $15 \mu m$  (in the reconstructed volume). For each of the 600  
 232 steps required for a full rotation of the test item, five projections were acquired and averaged to  
 233 improve the signal to noise ratio. The full scan-time for each test case was approximately equal to one  
 234 hour, since a projection acquisition time of 1000 ms was used. During the image acquisitions no beam  
 235 filtration was applied to the incoming x-rays produced by the source, which was set up to 60 kV and  
 236  $40.2 \mu A$ . Further details regarding the experimental procedure utilized are documented in more detail  
 237 in Mitroglou, Lorenzi et al. (2015). The reconstruction algorithm employed is based on the work of  
 238 Feldkamp, Davis et al. (1984). Experimental errors of the density reconstruction were estimated using  
 239 the standard deviation of density for air-liquid calibration and are around 4%.

240 In practice, orifice dimensions were not perfectly cylindrical, due to manufacturing defects but also  
 241 due to self-induced hydro-grinding of the sharp features, occurring during the early testing of the  
 242 operation of the device. The end result was giving up to 10% difference in the measured (as well as  
 243 the calculated) flow rate through the orifice, relative to the ideal cylindrical, sharp edge shape.

244 Nevertheless, the resulting geometry was stabilised and remained unchanged after a short operating  
 245 time; its shape is shown in Figure 3. ANSA (Stampouli & Pappas 2014) and MeshLab (Cignoni,  
 246 Callieri et al. 2014) software were used for cleaning and manipulating the CT scan geometry and  
 247 obtaining CAD representations (shown in figures later on) and for meshing. The actual orifice  
 248 geometry has an average radius of  $R_{av} = 1503 \mu\text{m}$  with a standard deviation  $\sigma_R = 14.7 \mu\text{m}$  from ideal  
 249 cylinder. Considering that the spatial resolution of the X-ray scan ( $\varepsilon = 15 \mu\text{m}$ ), the total error of the  
 250 geometry representation is  $\pm 46 \mu\text{m}$  (estimated as  $\pm \sqrt{(3\sigma_R)^2 + \varepsilon^2}$  for 99.7% level of confidence  
 251 (Bevington & Robinson 2003)), or  $\sim \pm 1.5\%$  of the nominal diameter. The CT-scanned orifice  
 252 geometry has been used for the numerical simulations, in order to take into account any geometry  
 253 deviations that can affect the flow pattern, such as the smooth orifice entrance due to hydro-grinding.  
 254 Prior investigation has shown that using an idealized cylindrical orifice with sharp turns at the  
 255 entrance, can lead to a severe underestimation of the flow rate.



256  
 257 FIGURE 3. Close up view at the orifice, coloured according to the deviation from ideal cylinder; a smooth transition is visible  
 258 at the orifice entrance, at the bottom left. Flow moves from left to right.  
 259

## 260 2.2. Simulation model and methodology

261 The numerical methodology used in the present work is based on the mixture level approach, i.e.  
 262 the mixture continuity (2.2) and momentum equations (2.3) are solved:

$$263 \quad \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (2.3)$$

$$264 \quad \frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} \quad (2.4)$$

265 where  $\rho$  is the mixture density,  $\mathbf{u}$  is the velocity vector,  $p$  is pressure and  $\boldsymbol{\tau}$  denotes the stress tensor, as  
 266 described below:

$$267 \quad \boldsymbol{\tau} = \mu_{eff} [\nabla \mathbf{u} + (\nabla \mathbf{u})^T] + \lambda (\nabla \cdot \mathbf{u}) \mathbf{I} \quad (2.5)$$

268 with  $\mu_{eff}$  the total effective viscosity of the mixture, including laminar,  $\mu$ , and eddy viscosity,  $\mu_t$ ,  
 269 contributions and  $\mathbf{I}$  is the identity matrix. The effect of bulk viscosity,  $\lambda$ , is omitted from equation 2.4,  
 270 due to lack of data for Diesel fuel; in any case, bulk viscosity only acts to passing waves so its effect  
 271 to the general dynamics of the flow is negligible. In the present study, turbulence effects are taken into  
 272 account with an Improved Delayed Detached Eddy Simulation model (IDDES), which is blend of  
 273 RANS (k- $\omega$  SST) and an LES Smagorinsky-like model (Mockett 2007; Shur, Spalart et al. 2008),  
 274 depending on the mesh spacing and the local turbulent length scale. Such a treatment allows for low  
 275 resolution in areas where there is little interest, such as upstream the annular passage, or downstream  
 276 the collector, reducing the computational cost, since it reverts to the k- $\omega$  SST model. On the other  
 277 hand, in areas of adequate resolution, the model switches to a Smagorinsky-like subgrid model,  
 278 enabling the accurate reproduction of turbulent structures, which have been shown in previous studies

279 to be crucial in describing cavitation shedding and unsteadiness (Coutier-Delgosha, Reboud et al.  
280 2003; Edelbauer, Struel et al. 2016).

281 Cavitation is tracked with a transport equation for the vapour fraction  $a$ , which has the general  
282 form of:

$$283 \quad \frac{\partial(a\rho_v)}{\partial t} + \nabla \cdot (a\rho_v \mathbf{u}) = R_e - R_c \quad (2.6)$$

284 where  $\rho_v$  is the vapour density and  $R_e$  and  $R_c$  are mass transfer terms associated with the evaporation  
285 and condensation of vapour. In this work, we propose a different formulation of the evaporation and  
286 condensation terms from existing cavitation models, as follows:

$$287 \quad R_e = F(1-a)\rho_v \sqrt{\frac{2 \max(p_v - p, 0)}{3 \rho_l}} \quad (2.7)$$

$$288 \quad R_c = Fa\rho_v \sqrt{\frac{2 \max(p - p_v, 0)}{3 \rho_l}} \quad (2.8)$$

289 where  $\rho_l$  is the liquid density and  $F$  is the ratio of available interfacial surface area (in  $\text{m}^2$ ) for mass  
290 transfer per unit volume (in  $\text{m}^3$ ) of fluid; its units are  $\text{m}^{-1}$ . The formulation of the mass transfer terms  
291 is based on the Zwart, Gerber et al. (2004) model (see also Bakir, Rey et al. (2004)), however there  
292 are several fundamental differences. First of all, the aforementioned mass transfer model can be  
293 considered as a thermodynamic non-equilibrium model, since metastable conditions of liquid tension  
294 may develop if the mass transfer is low, as in e.g. the original Zwart, Gerber et al. (2004) formulation.  
295 On the other hand, if the mass transfer is sufficiently high, theoretically tending to infinity, as  
296 proposed in the present work, the two-phase model tends asymptotically to a barotropic cavitation  
297 model behaviour in e.g. shock tube (Koukouvinis & Gavaises 2015) or Rayleigh collapse tests  
298 (Koukouvinis, Naseri et al. 2016). Adopting such an approach has several advantages, since:

299 - It can inherently predict the Rayleigh collapse of a vapour structure. As a comparison, cavitation  
300 models like the baseline Zwart, Gerber et al. (2004), or Schnerr & Sauer (2001) models may seriously  
301 over predict the collapse time of a vaporous bubble, due to the very small mass transfer rate  
302 (Koukouvinis, Naseri et al. 2016).

303 - The amount of unphysical tension in the liquid volume is significantly reduced and minimum  
304 pressure is much closer to the vapour pressure (Koukouvinis, Naseri et al. 2016). Again, the  
305 aforementioned cavitation models may predict liquid tension up to two orders of magnitude higher  
306 than the present model (Koukouvinis, Gavaises et al. 2016).

307 - The approach of the model resembles a thermodynamic equilibrium assumption for cavitation, i.e.  
308 pressure in the cavitation zone is practically equal to the vapour pressure, which is supported by  
309 experimental data in literature, see e.g. Washio (2014).

310 - There is only a single tuning factor,  $F$ , instead of four of the original Zwart, Gerber et al. (2004)  
311 model. Moreover, its value is a priori known since, in order to be at thermodynamic equilibrium, it  
312 should be theoretically infinite. Obviously an infinite value is not possible to be handled by the  
313 numerical solver, and very high values will cause serious numerical difficulties. However, since there  
314 is asymptotic convergence with the increase of  $F$  towards thermodynamic equilibrium, extreme values  
315 are not needed; in practical cases, such as the one described here, values of  $F$  in the order of  $10^8 \text{ m}^{-1}$   
316 are adequate to maintain numerical stability and prevent liquid tension from becoming unphysical.

317 In the simulations presented here, the liquid phase is modelled as a compressible liquid, with the  
318 Tait equation of state:

$$319 \quad p = B \left[ \left( \frac{\rho}{\rho_0} \right)^n - 1 \right] + p_{ref} \quad (2.9)$$

$$B = \frac{\rho_0 c_0^2}{n} \quad (2.10)$$

where  $B$  is the bulk modulus,  $\rho_0$  is the reference density,  $c_0$  is the reference speed of sound at reference density,  $p_{ref}$  is a reference pressure (here equal to the vapour pressure  $p_v$ ) and  $n$  is an exponent adjusting the stiffness of the liquid. The values used are shown in Table 2 and were chosen based on a Diesel property library (Kolev 2007), at the temperature of the experiment (40°C). While the operating pressure levels, as shown in Table 1, are not high enough to justify a compressible treatment of the liquid, the collapse of vaporous cavities induces locally very high pressures, of the order of 1000 bar, and compressibility becomes important. Moreover, numerical handling would be problematic the instant when vapour disappears completely at the end of the collapse of a vaporous structure if the incompressible assumption was to be enforced, since liquid converges to a singular point with high velocity, obviously violating the incompressibility assumption of  $\text{div}(\mathbf{u}) = 0$ . Vapour properties were assumed to be constant, based on saturation conditions at 40°C.

<i>Property</i>	<i>Liquid (l) – Tait equation</i>	<i>Vapour (v)</i>
Reference density, $\rho_0$ (kg/m <sup>3</sup> )	830	1.123
Bulk modulus, $B$ (Pa)	$172.01 \cdot 10^6$	
Tait equation exponent, $n$ (-)	7.15	
Reference speed of sound, $c_0$ (m/s)	1217.3	
Reference pressure, $p_0$	17200	
Dynamic viscosity, $\mu$ (Pa·s)	$2.16 \cdot 10^{-3}$	$8.00 \cdot 10^{-6}$

TABLE 2. Fluid properties used for the simulations presented hereafter. Properties are estimated at 40°C, based on the work of Kolev (2007)

Despite vapour density being assumed as constant, the liquid/vapour mixture is highly compressible, due to mass transfer terms. In the simulations to be presented, the speed of sound,  $c$ , varies from the liquid speed of sound ( $c_l \sim 1200$  m/s) to even 0.01 m/s in the cavitating mixture region, depending on the local mass transfer from liquid to vapour ( $\delta m$ , kg per m<sup>3</sup> of mixture) and pressure change,  $\delta p$ , see Franc & Michel (2005):

$$\frac{1}{\rho c^2} \cong \frac{1-a}{\rho_l c_l^2} + \frac{a}{\rho_v c_v^2} - \frac{\delta m}{\delta p} \left( \frac{1}{\rho_v} - \frac{1}{\rho_l} \right) \quad (2.11)$$

Mixture compressibility affects the collapse of cavities; a high mass transfer rate (i.e. large absolute value of  $\delta m/\delta p$ ) leads to very low speed of sound within the mixture and supersonic cavity collapse (supersonic in terms of mixture speed of sound inside the cavity, not the liquid speed of sound). Thus during the collapse, pressure within the cavity remains approximately equal to vapour pressure since no information can propagate in the liquid/vapour mixture, leading to the well-known Rayleigh collapse behaviour. A small mass transfer rate will render the mixture stiffer, preventing a violent cavity collapse.

For the computational model, upstream elements of the annular passage have been omitted to conserve computational resources. A complexity of the simulated geometry is the fact that it has a high aspect ratio, i.e. its length is considerably larger than the rest dimensions. In order to generate a high quality mesh, suitable for Detached Eddy Simulation (DES) studies, while efficient in terms of cell count, the Cartesian Cut Cell methodology was employed (Thompson, Soni et al. 1998). Furthermore, five successive levels of refinement were used, to capture important details in the orifice; the base resolution level was 0.8 mm progressively refined to 0.4, 0.2, 0.1, 0.05 mm in the area of interest and an additional refinement level of 0.025 mm near the orifice entrance, see Figure 4.

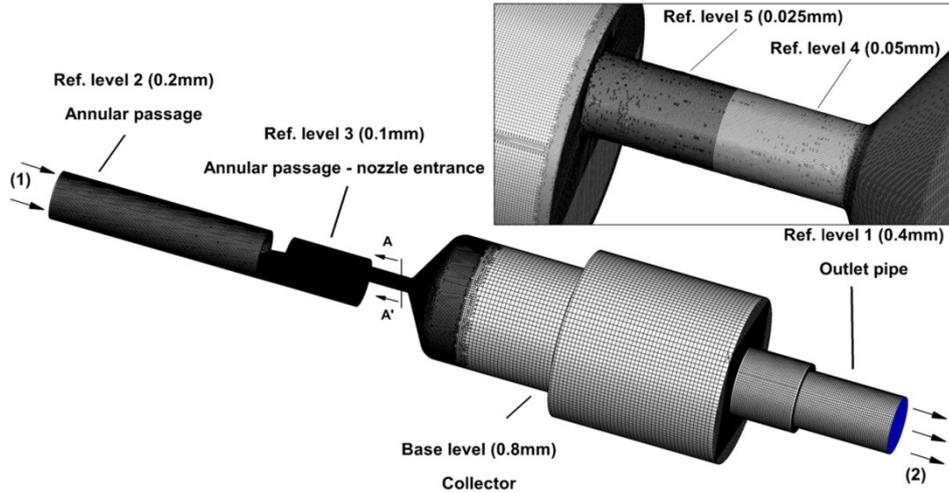


FIGURE 4. The Cut Cell computational mesh with the successive refinement levels visible. The magnified insert shows the orifice mesh in more detail. The boundary conditions are indicated as well with numbers: (1) corresponds to fixed total pressure and (2) to fixed static pressure. Grey colour corresponds to no-slip walls.  $AA'$  view indicates the upstream view in the orifice cross-section.

354  
355  
356  
357  
358  
359

Near wall refinement was also employed at the vicinity of walls to get a better reproduction of boundary layers. Five inflation layers were placed near the walls of the geometry, with the closest cell placed at  $4.5\text{-}9\ \mu\text{m}$  from the wall (in ref. level 5 and 4 respectively, see Figure 4), resulting to a maximum  $y^+$  of  $\sim 5$ . Areas of high  $y^+$  values are mainly located at the entrance of the orifice and away of the area of interest (e.g. annular passage, collector), thus it is not expected to have a negative impact in the quality of the results. In any case, the aim of the turbulence model employed is to resolve the highly unsteady separated regions downstream the turn which contribute to the cavitation dynamics, whereas near wall regions are treated with the RANS sub-model ( $k\text{-}\omega$  SST) which is relatively insensitive to  $y^+$  values.

360  
361  
362  
363  
364  
365  
366  
367  
368  
369  
370  
371  
372  
373  
374  
375  
376  
377  
378  
379  
380

As the computational mesh has resolution of  $50\ \mu\text{m}$  in ref. level 4, which is more than three times the resolution of the micro-CT scan, it was ensured that large scale features of the geometry have been maintained during the discretization procedure. Indeed, the discretized geometry of the orifice has an average radius of  $R_{av} = 1502.2\ \mu\text{m}$  and a standard deviation of  $\sigma_R = 14.2\ \mu\text{m}$  from ideal cylinder. The current computational mesh consists of  $8 \cdot 10^6$  cells. Further mesh refinement towards the CT scan resolution would dramatically increase the cell count and consequently the computational cost. The integration time step used in the present studies is  $0.05\ \mu\text{s}$ , resulting to a convective Courant number of  $\sim 0.2$ . Sampling time for obtaining averages is  $5\ \text{ms}$ , which is equivalent to  $100\ 000$  time steps. As will be shown later, the shedding cycle period is  $\sim 85\ \mu\text{s}$ , which means that  $\sim 60$  shedding cycles were used for averaging and statistic collection; in fact, statistic results changed less than 1% when sampling more time instances, thus the time interval was considered enough for sample-independent results.

381  
382  
383  
384

Total pressure is imposed at the inlet and static pressure at the outlet, see also Figure 4. Since pressure measurements were taken at slightly different locations, pressure losses had to be estimated for the omitted parts. Pressure losses were assumed to obey the generic Darcy loss formula (White 2011):

$$\Delta p = k \frac{\rho \dot{Q}^2}{2S^2} \quad (2.12)$$

385  
386  
387  
388

where  $k$  is a constant dependent on the geometrical features of the omitted part,  $\rho$  the liquid density,  $S$  the cross-section of the fluid passage and  $\dot{Q}$  the volumetric flow rate. The  $k$  constant in (2.12) was determined by performing steady state precursor simulations of the whole tested geometry at non-

389 cavitating conditions. After determining  $k$ , upstream and downstream pressure losses were properly  
 390 compensated in the boundary condition definition at both inlet/outlet in the present simulations. The  
 391 value of  $k$  is  $\sim 1$  and  $\sim 4$  for the omitted inlet and outlet sections respectively. The flow at the omitted  
 392 parts is turbulent, with a Reynolds number of  $\sim 40000$ , however pressures are much higher than the  
 393 orifice, so pure liquid flow is expected. Since the friction coefficient is approximately constant at fully  
 394 turbulent conditions (White 2011),  $k$  values are not expected to vary significantly for the examined  
 395 operating conditions. In any case, pressure losses of the omitted parts are  $\sim 0.38$  and  $\sim 0.27$  bar for the  
 396 upstream and downstream sections respectively (or less than 1% of the upstream pressure and 1.5% of  
 397 the downstream pressure) for the flow rates examined here.

398 At the inlet boundary, zero velocity gradient was imposed at the normal, to the boundary, velocity  
 399 component, while the rest velocity components were set to zero. At the outlet boundary, zero gradient  
 400 boundary conditions were set to all velocity components and transported quantities. Backflow was not  
 401 observed at the outlet boundary throughout the complete simulation duration. Velocity fluctuations  
 402 have not been applied at the inlet, mainly because the mesh there is rather coarse to accurately resolve  
 403 turbulent structures. Nevertheless, as will be shown later, turbulence is generated by the needle  
 404 support structure and this region is well resolved by the computational mesh.

405

### 406 **3. Numerical discretization, mesh convergence and uncertainty**

407 The numerical schemes used for the present study are second order accurate in time and space, as  
 408 shown below, for reference see Ferziger & Peric (2002):

- 409 - bounded central differencing with deferred correction technique for momentum equations
- 410 - Quadratic Upwind Interpolation for Convective Kinematics (QUICK) for the vapour volume  
 411 fraction transport equation.
- 412 - second order upwind for density interpolation, and turbulence transport equations.
- 413 - time integration is performed with an implicit second order backward differencing method.

414 Solution is achieved with a pressure based coupled approach (Chen & Przekwas 2010), which  
 415 ensures a fast convergence rate and residual drop of at least 3 orders of magnitude in each time step.

416 Numerical uncertainties are determined following a grid dependence analysis and a sensitivity  
 417 analysis. Due to the time consuming nature of the simulations, both analyses have been conducted for  
 418 the high intensity cavitation number, at  $Cn = 2.18$ , and the relative errors are assumed to be similar for  
 419 low cavitation number operation as well.

420 First of all, a mesh dependence study was conducted to determine the errors arising from the finite  
 421 discretization. The mesh was coarsened by a factor of 2 everywhere and the conditions of case 2 ( $Cn$   
 422 = 2.18) were evaluated to determine the flow rate, cavity length and volume dependence on the mesh  
 423 resolution. The change between the coarse and fine mesh resolutions may be used to determine the  
 424 Richardson error estimator at the fine mesh ( $E_{fine}$ ) and the Grid Convergence Index ( $GCI$ ), following  
 425 Roache (1997), as shown below:

$$426 \quad E_{fine} = \frac{\varepsilon}{1 - r^k} \quad (3.1)$$

$$427 \quad GCI_{fine} = F_s |E_{fine}| \quad (3.2)$$

428

429 In the aforementioned equations,  $\varepsilon$  stands for the difference in the observed quantity between fine and  
 430 coarse meshes,  $r$  is the ratio of the fine to coarse mesh spacing (here it is 2), the exponent  $k$  represents  
 431 the formal order of accuracy of the employed method (here equal to 2) and  $F_s$  is a safety factor, equal  
 432 to 3 according to the relevant recommendation by Roache (1997).

433

Mesh level	Volume flow rate (lt/s)	Cavity length (mm)	Cavity volume (mm <sup>3</sup> )
Fine	0.466	7.17	14.7
Coarse	0.468	7.10	14.0
Error estimator	7.2·10 <sup>-4</sup> (0.16%)	0.02 (0.32%)	0.23 (1.56%)
GCI	2.2·10 <sup>-3</sup> (0.47%)	0.07 (0.98%)	0.69 (4.69%)

TABLE 3. Error estimator and Grid Convergence Index (*GCI*) for case 2,  $Cn = 2.18$ . The cavity length is estimated using the 50% average liquid fraction isosurface. Cavity volume is based on the volume integral of the time-average vapour fraction. Both the Error estimator and *GCI* are expressed at fine mesh level and percentages are expressed in reference to fine mesh solution.

434

435

436

437

438

439

440

441

442

443

444

A complete analysis of the numerical uncertainties should, ideally, include the sensitivity of all simulation parameters (Coleman & Steele 2009), such as boundary conditions, model coefficients, material properties, etc. However, this would involve an enormous computational effort, given the fact that the discussed simulations are inherently unsteady and need statistical sampling. For this reason it was decided to focus mainly on parameters affecting the simulation for which uncertainty data are available, such as the upstream and downstream pressure boundaries. The influence of the boundary condition input uncertainty was analyzed by determining the sensitivity of flow rate, cavity length and cavity volume to the upstream and downstream pressures, for which uncertainties are known. Sensitivities were estimated by shifting pressure conditions at boundaries by a small amount, equal to 0.5% of their nominal value (nominal values are reported in table 1).

Sensitivity	Upstream pressure ( $p_{up}$ )	Downstream pressure ( $p_{down}$ )	Total uncertainty
Vol. flow rate $\frac{\partial \dot{Q}}{\partial p}$	5.23·10 <sup>-11</sup> m <sup>3</sup> /(s·Pa)	-2.64·10 <sup>-11</sup> m <sup>3</sup> /(s·Pa)	±2.2·10 <sup>-3</sup> lt/s (±0.5%)
Cavity length $\frac{\partial l}{\partial p}$	1.33·10 <sup>-9</sup> m/Pa	-4.23·10 <sup>-9</sup> m/Pa	±0.092 mm (±1.3%)
Cavity volume $\frac{\partial V}{\partial p}$	6.35·10 <sup>-15</sup> m <sup>3</sup> /Pa	-2.26·10 <sup>-14</sup> m <sup>3</sup> /Pa	±0.475 mm <sup>3</sup> (±3.2%)

TABLE 4. Uncertainty determination based on sensitivities of vol. flow rate ( $\dot{Q}$ ), cavity length ( $l$ ) and cavity volume ( $V$ ) to upstream and downstream pressures. Uncertainty values inside parentheses are relative in respect to reference flow rate (0.466 lt/s), cavity length (7.2 mm, corresponding to the 50% average liquid isosurface) and cavity volume (14.7 mm<sup>3</sup>).

445

446

#### 4. Experimental and numerical results

447

448

449

450

451

452

453

454

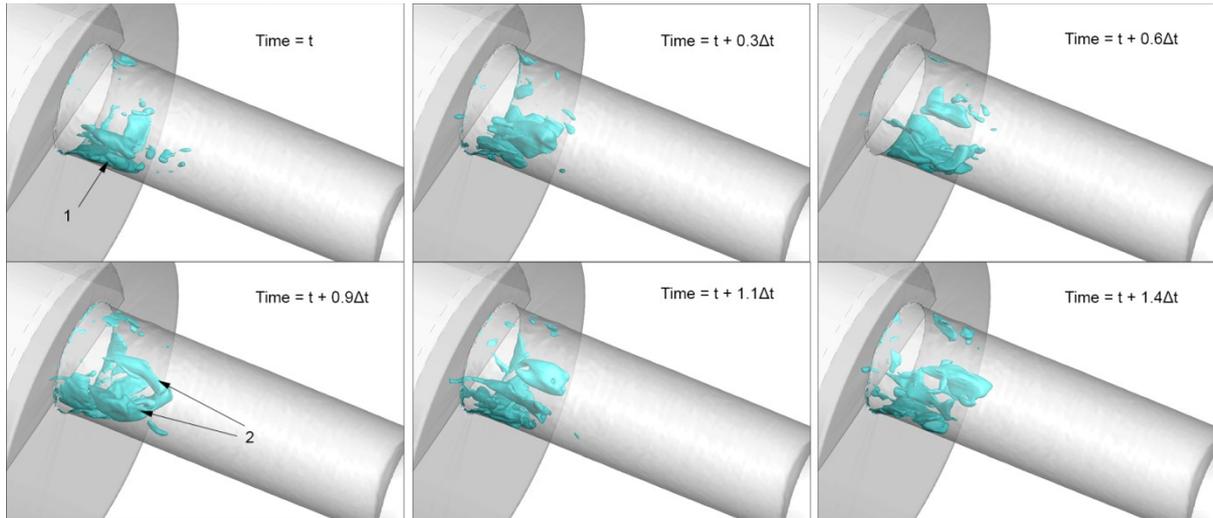
455

456

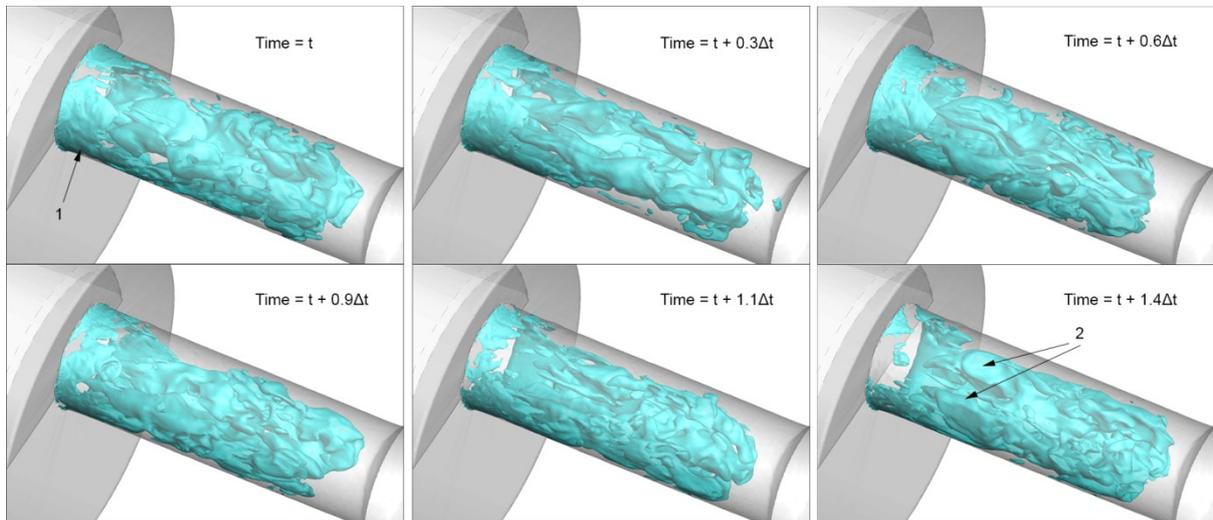
457

In this section indicative instances mainly of the high cavitation number ( $Cn = 2.18$ ) operation will be presented. The reason for this decision is that at low cavitation number ( $Cn = 1.5$ ) cavitation is sparse and mainly limited near the orifice entrance, so relevant shedding phenomena are not that pronounced. Example instances of a 3D representation of cavitation are shown in Figure 5 and Figure 6. The topology of the cavitation pattern may be divided in the following structures: (1) a sheet cavity, formed at the edge of the orifice entrance, oscillating at a Strouhal number of ~0.35-0.38, based on orifice diameter and average flow velocity (attached cavity shedding period of 78-95 μs for high and low cavitation number operation respectively), (2) two large cavitation lobes, which essentially are large cavitating vortices, formed in the core of the orifice and (3) smaller cavitating vortices that occasionally detach from the aforementioned structures and travel downstream the orifice, or even inside the collector volume (see indicative instances for  $Cn = 2.18$ , Figure 16). At low cavitation

458 number ( $Cn = 1.5$ ), it is relatively easy to identify the cavitation structure topology, due to its sparse  
 459 distribution; in fact, the two cavitating lobes may extend from the needle surface, up to 3 mm  
 460 downstream the orifice entrance and occasionally may completely disappear, leaving only the  
 461 attached sheet cavity at the orifice entrance (see sequence at Figure 5). On the other hand, at high  
 462 cavitation number, the two cavitating lobes persist indefinitely, extending up to 8 mm downstream the  
 463 orifice entrance, obstructing a large portion of the flow passage and blocking the view to the sheet  
 464 cavity (see sequence at Figure 6). An alternative view of the sheet cavity development is in Figure 7,  
 465 shown as a slice at the midplane of the orifice. In both cases, the location of the cavitating lobes  
 466 coincides with the position of large counter-rotating vortical structures formed in the cross-section of  
 467 the orifice (see also Figure 8, Figure 9 and Figure 10).



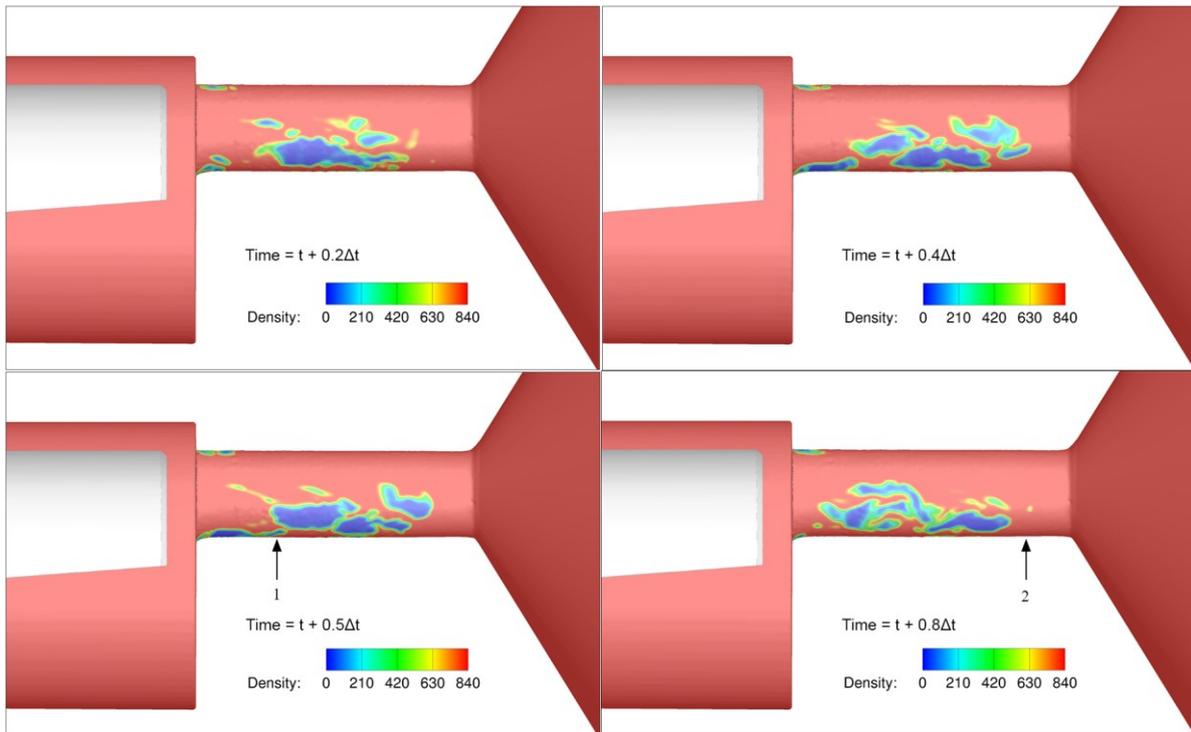
468  
 469 FIGURE 5. Indicative instances of a cavity shedding cycle, shown for  $Cn = 1.5$ : 50% liquid isosurface. (1) indicates the  
 470 attached cavity sheet, (2) the pair of cavitating vortices.  $\Delta t$  is the attached cavity shedding cycle period, corresponding to  
 471 95  $\mu\text{s}$ .  
 472



473  
 474 FIGURE 6. Indicative instances of a cavity shedding cycle, shown for  $Cn = 2.18$ : 50% liquid isosurface. (1) indicates the  
 475 attached cavity sheet, (2) the pair of cavitating vortices, which occupy the whole orifice cross-section.  $\Delta t$  is the attached  
 476 cavity shedding cycle period, corresponding to 78  $\mu\text{s}$ .  
 477

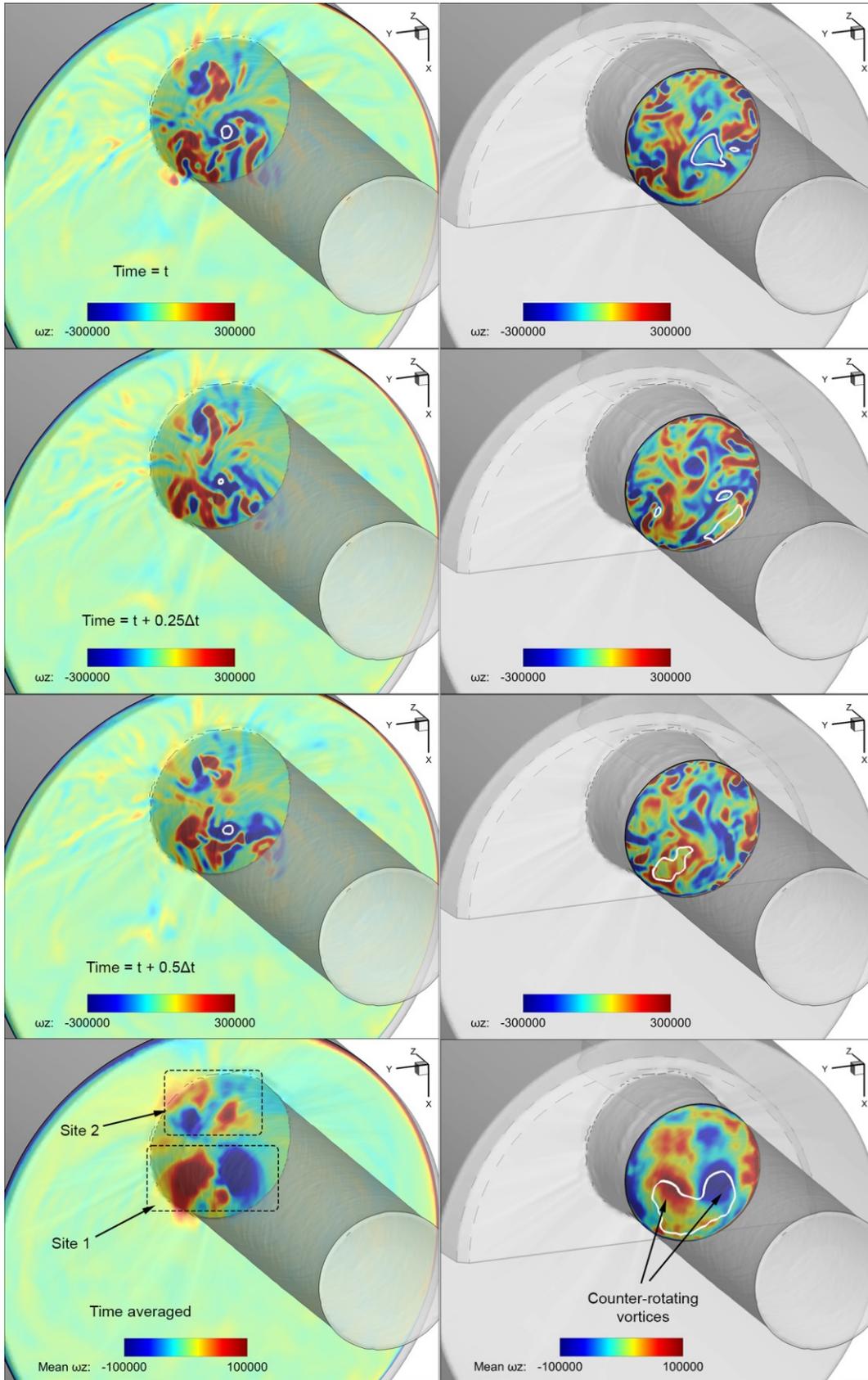
478 Figure 7 shows the density distribution at the midplane of the geometry over a cavity shedding  
 479 cycle, for  $Cn = 2.18$ , to obtain clear view, unimpeded by the two cavitation lobes. As it is expected,  
 480 cavitation structures are formed at the side where the majority of the flow enters, i.e. the side that is  
 481 relatively unobstructed by the throttling needle presence. Examining the orifice side where extended

482 cavitation develops, cavitation structures form at the edge of the orifice entrance and expand further  
 483 downstream (see the instances at  $0.2 \Delta t$  and  $0.4 \Delta t$ ). These structures remain attached on the orifice  
 484 wall and extend up to 2mm from entrance. At the trailing edge of these structures, cavities may detach  
 485 and travel further downstream dragged by the flow. The sheet cavity growth/collapse cycle is repeated  
 486 every  $\sim 78 \mu\text{s}$ , corresponding to a Strouhal number ( $St = f L/U$ , where  $f$  the shedding frequency,  $L$   
 487 the orifice diameter, equal to  $\sim 3\text{mm}$  and  $U$  the characteristic velocity of  $100 \text{ m/s}$ ) of  $0.38$ . Cavitation  
 488 structures collapse at different locations; the attached cavity, collapses at  $\sim 2 \text{ mm}$  from the orifice  
 489 entrance, whereas shed cavities at approximately  $7\text{-}8 \text{ mm}$  downstream the entrance (see also the  
 490 instances at  $0.5 \Delta t$  and  $0.8 \Delta t$ , respectively). The obstructed side of the orifice exhibits cavitation  
 491 formation, but mainly in the form of attached cavities, without significant shedding. This side is also  
 492 free of erosion, as will be shown later on.



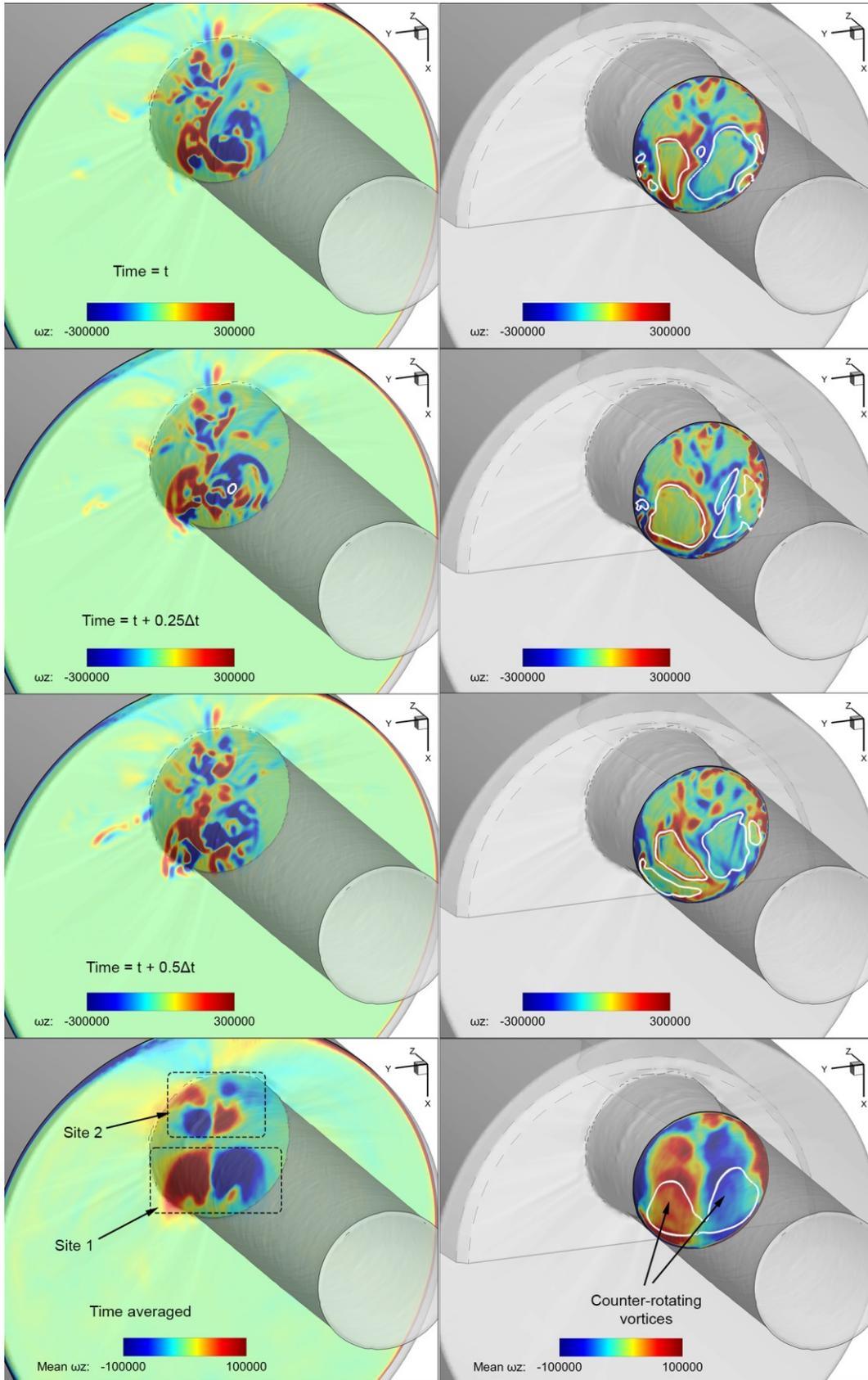
493  
 494 FIGURE 7. Indicative instances of a cavity shedding cycle, shown for  $Cn = 2.18$ . Two collapse sites can be identified at points  
 495 indicated with (1) and (2).  $\Delta t$  is the attached cavity shedding cycle period ( $\sim 78 \mu\text{s}$ ). The shedding cycle corresponds to a  
 496 Strouhal number of  $\sim 0.38$ . Density units are  $\text{kg/m}^3$ . The flow moves from left to right.

497 Figure 8 and Figure 9 show the  $z$ -vorticity distribution at two indicative slices, one  $0.5 \text{ mm}$   
 498 upstream and one  $2 - 2.5 \text{ mm}$  downstream the orifice entrance. Several instances are selected for both  
 499 low and high cavitation number operation. Despite the highly irregular structure of the instantaneous  
 500  $z$ -vorticity distribution, some features are clearly observable. First of all, at  $0.5 \text{ mm}$  upstream the  
 501 orifice entrance, it is clear that there is an alternating pattern of vorticity sign, indicating counter-  
 502 rotating vortices. These counter-rotating vortices may be grouped in two distinct sites ("Site 1" and  
 503 "Site 2"), note that it is easier to conceive the pattern through the time-averaged velocity field. These  
 504 sites are associated with the coherent vortical structures shown in Figure 12 and cavitation may form  
 505 there occasionally (see also Figure 14 and Figure 15). Further downstream the orifice entrance,  $z$ -  
 506 vorticity distribution is more irregular, though it is still recognizable that the left and right parts of the  
 507 cross-section are mainly dominated by structures of opposite vorticity signs, i.e. highly transient, large  
 508 and irregular counter-rotating vortices, which may cavitate, intermittently at low intensity cavitation  
 509 ( $Cn = 1.5$ ) or continuously at high intensity cavitation ( $Cn = 2.18$ ).



510  
 511  
 512  
 513  
 514

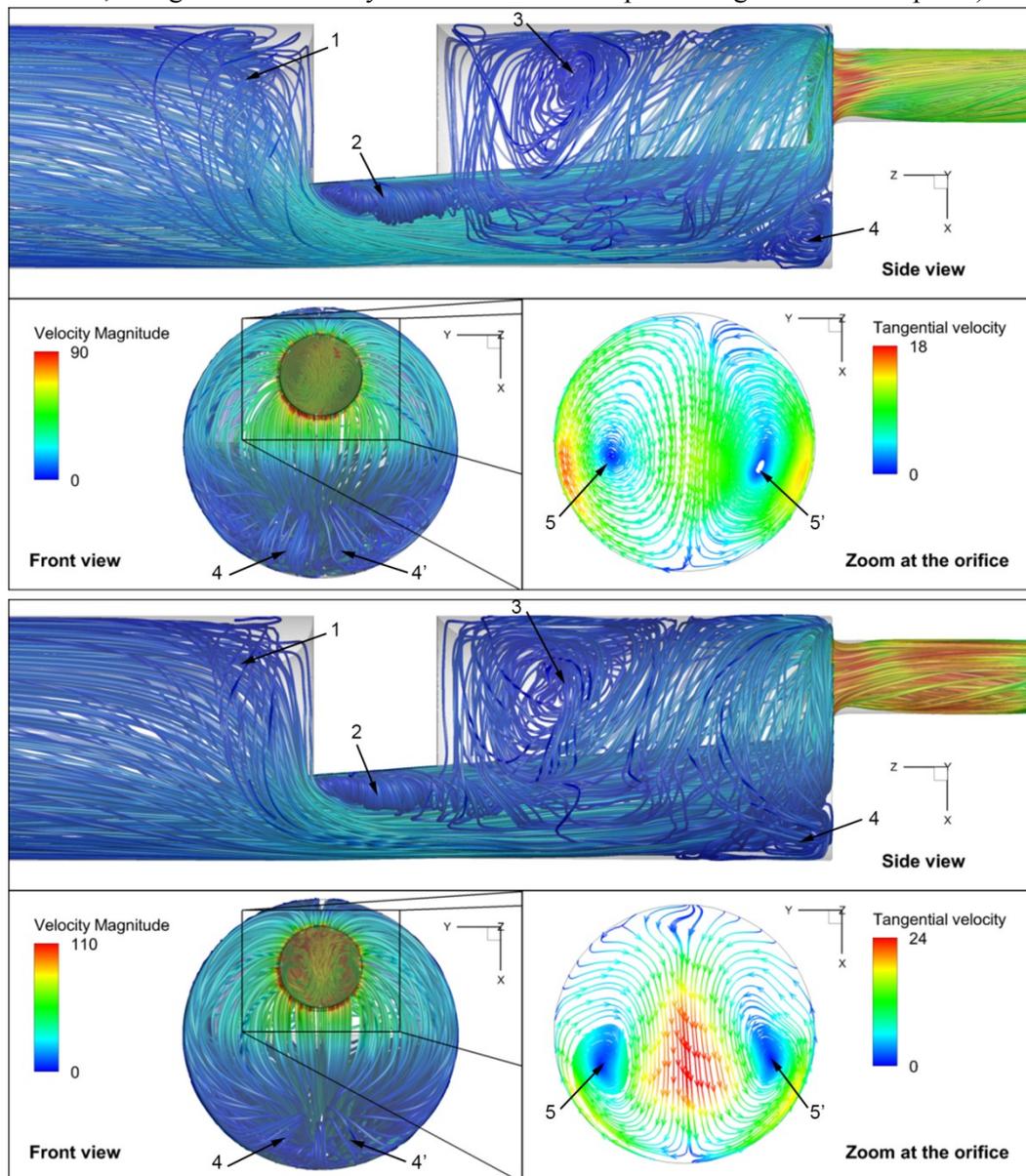
FIGURE 8. Z-Vorticity ( $\omega_z$ ) distribution at the cross-section 0.5 mm upstream the entrance (left) and 2mm downstream the orifice entrance (right),  $Cn = 1.5$ , for indicative instances and for the time-averaged velocity field. Units are  $1/s$ .  $\Delta t$  corresponds to the shedding period, i.e.  $\sim 95 \mu s$ . The thick white line indicates 50% liquid volume fraction isoline for instantaneous and 90% liquid volume fraction for time-averaged. The collector volume has been blanked for clarity.



515  
 516  
 517  
 518  
 519

FIGURE 9. Z-Vorticity ( $\omega_z$ ) distribution at the cross-section 0.5 mm upstream the entrance (left) and 2.5mm downstream the orifice entrance (right),  $Cn = 2.18$ , for indicative instances and for the time-averaged velocity field. Units are  $1/s$ .  $\Delta t$  corresponds to the shedding period, i.e.  $\sim 78 \mu s$ . The thick white line indicates 50% liquid volume fraction isoline for instantaneous and 75% liquid volume fraction for time-averaged. The collector volume has been blanked for clarity.

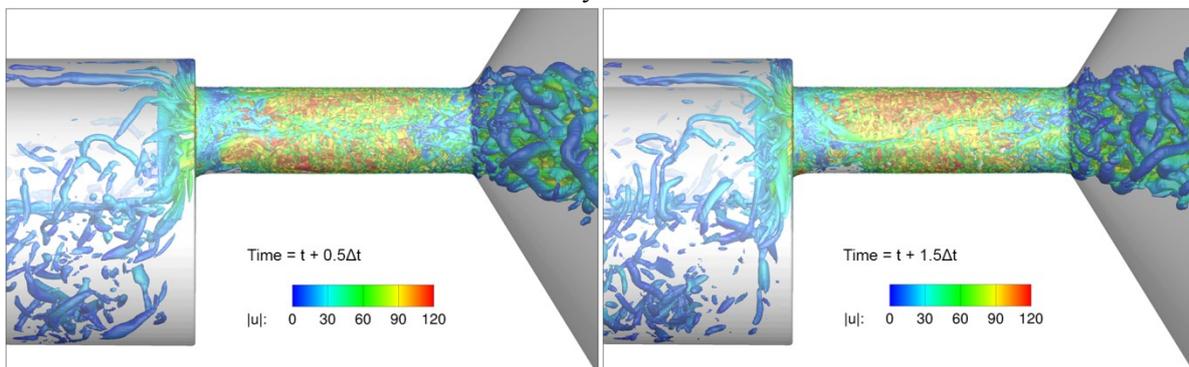
520 Another observation is that vorticity distribution is somewhat more irregular at low cavitation  
 521 intensity,  $Cn = 1.5$ , than high cavitation intensity,  $Cn = 2.18$ . As will be explained later on, this is an  
 522 effect of turbulence suppression due to the extended cavitation formation at high  $Cn$  operation. The  
 523 averaged flow field vorticity distribution shows clearly the extents of the two counter-rotating vortices  
 524 occupying the core of the orifice cross-section, also depicted in Figure 10, as the time-average flow  
 525 field streamlines. Both operating conditions at  $Cn = 1.5$  and  $2.18$  show a similar average flow  
 526 topology, but flow velocities are slightly higher for the  $Cn = 2.18$  case. Moving from the inlet and  
 527 following the flow towards the orifice, the first geometric feature to encounter is the needle support  
 528 structure, which obstructs part of the flow path. The blockage induces flow detachment, which is  
 529 associated with a longitudinal horse-shoe vortex, indicated with number 1 in Figure 10 (shown only  
 530 from side view, though the vortex is symmetric and the two parts bridge over the midplane).



531  
 532 FIGURE 10. Average flow topology upstream and inside the orifice. Side view and front view streamlines are coloured  
 533 according to the average velocity magnitude. The zoomed view at the orifice shows a slice (upstream view inside the orifice  
 534 at 7 mm downstream the entrance) with the average tangential velocity streamlines (coloured according to the tangential, to  
 535 the slice, velocity magnitude). The numbers indicate vortices, whereas the prime symbol (') indicates symmetric, to the  
 536 midplane, structures. Top row results correspond to  $Cn = 1.5$ , bottom row  $Cn = 2.18$ ; the average flow topology is essentially  
 537 identical.

538 The edges of the needle support structure induce a symmetric pair of vortices, indicated with 2 in  
 539 Figure 10, which extend in the expanding fluid region upstream the orifice entrance. The sharp  
 540 expansion after the needle support causes flow detachment, with the formation of a symmetric pair of  
 541 vortices at the sides of the needle tip, similar to those occurring downstream a backward facing step,  
 542 denoted with the number 3. Also the direction change and the strong constriction at the orifice  
 543 entrance causes flow detachment and the associated vortex tube indicated with number 4 in Figure 10.  
 544 Examining the flow inside the orifice, 7 mm downstream the entrance, a pair of counter-rotating  
 545 vortices are visible, indicated with the numbers 5 and 5'. An observable difference is that at  $Cn = 1.5$   
 546 the average tangential velocity is slightly higher at the circumference of the cross-section, whereas at  
 547  $Cn = 2.18$  the average tangential velocity is higher at the midplane. A slight asymmetry is also  
 548 observable in the average tangential distribution in both cases; it is speculated that this is related to the  
 549 asymmetric geometry features of the orifice surface, since such asymmetries were observed in the  
 550 average cavitation distribution as well.

551 The flow in the simulated section is turbulent; Reynolds number ranges from 15000 at the annulus,  
 552 upstream the orifice, to even 100000, instantaneously, downstream the collector. Figure 11 shows the  
 553 coherent turbulent structures (i.e. vortices) at the vicinity of the orifice. The turbulent structures are  
 554 indicated using the second invariant of the velocity gradient tensor, also known as  $q$ -criterion (Haller  
 555 2005; Green, Rowley et al. 2007), for a value of  $q = 10^9 \text{ s}^{-2}$ . As shown, the flow field is highly  
 556 transient with velocities reaching even 120 m/s. A particular complexity of the flow in this geometry  
 557 is that turbulent structures are generated upstream the orifice entrance, due to the constriction imposed  
 558 by the support structure of the needle. It was found that these vortices can have a detrimental effect on  
 559 the flow further downstream. However due to the low local velocities upstream the orifice, a long  
 560 simulation time was needed as a transitional stage, for the build-up of these structures and their  
 561 propagation to the orifice entrance. While upstream the orifice turbulent structures have a relatively  
 562 random orientation, as the flow moves near the entrance, vortices start to elongate and stretch. The  
 563 needle flat surface itself acts as a starting point of vortical structures, which extend inside the orifice,  
 564 at the streamwise direction. As shown in Figure 5 and later on, these vortical structures may  
 565 occasionally cavitate, due to the strong centrifugal forces. The formation of such structures, starting  
 566 from the needle is a manifestation of Helmholtz's second theorem (Batchelor 2000), which states that  
 567 a vortex tube cannot start or end in a (inviscid) fluid; it must extend to a boundary or form a loop.  
 568 Additional vortical structures are formed at the region surrounding the high velocity jet expelled in the  
 569 collector chamber and are the outcome of shear layer instabilities.



570  
 571 FIGURE 11. Representation of coherent vortical structures at two indicative instances, employing the  $q$ -criterion (using a  
 572 value of  $q = 10^9 \text{ s}^{-2}$ ), at  $Cn = 2.18$ . The colouring is according to the local velocity magnitude (units are m/s). The flow  
 573 moves from left to right.  $\Delta t$  is the attached cavity shedding period, 78  $\mu\text{s}$ .

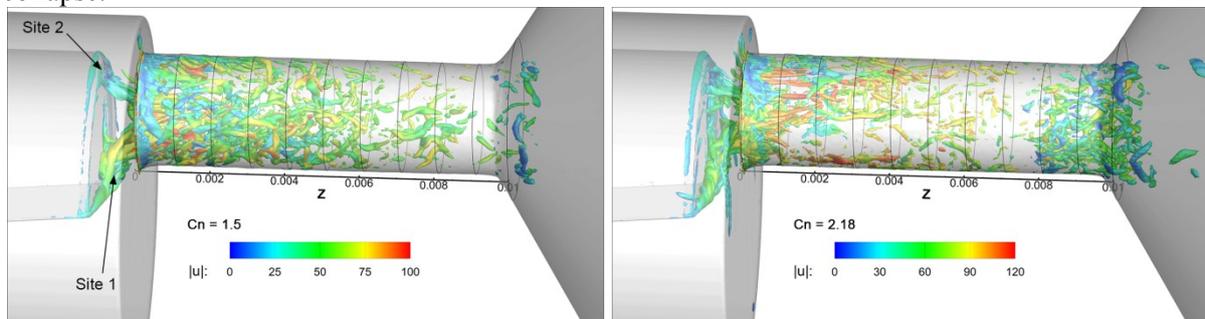
574  
 575 The instantaneous vortical structures at high and low cavitation number are shown in Figure 12; in  
 576 both cases the same value of  $q = 2 \cdot 10^{10} \text{ s}^{-2}$  is shown, in order to have an objective comparison of  
 577 structures of the same strength. It is reminded that both are fully turbulent conditions, but  $Cn = 2.18$  is

578 at a slightly higher Reynolds number ( $\sim 10\%$  higher). For both cases it is clear that there are mainly  
 579 two agglomerations of vortical structures starting from the needle surface, at regions indicated as "Site  
 580 1" and "Site 2" and extending inside the orifice (see also figure FIGURE 8 and figure FIGURE 9). At  
 581 low cavitation number,  $Cn = 1.5$ , these are clearly separated and distinct, whereas at high cavitation  
 582 number,  $Cn = 2.18$ , they appear fuzzier, entangled and twisted. At low cavitation number vortical  
 583 structures are mainly concentrated near the orifice entrance and slowly diminish towards the orifice  
 584 exit, following the flow. On the other hand, at high cavitation number, vortical structures have a more  
 585 clear streamwise direction near the orifice entrance, whereas they are, in general, sparse in the middle  
 586 of the orifice, due to cavitation presence which has the effect of locally suppressing turbulence, see  
 587 also Gnanaskandan & Krishnan (2016). However, there is a concentration of turbulent structures at  
 588 the area of collapse, i.e. at 7-8 mm downstream the entrance.

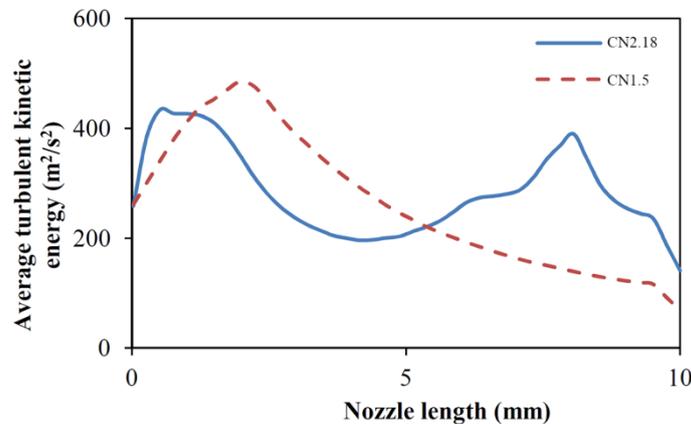
589 These observations may be represented in a more concise manner, considering the averaged  
 590 turbulent kinetic energy (containing simulated and modeled/subgrid scales) across the length of the  
 591 orifice, see Figure 13. As described above, operation at  $Cn = 1.5$  shows a peak of turbulent kinetic  
 592 energy at  $\sim 2$  mm downstream the orifice entrance and slowly diminishes afterwards. On the other  
 593 hand, high cavitation number operation shows two peaks, one at  $\sim 1$ mm downstream the entrance and  
 594 another at  $\sim 8$  mm downstream the entrance. The locations of the turbulent kinetic energy peaks seem  
 595 to be related with the areas of cavitation collapse in both examined cases. This can be explained by  
 596 two mechanisms:

597 (a) turbulence suppression caused by cavitation presence. This explains the low values of turbulent  
 598 kinetic energy near the orifice entrance for low cavitation intensity ( $Cn = 1.5$ ) operation and the local  
 599 minimum from 2 to 8 mm for high cavitation intensity ( $Cn = 2.18$ ) operation.

600 (b) cavitation collapse induces a strong disturbance of the flow field; during collapse very high  
 601 pressures and velocities are generated locally, increasing the turbulent fluctuations in the areas of  
 602 collapse.

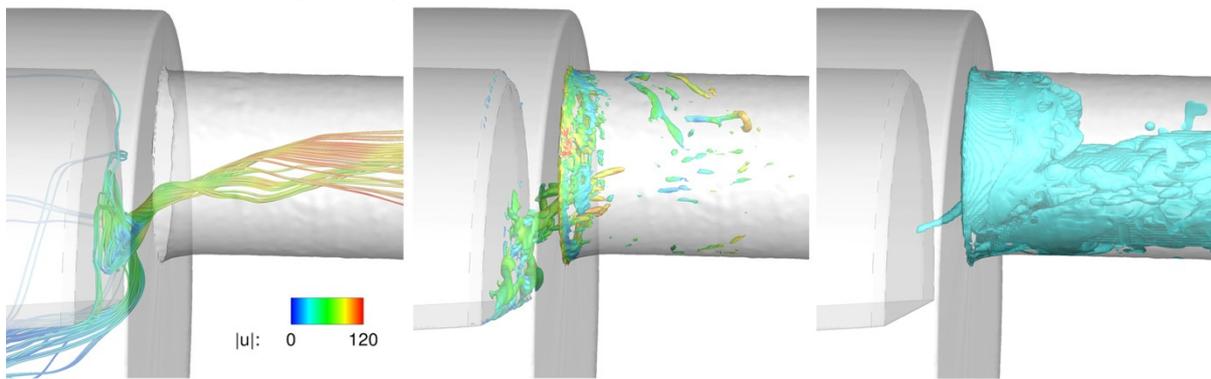


603  
 604  
 605  
 606  
 FIGURE 12. Comparison of the turbulent structures between the two operating conditions at  $Cn = 1.5$  and  $Cn = 2.18$  in an  
 indicative instance. The isosurface of  $q = 2 \cdot 10^{10} \text{ s}^{-2}$  is used for the representation, coloured according to the local velocity  
 magnitude (units are m/s). The flow moves from left to right.

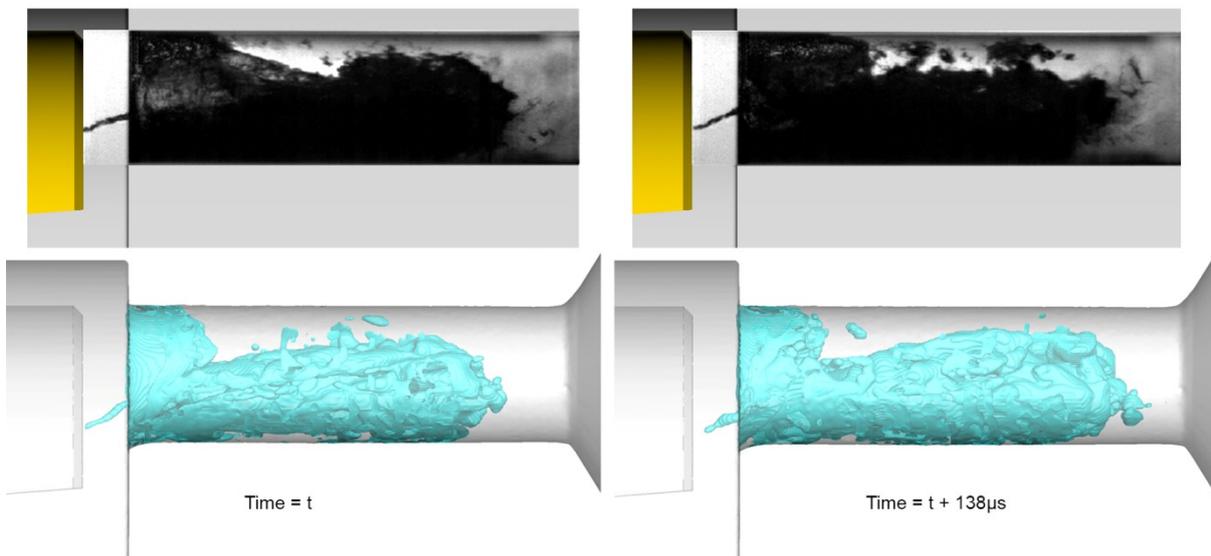


607  
 608  
 FIGURE 13. Comparison of the turbulent kinetic energy averaged at slices across the orifice for the two conditions examined.

609 In Figure 14 and Figure 15 details of a cavitating vortex, starting from the needle, are shown. The  
 610 swirling motion of the flow is evident by observing the flow streamlines or by considering the local  
 611 isosurface of the  $q$ -criterion (Figure 14). The intensity of the swirling motion is enough to reduce the  
 612 pressure in the vortex core and trigger the formation of a vaporous cavity that has an elongated shape,  
 613 following the shape of the corresponding vortex, as visualized with the density isosurface in Figure  
 614 14. This particular cavitating vortex starts from "Site 1", as was discussed before, though both sites  
 615 are prone to cavitating vortex formation. Such effects have been found in the visualization of  
 616 experiments as well, see Figure 15. Even though the vortical fluid motion persists in the local vicinity  
 617 of the two aforementioned sites, the existence of cavitating vortices is highly unsteady. It is stressed  
 618 that visualization of a cavitating vortex in Figure 15 does not exclude the existence of other, non-  
 619 cavitating, vortices, since shadowgraphy cannot reveal the presence of the latter. As illustrated in  
 620 Figure 14,  $q$ -criterion representation indicates many vortical structures starting from the needle, only  
 621 one of which is cavitating at the given instance.



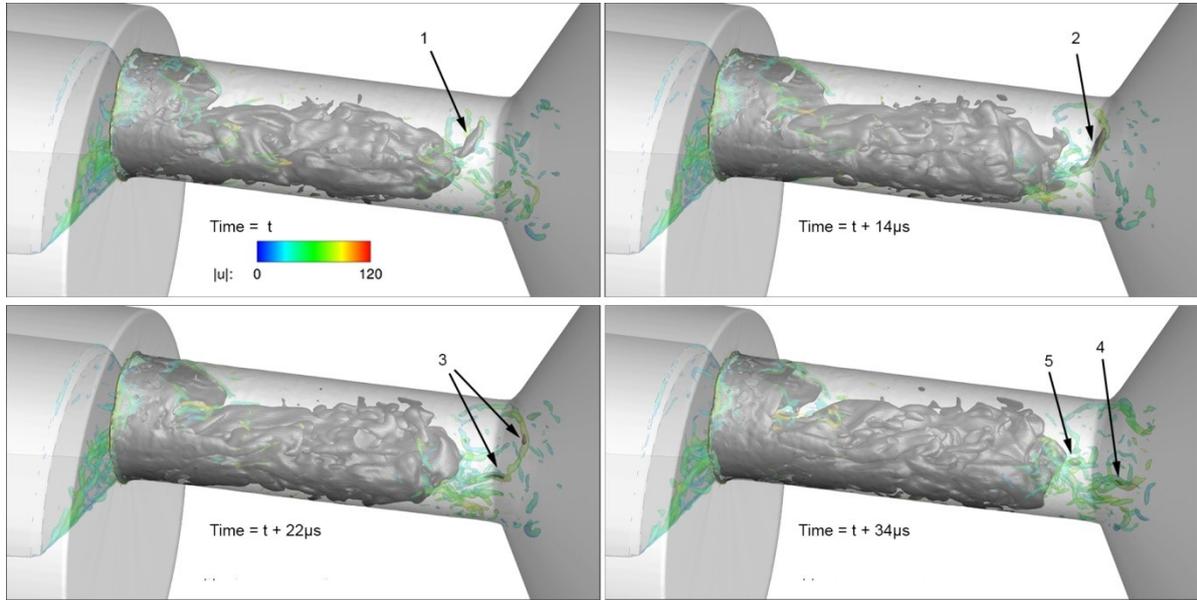
622  
 623 FIGURE 14. Indicative images showing the flow features in the vicinity of a cavitating vortex, starting from the needle and  
 624 moving inside the orifice for high cavitation number ( $Cn = 2.18$ ). From left to right: flow streamlines coloured according to  
 625 velocity magnitude (in m/s), isosurface of  $q$ -criterion (value  $7.5 \cdot 10^{10} \text{ s}^{-2}$ ), coloured according to velocity magnitude and  
 626 isosurface of the liquid fraction at 95%. The flow moves from left to right.



627  
 628 FIGURE 15. Indicative instances showing the formation of cavitating vortices, starting from the needle and extending inside  
 629 the orifice,  $Cn = 2.18$ . Upper row, shadowgraphy from experiment, lower row simulation results, visualized with a liquid  
 630 fraction isosurface at 95%. The flow moves from left to right.

631  
 632 Another manifestation of vortical cavities is downstream the area of the cavity collapse; these  
 633 correspond to the 3<sup>rd</sup> topological type of cavities, as discussed in the relevant paragraph. The intensely  
 634 swirling flow formed downstream the area of collapse may protect some detached vaporous cavities,

635 allowing them to travel further downstream despite the pressure field recovery. An example of this  
 636 effect is shown in Figure 16, where a vaporous cavity is able to reach the exit of the orifice and enter  
 637 the collector volume. It is reminded that the collector pressure is around 17 bar. In the indicative  
 638 instances provided, a cavity is detaching (1), then it elongates, following the stretching of the vortex  
 639 (2), later on breaks (3) and eventually collapses (4). At the end of the process, another cavity detaches  
 640 (5).  
 641



642  
 643 FIGURE 16. Cavitating vortices are formed at the trailing edge of the cavity inside the orifice, high cavitation number  
 644 operation ( $Cn = 2.18$ ). Cavitation structures (75% liquid) are shown as a black/dark grey isosurface for contrast, whereas the  
 645 semi-translucent isosurface represents vortex cores ( $q = 5 \cdot 10^{10} \text{ s}^{-2}$ , coloured according to velocity magnitude); as visible,  
 646 detached cavitation structures may be associated with relevant vortex cores. The flow moves from left to right.

647  
 648 The average flow rate through the orifice is shown in Table 5. As shown, the maximum error is  
 649 1.3%, at the highest cavitation number, which is close to the validation uncertainty of  $\sim 0.8\text{-}0.9\%$ .

650

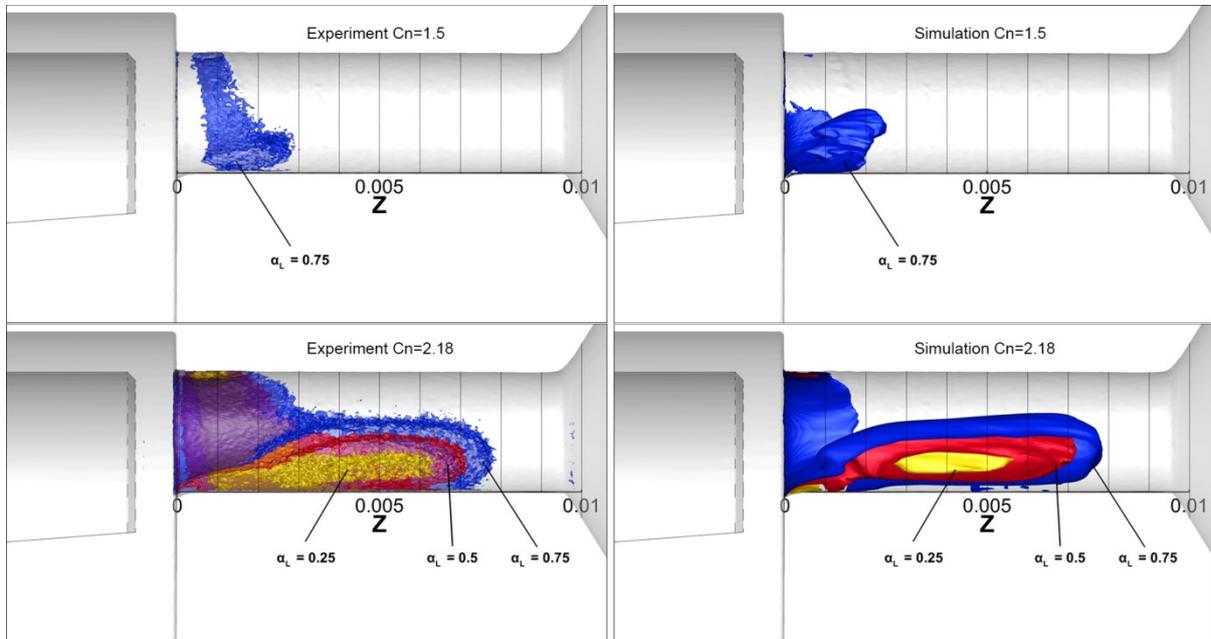
	Upstream pressure $p_{up}$ (bar)	Downstream pressure $p_{down}$ (bar)	Cavitation number $Cn$ (-) $\sigma$ (-)		Flow rate - experiment $\dot{Q}$ (lt/s)	Flow rate - simulation $\dot{Q}$ (lt/s)	Relative Error	Validation uncertainty
1	43.05	17.30	1.5	0.67	0.397	0.399	0.7%	0.9%
2	55.00	17.45	2.18	0.46	0.460	0.466	1.3%	0.8%

TABLE 5. Volumetric flow rate comparison for the two examined cases. Validation uncertainty for case 1 is based on the same absolute values of uncertainties as case 2.

651

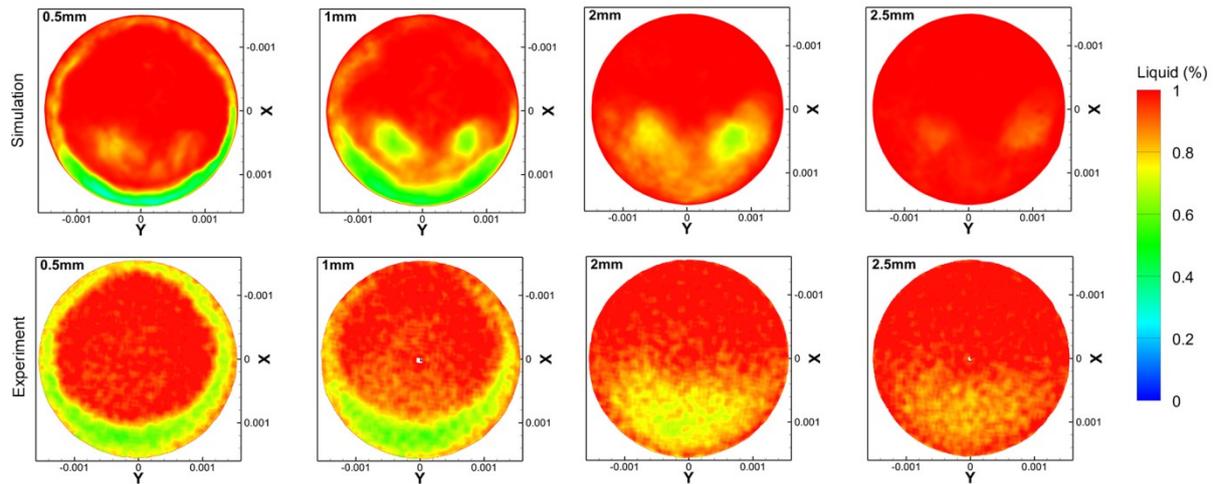
652 At low cavitation number ( $Cn = 1.5$ ), the average amount of vapour is low and mainly located near  
 653 the entrance of the orifice. The average cavitation extent from the CT measurements spans from 0.5  
 654 mm to  $\sim 2.8$  mm downstream the orifice entrance, estimated using an isosurface of 75% liquid  
 655 fraction; lower liquid fractions cover much smaller areas and are more noisy. Simulation results  
 656 indicate that cavitation starts from the entrance of the orifice and reaches 2.5 mm downstream, for the  
 657 same liquid fraction isosurface (75%). The picture is substantially different at the high cavitation  
 658 number case examined, at  $Cn = 2.18$ , where cavitation spans from the orifice entrance and covering  
 659 80% of its length. The experimental results indicate that the isosurfaces of 25%, 50% and 75% liquid  
 660 reach  $\sim 6$  mm, 7 and 8mm respectively. Simulation results indicate the same pattern as well, with the

661 isosurface of 25% liquid slightly less extended. It is notable that both numerical simulation and  
 662 experiment show that the isosurface of 25% liquid is detached from the wall. An indicative averaged  
 663 liquid fraction distribution is shown in Figure 17 at low and high cavitation number,  $Cn = 1.5$  and  
 664 2.18, for both the experiment and simulations. Differences in the average cavity length are within the  
 665 validation uncertainty ( $\sim 0.1$  mm) for the 50% average liquid isosurface at  $Cn = 2.18$ .  
 666

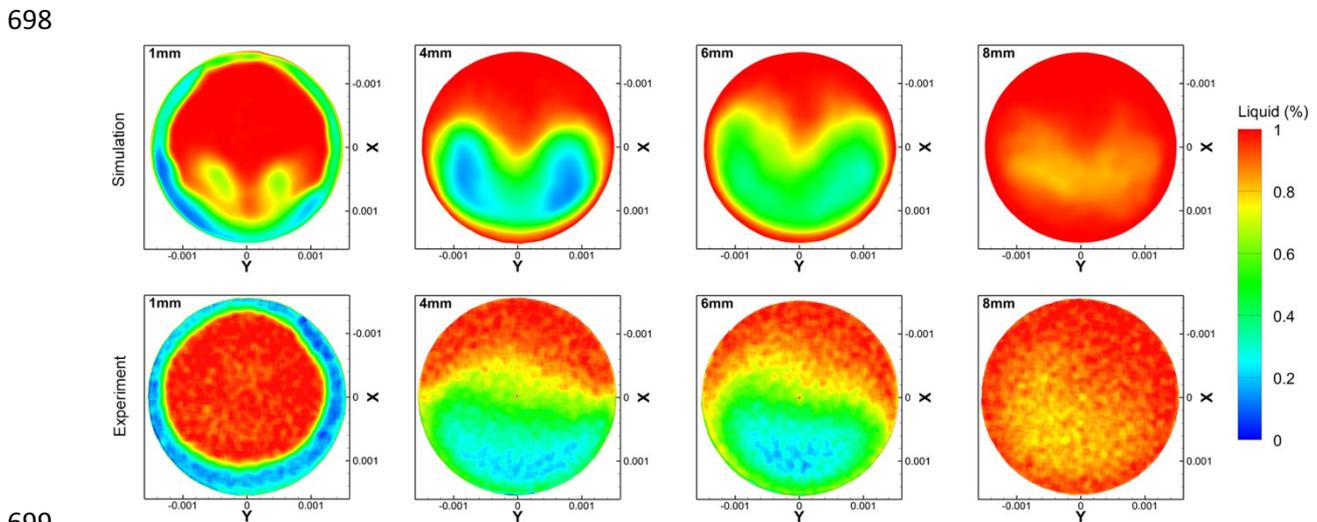


667  
 668 FIGURE 17. Side view of the orifice, showing average liquid fraction isosurfaces ( $\alpha_L = 0.25, 0.5$  and  $0.75$  represented with  
 669 yellow, red and blue colour respectively) for  $Cn = 1.5$  (top),  $Cn = 2.18$  (bottom). Isosurfaces are clipped by the midplane, to  
 670 show liquid distribution extents with clarity. Black vertical lines are placed every 1 mm, starting from the orifice entrance.  
 671 The flow moves from left to right.  
 672

673 Figure 18 and Figure 19 show the average liquid fraction in slices across the length of the orifice  
 674 for high and low cavitation numbers, based on numerical and experimental data. In general, average  
 675 cavitation pattern in both cases shows similar features. Near the orifice entrance there is an attached  
 676 vapour sheet, while further downstream there is a detached structure. It has to be kept in mind, that  
 677 slices from the CT scans show an artefact at the centre of the orifice, due to the reconstruction  
 678 algorithm employed; this has been omitted from visualization, hence the white dot appearing at origin.  
 679 The experiment shows a more distinct cavitation ring around the entrance of the orifice, both at high  
 680 and low cavitation number operation. At low cavitation number, the average vapour cavity predicted  
 681 by the simulation shows two distinct cavitation lobes, with diffuse vapour between them. In the  
 682 experiment, the two cavitation lobes are not clearly visible, though the left part of the cross-section  
 683 shows a higher vapour content than the right; this is something found in the simulation results as well  
 684 (see Figure 18 at 1 mm). Simulation results at high cavitation number still show the two distinct lobes  
 685 (see Figure 19 at 1 mm), though the two lobes quickly merge, forming an agglomeration (see Figure  
 686 19, from 4mm and further downstream). Experimental results show a more disperse and noisy  
 687 cavitation distribution, though the general pattern and extent agrees with the simulation. Note that at  
 688 high cavitation number, as well, the left part of the cross-section shows a higher occupation by  
 689 cavitation (see e.g. Figure 19 at 4 or 6 mm). Cavitation distribution is not entirely symmetric, due to  
 690 the geometric features of the orifice geometry used, while the asymmetry is lower at the highest  
 691 cavitation number operation ( $Cn = 2.18$ ). The two vapour lobe locations predicted by the simulation  
 692 coincide with the counter-rotating vortices of the average flow field (see also Figure 10), occurring  
 693 inside the orifice.



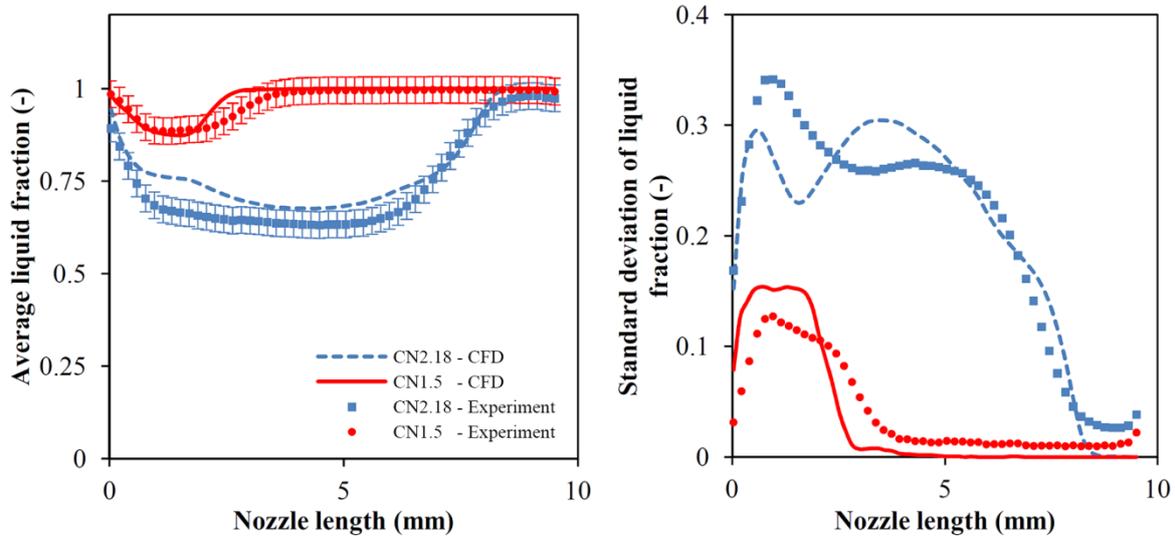
694  
 695  
 696  
 697  
 FIGURE 18. Average liquid fraction distribution at the orifice cross-section for various locations, for  $Cn = 1.5$ , simulation and experiment (micro-CT) density measurement. The number at the upper left corner indicates the distance from the orifice entrance. Upstream view inside the orifice ( $AA'$  view, see Figure 4,  $A$  being at the top and  $A'$  at the bottom).



699  
 700  
 701  
 702  
 FIGURE 19. Average liquid fraction distribution at the orifice cross-section for various locations, for  $Cn = 2.18$ , simulation and experiment (micro-CT) density measurement. The number at the upper left corner indicates the distance from the orifice entrance. Upstream view inside the orifice ( $AA'$  view, see Figure 4,  $A$  being at the top and  $A'$  at the bottom).

703 To make a more clear comparison between the simulation and the experimental distribution of the  
 704 liquid fraction, results of average and standard deviation distribution of the liquid fraction at slices  
 705 across the length of the orifice are shown in Figure 20. The average liquid fraction expresses the  
 706 blockage of the flow by cavitation, whereas standard deviation shows the dispersion of vaporous  
 707 structures across the cross-section of the orifice. At low cavitation number, simulation results show a  
 708 slight under prediction of the vaporous cavity length, though minimum liquid fraction is correctly  
 709 predicted at  $\sim 88\%$ . Still, the numerical results are within the validation uncertainty, which is  $\sim \pm 7\%$ ,  
 710 including experimental errors ( $\sim \pm 4\%$ ) and numerical uncertainty as determined from the sensitivity  
 711 analysis and grid dependence on average cavity volume (tables 3 and 4). The standard deviation of the  
 712 liquid fraction shows a peak at 1 mm downstream the entrance of the orifice, while the simulation  
 713 predicts a slightly higher standard deviation. Moving further downstream of the cavitation region, the  
 714 standard deviation drops rapidly. It has to be highlighted that simulation predicts a standard deviation  
 715 very close to zero from 5 mm downstream the entrance until the orifice exit, whereas in the  
 716 experiment there is a non-zero standard deviation of liquid fraction until the orifice exit; this is an  
 717 indication of noise from the micro-CT 3D reconstruction of cavitation. On the other hand, simulation

718 tends to over predict slightly the average liquid volume fraction at high cavitation number, while  
 719 predicting the correct average cavity length. The mismatch is mainly pronounced at 1.4 mm  
 720 downstream the orifice entrance. Still, the minimum liquid volume fraction is predicted within the  
 721 validation uncertainty. The minimum liquid fraction predicted by the simulation is 67% whereas in  
 722 the experiment it is found to be 63%. Both experimental and numerical results of the standard  
 723 deviation of liquid fraction at the cross-section of the orifice show two peaks, one at  $\sim 0.7$  mm and  
 724 one at  $\sim 4$  mm downstream the entrance. The simulation, though, under predicts the first peak,  
 725 whereas over predicts the other one. The shape of the standard deviation variation at the cavity closure  
 726 is in a close agreement with the experiment.  
 727



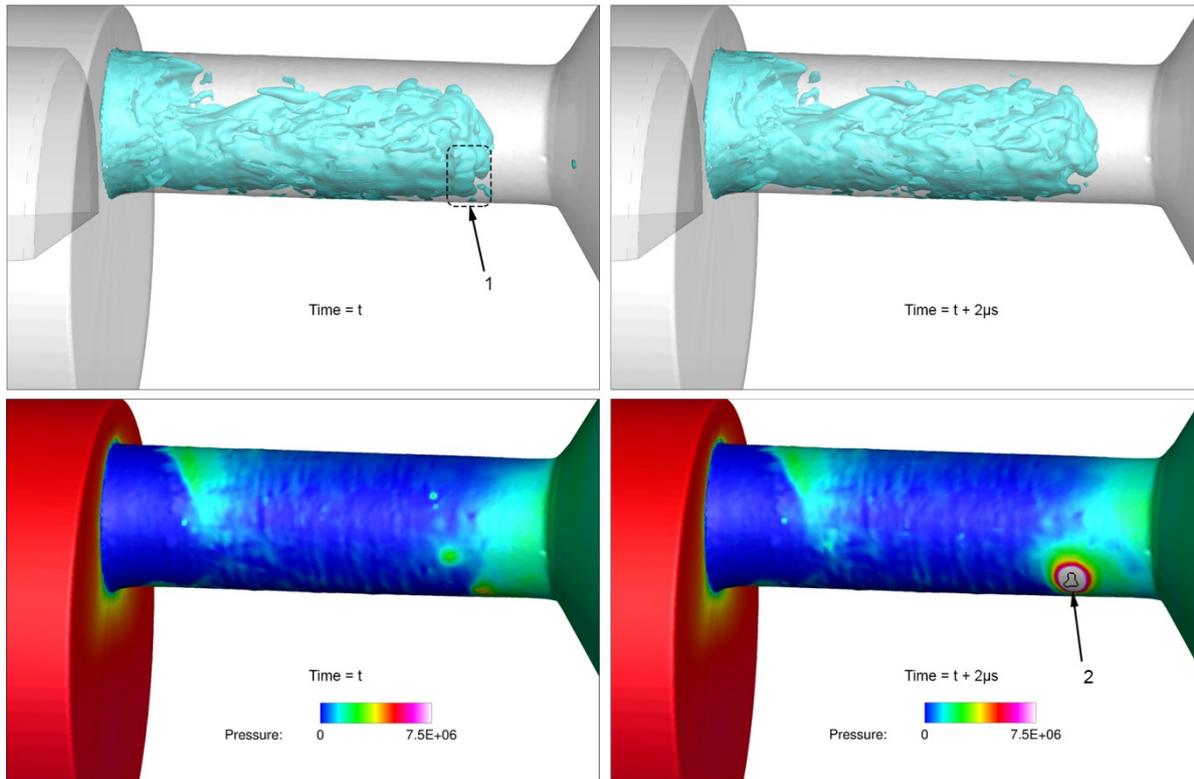
728  
 729 FIGURE 20. Average and standard deviation of liquid fraction at slices across the orifice length.  
 730

731 CT scans of the described configuration indicated that over time the orifice geometry was  
 732 changing; it was apparent that exposure to cavitation caused erosion to the material. The simulation  
 733 with the proposed cavitation model can shed light in the formation of erosion. Indeed, the collapse of  
 734 arbitrarily shaped vaporous structures can be tracked by the proposed cavitation model, since it  
 735 predicts very high pressures at the areas of collapse. Indicative flow instances from the simulation are  
 736 provided below in Figure 21 for  $Cn = 2.18$ , showing density isosurface inside the orifice and the  
 737 pressure field on the wall. In the instances provided, the collapse of a cavitation structure at the  
 738 trailing edge of the cavitation cloud is visible, see 1. This collapse induces a very high pressure to the  
 739 nearby wall surface, see 2. The violence and intensity of the collapse may be understood considering  
 740 that pressure, locally, rises from vapour pressure ( $\sim 0.172$  bar) to 200 bar within  $2 \mu s$ . The location of  
 741 such high pressure peaks can provide information on the erosion prone areas, for more information  
 742 see also Mihatsch, Schmidt et al. (2015).

743 Erosion prone areas have been identified through the difference of the orifice surface before and  
 744 after exposure to cavitation. Indicative results are shown in Figure 22 (a). Erosion develops in two  
 745 distinct zones, at the orifice sides exposed to cavitation: the first zone spans from  $\sim 1.4$  mm until 3.7  
 746 mm downstream the entrance and the second zone from  $\sim 5.4$  mm until 8.8 mm downstream the  
 747 entrance. Undeniably, at low cavitation number conditions cavitation is limited near the orifice  
 748 entrance, thus it is certain that the far downstream erosion site is caused during the high cavitation  
 749 number operation.

750 Numerical simulation can provide insight in the erosion development, by recording areas of high  
 751 pressures, caused by the collapse of vapour structures, as illustrated in Figure 21, Figure 23, Figure  
 752 24. Indeed, the accumulated local maximum of the instantaneous pressure field at every

753 computational cell shows a distinct pattern which can be well correlated with Figure 22 (a). At low  
 754 cavitation number high pressure peaks are detected close to the orifice entrance, starting at  $\sim 1$  mm  
 755 until  $\sim 2.5$  mm downstream the entrance. At high cavitation numbers, see Figure 22 (b), the high  
 756 pressure peaks are clustered in two zones, one from  $\sim 1$  mm until  $\sim 3.5$  mm and another from  $\sim 5.5$   
 757 mm until  $\sim 8.5$  mm downstream the entrance. These two zones are associated with the collapse  
 758 locations of the attached cavity and cavity shedding, as shown in Figure 7, and show a correlation to  
 759 the average turbulent kinetic energy distribution, shown in Figure 13.  
 760



761  
 762 FIGURE 21. Indicative instances of the cavitating flow, at  $Cn = 2.18$  ( $p_{up} = 55$  bar,  $p_{down} = 17.4$  bar). Top line: density  
 763 isosurface (75% liquid), bottom: pressure on the wall. Flow is moving from left to right. Notice the cavitation structure  
 764 marked with (1), which, due to its collapse, causes a pressure pulse on the wall at (2). The collapse occurs within  $2 \mu s$ .  
 765 Pressure in the area enclosed within the black line in (2) rises to more than 200 bar. Pressure units are Pascal.  
 766

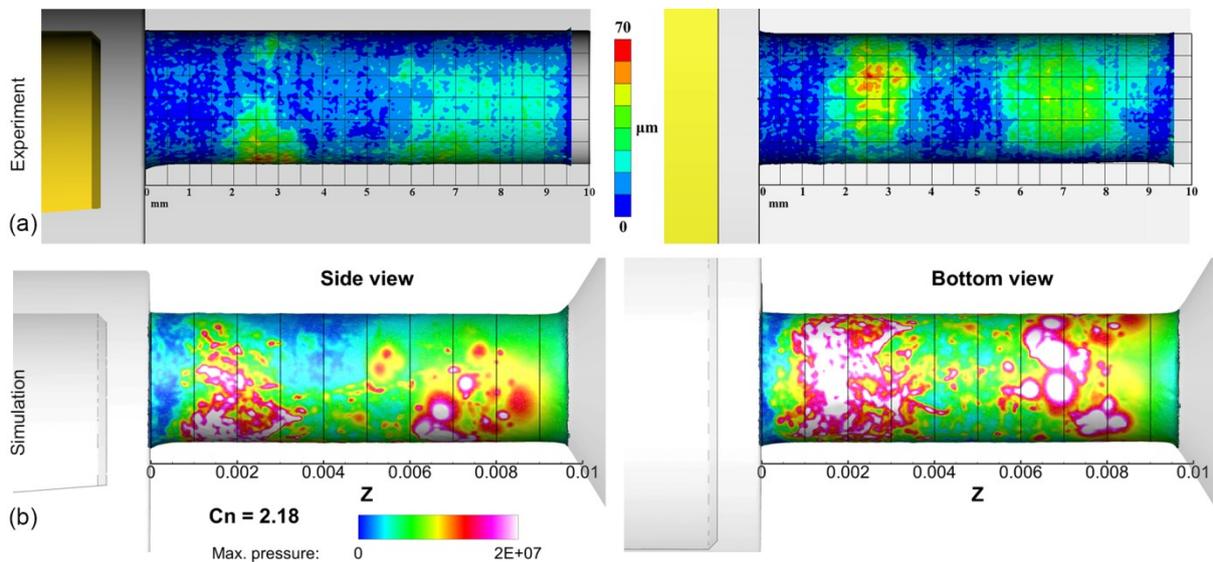
767 The pressure peaks indicated with white colour in Figure 22 (b) are of pressure levels equal to or  
 768 higher than 200 bar, which is higher or equal to  $\sim 20\%$  of the nominal yield stress of the PEEK  
 769 material used. Higher pressure magnitude ( $>500$  bar) peaks are concentrated mainly at  $\sim 2$  mm and  $\sim$   
 770  $7$  mm downstream the entrance; note that wall pressure may locally reach even 750 bar and pressure  
 771 inside the liquid bulk may reach 1850 bar. The fact that the predicted wall pressures are lower than the  
 772 nominal yield stress does not necessarily mean a weakness of the described model; the fatigue failure  
 773 stress may drop significantly for large numbers of loading cycles (Budynas & Nisbett 2011). As  
 774 mentioned before, an indicative cavity shedding time scale is  $\sim 78-95 \mu s$ ; considering that erosion was  
 775 observed over the course of 44 hours testing, the number of loading cycles is of the order of  $10^9$ . At  
 776 such numbers of loading cycles the material yield stress may drop by more than 50% comparing to the  
 777 nominal one. Also, since the smallest vapour scale cannot be smaller than the cell size, the peak  
 778 pressure is affected by the spatial and temporal resolution; however the location of the pressure peaks  
 779 can provide an indication of erosion sites. For more information the interested reader is referred to  
 780 Schmidt, Mihatsch et al. (2014) and Mihatsch, Schmidt et al. (2015).

781 A more detailed view of the collapse mechanism appearing in the two erosion sites is provided in

782 Figure 23 and Figure 24. As shown in Figure 23, the mechanism behind the erosion close to the  
 783 entrance (the one indicated in Figure 22, starting from 1 to 3.5 mm) is the collapse of the attached  
 784 sheet cavities. Such attached cavities may be disperse and small in size, as e.g. indicated with number  
 785 1, producing at their collapse a single pressure pulse. Pressure pulses are also found from the collapse  
 786 of larger attached cavitation sheets, as indicated with number 2; in that case a series of pressure pulses  
 787 are produced at the edge of the receding sheet cavity, as indicated by the dashed regions denoted with  
 788 the number 2'. On the other hand, the mechanism of erosion at the region towards the exit of the  
 789 orifice (the one indicated in Figure 22, starting from 5.5 to 8.5 mm) is mainly the collapse of detached  
 790 cavities. As shown in Figure 24, both small scale (indicated with numbers 1, 2) and large scale  
 791 (indicated with numbers 4, 5) cavitation structures cause pressure peaks in the nearby walls, albeit the  
 792 large scales produce higher magnitudes. Apart from detached cavities, there is evidence that small  
 793 cavity pockets, formed at roughness elements of the wall, may collapse causing local high pressures  
 794 as well, see 3 in Figure 24. At both erosion sites, cavity collapses may be either individual (i.e. a  
 795 single cavity collapse) or in the form of a "collapse cascade", where a number of collapses is observed  
 796 in rapid succession. Examples of such "collapse cascades" are e.g. the receding of the cavity sheet at 2  
 797 in the sequence of Figure 23 and collapses 4 and 5 in the sequence of Figure 24.

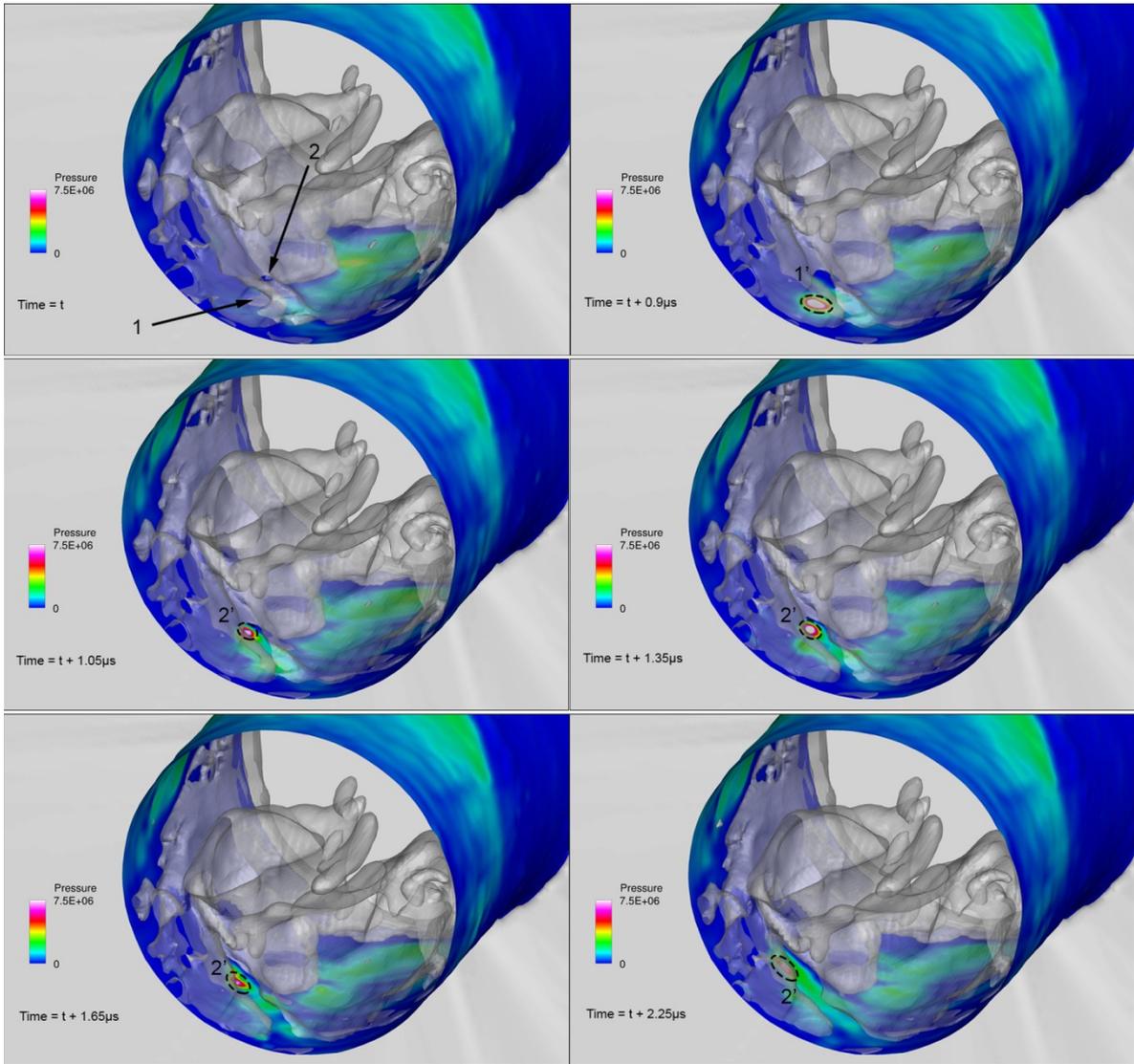
798 An alternative way of depicting the collapse distribution of cavitation structures is through the  
 799 volume representation of the collapse epicentres. The distribution of the collapses is shown in Figure  
 800 25 as a side view (similar to Figure 22) and along the orifice cross-section. Pressure peaks located  
 801 near the orifice entrance (0-5 mm downstream the entrance, first erosion site when moving with the  
 802 flow) are mainly concentrated very close to the walls, indicating the collapse of attached cavities. On  
 803 the other hand, collapse epicentres are much more scattered all across the orifice cross-section at the  
 804 second collapse site (5-10 mm downstream the throttle entrance), indicating the collapse of detached  
 805 cavities that travel in the bulk of the fluid.

806  
807  
808



809  
810  
811  
812  
813  
814

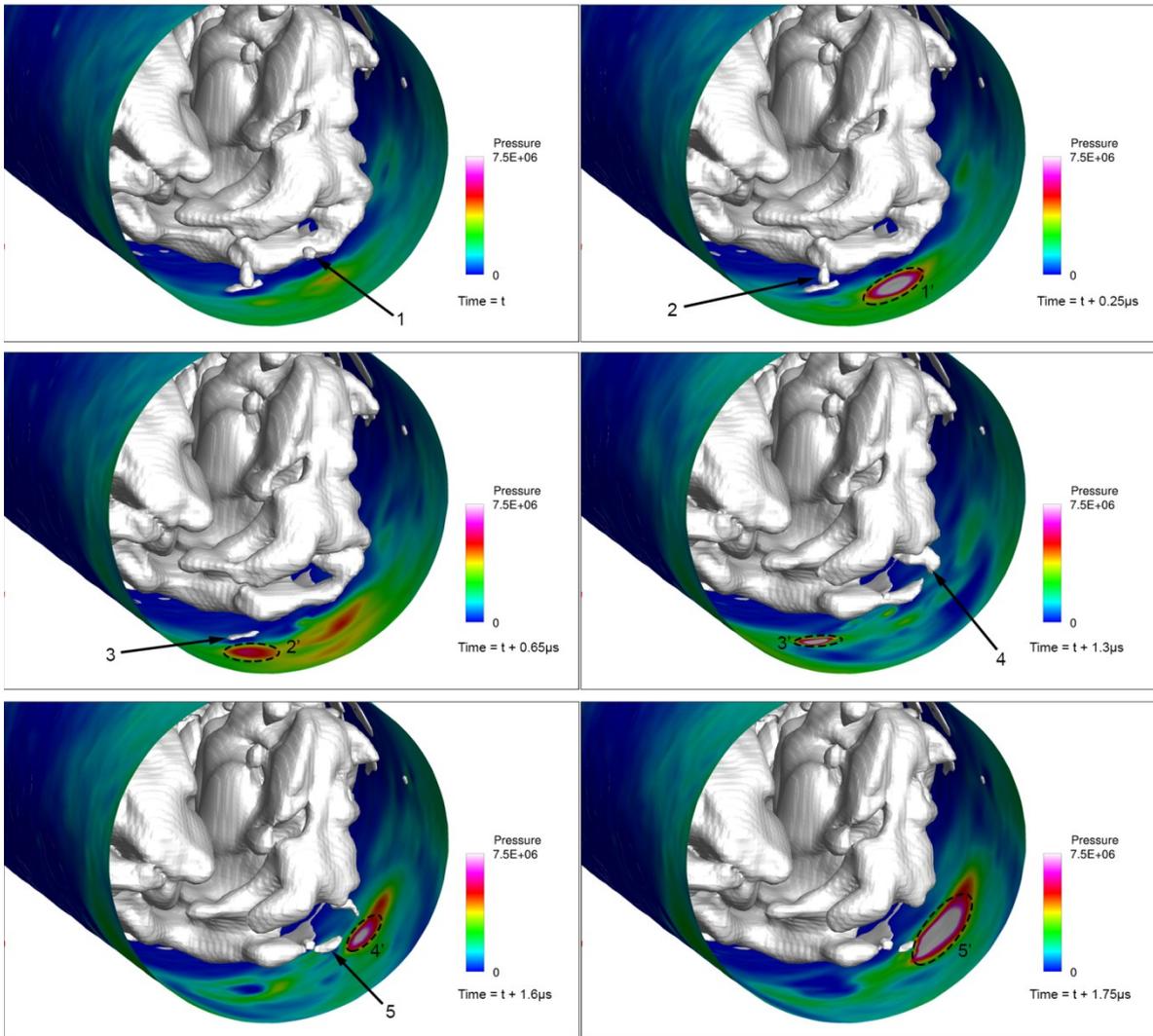
FIGURE 22. (a) Erosion location as determined from CT scans of the geometry, expressed as the difference of the orifice geometry prior and after exposure to cavitation. (b) Accumulated maximum of pressure (i.e. pressure peaks) over time on the wall surface for  $Cn = 2.18$ , simulation results. Pressure units are in SI, i.e. Pa. See also supplementary material for a 3D distribution of pressure peaks around the orifice.



815  
816  
817  
818  
819  
820

FIGURE 23. Instances of collapsing cavitation structures, denoted with the isosurface of the vapour fraction at 75% liquid (semi-transparent grey/white), with the corresponding pressure field on the wall. Collapsing structures are indicated by an arrow and the resulting pressure pulse with a dashed line. Upstream view inside the orifice ( $AA'$  view, see Figure 4) at 3.5 mm from the orifice entrance. The rest of the orifice and collector volume have been blanked to provide an unobstructed view to the collapse process. Units are in SI (pressure in Pa).

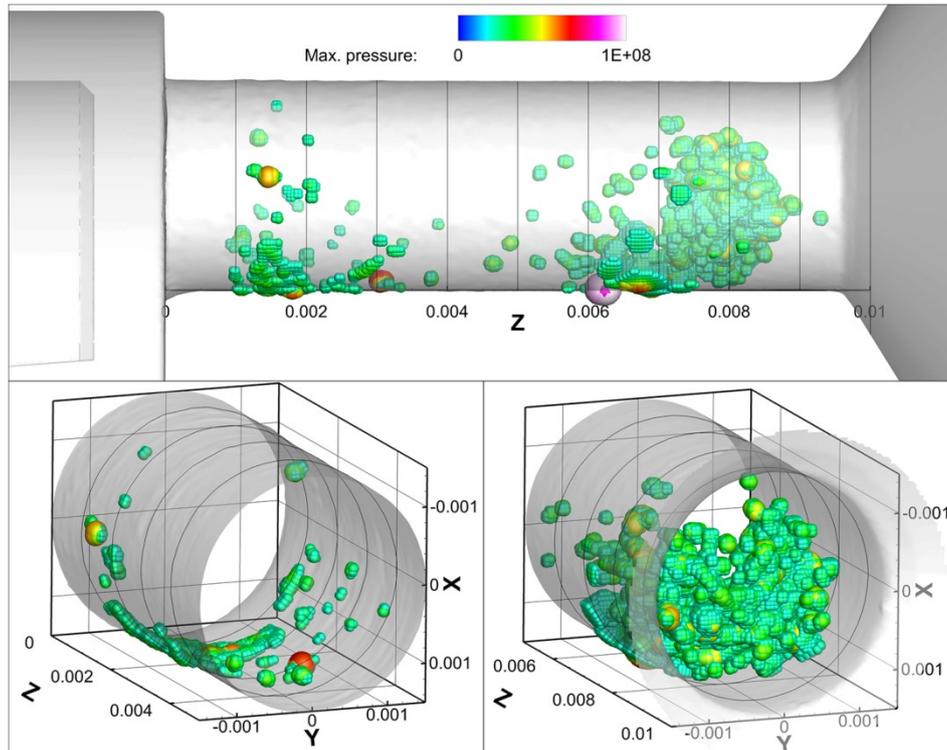
821  
822  
823  
824



825  
 826  
 827  
 828  
 829  
 830

FIGURE 24. Instances of collapsing cavitation structures, denoted with the isosurface of the vapour fraction at 75% liquid (grey/white), with the corresponding pressure field on the wall. Collapsing structures are indicated by an arrow and the resulting pressure pulse with a dashed line. Upstream view inside the orifice ( $AA'$  view, see Figure 4) at 9 mm from the entrance. The collector volume has been blanked to provide an unobstructed view to the collapse process. Units are in SI (pressure in Pa).

831



832  
833  
834  
835  
836

Figure 25. Collapse epicentres for  $Cn = 2.18$ , marked as scatter plot (spheres) inside the volume of the orifice; only collapse pressures equal or higher to 200 bar are shown. The size of the spheres is proportional to the pressure peak magnitude. Note that in the region between 0-5 mm downstream the entrance, collapse epicentres are very close to walls, whereas between 5-10 mm collapse epicentres are scattered all across the cross-section of the orifice.

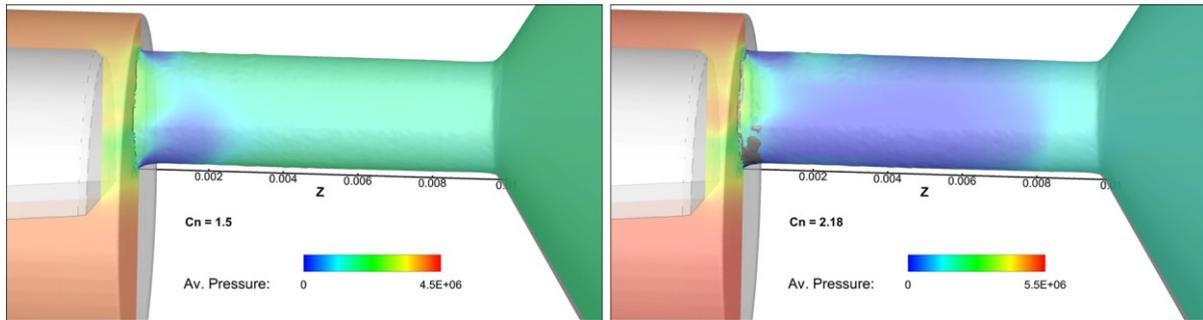
837

## 838 5. Discussion

839 The examined case illustrates the complexity of the flow in a relatively simple configuration that  
840 has both industrial and scientific interest due to the wealth of flow features that are occurring. The off-  
841 centred orifice geometry causes the formation of an asymmetric cavitation pattern and cavitating  
842 vortices, that are similar to those occurring in actual diesel injection systems. An important parameter,  
843 that affects all micro-fluidic devices, is the accuracy of the geometry representation. Unfortunately  
844 manufacturing constraints impose limits to the fidelity of the geometry. On the other hand, precise  
845 geometry representation is crucial for capturing accurately the involved flow phenomena. Early  
846 experimental and numerical investigations indicated that flow rate through the cylindrical orifice  
847 (idealized geometry) was considerably lower than the actual orifice geometry, due to the presence of  
848 sharp corners at the orifice entrance.

849 The main features of the cavitation structure are: (1) attached cavitation sheets around the orifice  
850 entrance, (2) a pair of large counter-rotating cavitating vortices inside the orifice, which may extend  
851 up to the needle, and (3) cavitation structures that detach from the aforementioned structures and  
852 travel downstream. The flow inside the orifice shows strong turbulent characteristics in the form of  
853 highly unstable coherent vortical patterns. All these turbulent features are closely related to cavitation  
854 formation. The cores of large scale vortices may act as cavitation nucleation regions, enabling the  
855 transient formation and propagation of cavitation structures in areas where time-averaged pressure  
856 may be well above vapour pressure. Examples of such cases are the formation of cavitating vortices,  
857 starting from the needle of the examined device, or the shedding of cavities inside the collector  
858 volume. Another example is shown in Figure 26, with the isosurface of the time-averaged pressure  
859 distribution at a pressure level equal to the vapour pressure. The region with time-averaged pressure  
860 less or equal to vapour is very limited and located near the entrance of the orifice, with no

861 resemblance to the time-averaged liquid fraction distribution shown in Figure 17.  
862



863  
864  
865  
866  
867  
868  
869  
870  
871  
872  
873  
874  
875  
876  
877  
878  
879  
880  
881  
882  
883  
884  
885  
886  
887  
888  
889  
890  
891  
892  
893  
894  
895  
896  
897  
898  
899  
900

FIGURE 26. Time-averaged pressure field at the midplane of the geometry for low and high cavitation numbers. The black isosurface denotes the area where time-averaged pressure is less than or equal to vapour pressure; for  $Cn = 1.5$  it is barely visible. Pressure units are Pa. The flow moves from right to left.

867  
868 Flow visualization of the experiment clearly shows unsteady flow features, but only in a  
869 qualitative manner. Moreover, only a fraction of these features is visible, due to the optical blockage  
870 of the passage from cavitation presence. X-ray micro-CT enabled a quantitative description of the  
871 flow field, giving volumetric density distribution and erosion information. The discussed numerical  
872 methodology with the proposed cavitation model and the DES turbulence model, is capable of  
873 predicting accurately many of the flow features, including cavitating vortices, providing information  
874 on the highly turbulent flow field and explaining the erosion formation by tracking the collapse of  
875 vaporous structures.

876 Minor discrepancies are observed at the cavity length mainly at low cavitation number. Reasons of  
877 the discrepancies are discussed below:

878 - At low cavitation number, cavitation effects are much more sensitive to small variations of the  
879 flow rate. Moreover, cavitation is more sensitive to the existence of turbulent structures that have not  
880 been resolved with the numerical resolution employed; as the turbulent model used in the present  
881 study is a RANS/LES hybrid, the computational mesh employed was not intended to resolve all  
882 turbulent structures, not to mention that such an effort would involve an immensely higher  
883 computational cost.

884 - It is reminded that not all wall features were resolved; inherently, due to the discretization  
885 procedure, any wall features below the discretization resolution have been smoothed out. Moreover,  
886 erosion development in the experiment alters wall features over time.

887 - Furthermore, parts upstream the annular passage (i.e. the flow splitting to three inlet pipes and  
888 funnel shape contraction) have been excluded, though such geometric features may induce additional  
889 turbulence at the inlet of the orifice, e.g. pipe bends are known to introduce streamwise vortices  
890 (Tunstall & Harvey 1968).

891 - Diesel properties were obtained from a property library, however they do not necessarily  
892 correspond to exactly the actual Diesel properties in the experiment. Diesel fuel samples from  
893 different sources have slightly different composition and properties. Moreover, even though Diesel  
894 fuel was periodically replaced to maintain its quality, the fuel properties do not remain constant after  
895 exposure to cavitation, see also Lockett & Jeshani (2013). As observed in the aforementioned  
896 experimental investigation, Diesel colour changed towards a more yellowish hue after exposure to  
897 cavitation, indicating slight changes in its composition.

898 - The experimental errors should not be considered negligible; as indicated in Figure 20, errors in  
899 average volume fraction are of  $\sim \pm 0.04$ . Moreover, noise is visible in the average liquid fraction  
900 isosurfaces and artefacts in the liquid fraction distribution due to the 3D reconstruction, can be

901 identified (Mitroglou, Lorenzi et al. 2015). In total, the discrepancies between numerical and  
902 experimental results are within or very close to the validation uncertainty margin, both for flow rate  
903 and average cavity volume (and liquid volume fraction).

904

## 905 **6. Conclusion**

906 In the present work the cavitating flow inside an orifice was analyzed with experimental  
907 techniques and numerical tools at low and high cavitation numbers. Shadowgraphy shows cavitation  
908 features, like cavitating vortices extending to the needle surface, but optical access inside the orifice is  
909 limited due to view blockage by cavitation. X-ray micro-CT scans on the other hand, can provide  
910 quantitative information on the average cavitation distribution inside the orifice.

911 Numerical simulations were used to predict unsteady features of the flow, vapour collapse  
912 locations and to derive average distribution of the liquid fraction. Three types of cavitation structures  
913 have been identified, namely attached cavitation at the orifice entrance, large scale cavitating vortices  
914 spanning from the needle and extending inside the orifice and smaller detached cavities travelling  
915 further downstream. The flow field is turbulent in the whole test section, however turbulence in the  
916 orifice region is affected by cavitation structures. Turbulence is dampened in areas of dense  
917 cavitation, whereas it is found to increase in areas of cavitation collapse. The increase in flow  
918 turbulence in areas of cavitation collapse is related to the absence of the turbulence damping  
919 mechanism and to the very high velocities and pressures induced by the collapse of cavitation  
920 structures.

921 High pressure locations, which indicate areas of vapour collapse, average liquid fraction  
922 distribution and average volumetric flow rate through the orifice were used to validate the results of  
923 the numerical simulation. Examination of the flow field in the vicinity of these erosion sites indicates  
924 that the one located near the orifice entrance is caused by the collapse of attached cavity sheets,  
925 whereas the other, towards the orifice exit, is caused by the collapse of detached cavities. The  
926 agreement between simulation and experiment, in all aforementioned terms, is within validation  
927 uncertainty, taking into account all uncertainties, numerical and experimental. Thus, despite the  
928 inherent complexities of the particular flow, a reasonable reproduction of the flow pattern was  
929 obtained, close or within the uncertainty margin.

930

931 The research leading to these results has received funding from the People Programme (IAPP  
932 Marie Curie Actions) of the European Union's Seventh Framework Programme FP7/2007-2013/  
933 under REA grant agreement n. 300410 and n. 324313.

934

## 935 REFERENCES

936

937 ANDRIOTIS, A., GAVAISES, M. AND ARCOUMANIS, C. 2008 Vortex flow and cavitation in Diesel  
938 injector nozzles. *J. Fluid Mech.* 610: 195-215.

939 ARNDT, R. E. A., ARAKERI, V. H. AND HIGUCHI, H. 1991 Some observations of tip-vortex cavitation.  
940 *J. Fluid Mech.* 229: 269-289.

941 BAKIR, F., REY, R., GERBER, A. G., BELAMRI, T. AND HUTCHINSON, B. 2004 Numerical and  
942 Experimental Investigations of the Cavitating Behavior of an Inducer. *Int. J. Rotat. Mach.* 10:  
943 15-25.

944 BATCHELOR, G. K. 2000 *An Introduction to Fluid Dynamics*. Cambridge, UK, Cambridge University  
945 Press.

946 BATTISTONI, M., DUKE, D. J., SWANTEK, A. B., TILOCCO, Z. F., POWELL, C. F. AND SOM, S. 2015  
947 Effects of noncondensable gas on cavitating nozzles. *Atomization Sprays* 25(6): 453-483.

948 BAUER, D., CHAVES, H. AND ARCOUMANIS, C. 2012 Measurements of void fraction distribution in  
949 cavitating pipe flow using x-ray CT. *Meas. Sci. Technol.* 23(5): 10.

- 950 BEVINGTON, P. R. & ROBINSON, K. D. 2003 *Data Reduction and Error Analysis for the Physical*  
951 *Sciences*, McGraw Hill.
- 952 BRENNEN, C. 1995 *Cavitation and Bubble Dynamics*, Oxford University Press.
- 953 BUDYNAS, R. & NISBETT, K. 2011 *Mechanical Engineering Design*, McGraw-Hill Education.
- 954 CARLTON, J. 2012 *Marine Propellers and Propulsion*. Oxford, UK, Elsevier, Butterworth-  
955 Heinemann.
- 956 CHEN, Z. J. & PRZEKWAŚ, A. J. 2010 A coupled pressure-based computational method for  
957 incompressible/compressible flows. *J. Comput. Phys.* 229: 9150–9165.
- 958 CIGNONI, P., CALLIERI, M., CORSINI, M., DELLEPIANE, M., GANOVELLI, F. AND RANZUGLIA, G.  
959 MeshLab. v1.3.3, 2014, from <http://meshlab.sourceforge.net/>.
- 960 COLEMAN, H. W. & STEELE, W. G. 2009 *Experimentation, validation and uncertainty analysis for*  
961 *engineers*. Hoboken, New Jersey, John Wiley & Sons.
- 962 COUTIER-DELGOSHA, O., REBOUD, J. L. AND DELANNOY, Y. 2003 Numerical simulation of the  
963 unsteady behaviour of cavitating flows. *Int. J. Numer. Meth. Fl.* 42: 527-548.
- 964 DECAIX, J., BALARAC, G., DREYER, M., FARHAT, M. AND MÜNCH, C. 2015 RANS and LES  
965 computations of the tip-leakage vortex for different gap widths. *J. Turbul.* 16(4): 309-341.
- 966 DUKE, D. J., KASTENGREN, A. L., SWANTEK, A. B., SOVIS, N., FEZZAA, K., NEROORKAR, K., MOULAI,  
967 M., POWELL, C. F. AND SCHMIDT, D. P. 2014. Comparing Simulations and X-ray Measurements  
968 of a Cavitating Nozzle. *ILASS-Americas 26th Annual Conference on Liquid Atomization and*  
969 *Spray Systems*, Portland, Oregon, U.S.
- 970 DUPLAA, S., COUTIER-DELGOSHA, O., DAZIN, A. AND BOIS, G. 2013 X-Ray Measurements in a  
971 Cavitating Centrifugal Pump During Fast Start-Ups. *J. Fluids Eng.* 135(4): 041204-041204.
- 972 DUTTWEILER, M. E. & BRENNEN, C. E. 2002 Surge instability on a cavitating propeller. *J. Fluid*  
973 *Mech.* 458: 133-152.
- 974 EDELBAUER, W., STRUCL, J. AND MOROZOV, A. 2016 Large Eddy Simulation of cavitating throttle  
975 flow: SIMHYDRO 2014. Advances in Hydroinformatics, Part III. P. Gourbesville, J. A. Cunge  
976 and G. Caignaert. Singapore, Springer Singapore: 501-517.
- 977 EGLER, W., GIERSCH, J. R., BOECKING, F., HAMMER, J., HLOUSEK, J., MATTES, P., PROJAHN, U.,  
978 URNER, W. AND JANETZKY, B. 2010 Fuel Injection Systems. Handbook of Diesel Engines. K.  
979 Mollenhauer and H. Tschöke, Springer-Verlag Berlin Heidelberg: 127-174.
- 980 FELDKAMP, L. A., DAVIS, L. C. AND KRESS, J. W. 1984 Practical cone-beam algorithm. *J. Opt. Soc.*  
981 *Am. A* 1(6): 612-619.
- 982 FERZIGER, J. H. & PERIC, M. 2002 *Computational Methods for Fluid Dynamics*. Berlin, Germany  
983 Springer-Verlag Berlin Heidelberg.
- 984 FRANC, J.-P. & MICHEL, J.-M. 2005 *Fundamentals of Cavitation*, Kluwer Academic Publishers.
- 985 GANESH, H., MĀKIHARJU, S. A. AND CECCIO, S. 2016 Bubbly shock propagation as a mechanism for  
986 sheet-to-cloud transition of partial cavities. *J. Fluid Mech.* 802: 37-78.
- 987 GIANNADAKIS, E., GAVAISES, M. AND ARCOUMANIS, C. 2008 Modelling of cavitation in diesel  
988 injector nozzles. *J. Fluid Mech.* 616.
- 989 GNANASKANDAN, A. S. & KRISHNAN, M. 2016 Numerical investigation of near-wake characteristics  
990 of cavitating flow over a circular cylinder. *J. Fluid Mech.* 790: 453-491.
- 991 GREEN, M. A., ROWLEY, C. W. AND HALLER, G. 2007 Detection of Lagrangian Coherent Structures in  
992 3D Turbulence. *J. Fluid Mech.* 572: 111-120.
- 993 HALLER, G. 2005 An objective definition of a vortex. *J. Fluid Mech.* 525: 1-26.
- 994 HULT, J., SIMMANK, P., MATLOK, S., MAYER, S., FALGOUT, Z. AND LINNE, M. 2016 Interior flow and  
995 near-nozzle spray development in a marine-engine diesel fuel injector. *Exp. Fluids* 57(4): 1-19.
- 996 IAEA 2008 Neutron Imaging: A Non-Destructive Tool for Materials Testing. Nuclear analytical  
997 techniques. Austria, Vienna, International Atomic Energy Agency. IAEA-TECDOC-1604.
- 998 KINI, V., BACHMANN, C., FONTAINE, A., DEUTSCH, S. AND TARBELL, J. M. 2000 Flow Visualization  
999 in Mechanical Heart Valves: Occluder Rebound and Cavitation Potential. *Ann. Biomed. Eng.*  
1000 28: 431-441.
- 1001 KOLEV, N. 2007 *Multiphase Flow Dynamics 3*, Springer Berlin Heidelberg.
- 1002 KOUKOUVINIS, P. & GAVAISES, M. 2015 Simulation of throttle flow with two phase and single phase  
1003 homogenous equilibrium model. *J. Phys.: Conf. Ser.* 656(1): 012086.

- 1004 KOUKOUVINIS, P., GAVAISES, M., LI, J. AND WANG, L. 2016 Large Eddy Simulation of Diesel injector  
1005 including cavitation effects and correlation to erosion damage. *Fuel* 175: 26-39.
- 1006 KOUKOUVINIS, P., NASERI, H. AND GAVAISES, M. 2016 Performance of Turbulence and Cavitation  
1007 Models in Prediction of Incipient and Developed Cavitation. *Int. J. Engine Res.*
- 1008 LI, S. 2000 *Cavitation of Hydraulic Machinery*. London, Imperial College Press.
- 1009 LINDAU, O. & LAUTERBORN, W. 2003 Cinematographic observation of the collapse and rebound of a  
1010 laser-produced cavitation bubble near a wall. *J. Fluid Mech.* 479: 327- 348.
- 1011 LOCKETT, R. D. & JESHANI, M. 2013 An experimental investigation into the effect of hydrodynamic  
1012 cavitation on diesel. *Int. J. Engine Res.* 14(6): 606-621.
- 1013 MAUGER, C., MÉÈS, L., MICHARD, M., AZOUZI, A. AND VALETTE, S. 2012 Shadowgraph, Schlieren  
1014 and interferometry in a 2D cavitating channel flow. *Exp. Fluids* 53(6): 1895-1913.
- 1015 MIHATSCH, M. S., SCHMIDT, S. J. AND ADAMS, N. A. 2015 Cavitation erosion prediction based on  
1016 analysis of flow dynamics and impact load spectra. *Phys. Fluids* 27(103302).
- 1017 MITROGLOU, N., LORENZI, M., SANTINI, M., GAVAISES, M. AND ASSANIS, D. 2015 Application of  
1018 cone-beam micro-CT on high-speed Diesel flows and quantitative cavitation measurements. *J.*  
1019 *Phys.: Conf. Ser.* 656(1)(012094).
- 1020 MITROGLOU, N., MCLORN, M., GAVAISES, M., SOTERIOU, C. AND WINTERBOURNE, M. 2014  
1021 Instantaneous and ensemble average cavitation structures in Diesel micro-channel flow orifices.  
1022 *Fuel* 116: 736-742.
- 1023 MOCKETT, C. 2007 *A comprehensive study of detached-eddy simulation*, TU Berlin  
1024 Universitätsbibliothek
- 1025 MOON, S., LIU, Z., GAO, J., DUFRESNE, E., FEZZAA, K. AND WANG, J. 2010. Ultrafast X-ray Phase-  
1026 Contrast Imaging of High-Speed Fuel Sprays from a Two-Hole Diesel Nozzle. *ILASS*  
1027 *Americas, 22nd Annual Conference on Liquid Atomization and Spray Systems*, Cincinnati,  
1028 Ohio, US.
- 1029 ÖRLEY, F., HICKEL, S., SCHMIDT, S. J. AND ADAMS, N. A. 2016 Large-Eddy Simulation of turbulent,  
1030 cavitating fuel flow inside a 9-hole Diesel injector including needle movement. *Int. J. Engine*  
1031 *Res.*
- 1032 PENNINGS, P. C., BOSSCHERS, J., WESERWEEL, J. AND VAN TERWISGA, T. J. C. 2015 Dynamics of  
1033 isolated vortex cavitation. *J. Fluid Mech.* 778: 288-313.
- 1034 REID, B. A., HARGRAVE, G. K., GARNER, C. P. AND WIGLEY, G. 2010 An investigation of string  
1035 cavitation in a true-scale fuel injector flow geometry at high pressure. *Phys. Fluids* 22(3).
- 1036 ROACHE, P. J. 1997 Quantification of uncertainty in computational fluid dynamics. *Ann. Rev. Fluid*  
1037 *Mech.* 29: 123-160.
- 1038 SCHMIDT, S. J., MIHATSCH, M. S., THALHAMER, M. AND ADAMS, N. A. 2014 Assessment of Erosion  
1039 Sensitive Areas via Compressible Simulation of Unsteady Cavitating Flows. *Advanced*  
1040 *Experimental and Numerical Techniques for Cavitation Erosion Prediction*. K.-H. Kim, G.  
1041 Chahine, J.-P. Franc and A. Karimi. Dordrecht, Springer Netherlands: 329-344.
- 1042 SCHNERR, G. H. & SAUER, J. 2001. Physical and Numerical Modeling of Unsteady Cavitation  
1043 Dynamics. *Fourth International Conference on Multiphase Flow*, New Orleans, USA.
- 1044 SHUR, M. L., SPALART, P. R., STRELETS, M. K. AND TRAVIN, A. K. 2008 A hybrid RANS-LES  
1045 approach with delayed-DES and wall-modelled LES capabilities. *Int. J. Heat Fluid Fl.* 29:  
1046 1638–1649.
- 1047 SOU, A., HOSOKAWA, S. AND TOMIYAMA, A. 2007 Effects of cavitation in a nozzle on liquid jet  
1048 atomization. *Int. J. Heat and Mass* 50: 3575–3582.
- 1049 STAMPOULI, M. & PAPPAS, M. 2014 CAE Process Workflow Management of an Automotive  
1050 Simulation Scenario. *SAE Technical Paper 2014-01-0297*: 11.
- 1051 SUN, T., GANESH, H. AND CECCIO, S. 2015. X-ray densitometry based void fraction flow field  
1052 measurements of cavitating flow in the wake of a circular cylinder. *68th Annual Meeting of the*  
1053 *APS Division of Fluid Dynamics*, Boston, Massachusetts, US.
- 1054 THOMPSON, J. F., SONI, B. K. AND WEATHERILL, N. P. 1998 *Handbook of Grid Generation*, CRC  
1055 Press.
- 1056 TUNSTALL, M. J. & HARVEY, J. K. 1968 On the effect of a sharp bend in a fully developed turbulent  
1057 pipe-flow. *J. Fluid Mech.* 34(3): 595-608.

- 1058 WASHIO, S. 2014 Recent Developments in Cavitation Mechanisms: Cavitation inception in separating  
1059 water flows. *A Guide for Scientists and Engineers*, Elsevier: 133-157.
- 1060 WHITE, F. M. 2011 *Fluid Mechanics* Avenue of the Americas, New York, US, McGraw-Hill  
1061 Education.
- 1062 ZIGAN, L., SCHMITZ, I., WENSING, M. AND LEIPERTZ, A. 2012 Reynolds number effects on  
1063 atomization and cyclic spray fluctuations under gasoline direct injection conditions. *Fuel*  
1064 *Systems for IC Engines*, Woodhead Publishing: 253-263.
- 1065 ŽNIDARČIČ, A., METTIN, R. AND DULAR, M. 2014 Modeling Cavitation in a Rapidly Changing  
1066 Pressure Field – Application to a Small Ultrasonic Horn. *Ultrason. Sonochem.* 22.
- 1067 ZWART, P. J., GERBER, A. G. AND BELAMRI, T. 2004. A two-phase flow model for predicting  
1068 cavitation dynamics *5th International Conference on Multiphase Flow* Yokohama, Japan.
- 1069
- 1070