Formation of Numerous Velocity Contours Under the Effect of Cavitation
Majid Dashtibadfarid, Hamed Baniameri

To cite this version:

HAL Id: hal-03615875
https://hal.science/hal-03615875
Submitted on 21 Mar 2022

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L’archive ouverte pluridisciplinaire HAL, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d’enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.
Formation of Numerous Velocity Contours Under the Effect of Cavitation

Majid Dashtibadfarid¹, Hamed Baniameri²
Qazvin Azad University¹, Shahroud Azad University²
United States of America¹, ²

Abstract:
In the present simulation, flow inside wink nozzle has been investigated thoroughly. At the beginning a very comprehensive review about the history of the cavitation phenomenon is provided inside the introduction section. Afterwards, contour of velocity magnitude, axial velocity, tangential velocity, and vorticity magnitude are shown separately. It was found that near the inlet of the orifice a sudden decrease in the amount of tangential velocity shows formation of the cavitation phenomenon. Moreover, it was found that in the region where cavitation tends to start vorticity magnitude increases intensely.

Keywords: Cavitation, Multi-Phase flow, Momentum, Continuity, CFD

I. INTRODUCTION
The study of the flow behavior of fuel injector nozzles has grown in popularity as people become more aware of efficient fuel combustion and automotive pollution reduction. It is widely known that increasing the performance of current direct injection engines, such as strengthening spray breakup and creating smaller droplets inside the combustion chamber, may lower exhaust emissions [1-4]. The fundamental goal of performance enhancement is to increase the available liquid surface area to aid mass, momentum, and heat transmission [5, 6]. This aim may be achieved by controlling the flow field within the nozzle orifice, which has a dominant impact on the behavior of the spray at the exit as well as the interaction between the fuel spray and the surrounding air, affecting both the mixing and the spray behavior [7]. Regulating the internal flow field becomes vital to forecast and regulate spray patterns produced by fuel injectors since the properties of flow inside the injector have a significant impact on the ensuing spray and the subsequent combustion process [8, 9]. Injection nozzles create a fuel spray of tiny droplets at fast speeds by applying large injection pressures. In diesel engines, fuel injectors typically include cylindrical plungers that oscillate up and down to act as small pumps, supplying pressurized fuel to the injector nozzle and combustion chamber [10, 11]. In a normal combustion process, the injection pressure in diesel engines may be as high as 1000 to 2000 bar, whereas modern direct injection techniques in Otto engines can be as low as 100 bar. This high level of pressure guarantees atomization [12]. Cavitation in fuel injector nozzles is a very frequent but extremely beneficial phenomena since it may be readily controlled by increasing or reducing the injection or exit pressures. The following is a description of how cavitation is created. Sharp edges on the intake of the nozzle hole will alter the internal flow [13, 14]. The effective cross section for flow passage narrows as the streamlines compress, and the flow velocity rises. According to the Bernoulli equation, the loss of pressure head causes an increase in kinetic energy. The local static pressure drops to a level below the liquid's vapor pressure at some places [15, 16]. When the local liquid pressure in a fuel injector nozzle is lower than the saturation pressure at a particular temperature, cavitation occurs when the vapor phase of a liquid increases and collapses. The cavitation phenomena cause bubbles to develop inside the nozzle hole. The created bubbles will rupture as they exit the nozzle and are released into the surrounding air, speeding up the spray breakup process. A violent breakdown creates finer droplets and speeds up the evaporation of the fuel. As a result, cavitation may both improve liquid atomization and minimize drag, which is especially beneficial for fuel injector nozzles and spray cooling. In addition to these positive impacts, cavitation has several drawbacks that must be considered. Cavitation is regarded harmful to a hydraulic system because it generates erosion, noise, and vibration, which causes positive displacement and even failure of turbomachines. To maximize the favorable effects of cavitation while limiting the negative effects, a thorough understanding of cavitation is required [17]. Cavitation has a dominant influence on produced sprays when combined with turbulence [18]. The findings of published research on cavitation inside injector nozzles are reviewed in this publication. The physics of cavitation will be given first, followed by a detailed discussion of experimental and computational examinations of cavitating flow. The creation of cavitation occurs when the local tension surpasses the liquid's tensile strength, allowing bubbles to form inside the liquid [19]. When the pressure in the liquid flow surrounding the bubbles exceeds that of the bubbles, however, cavitation begins to collapse as the vapor phase of the liquid departs. Despite the fact that the heat of vaporization of the liquid is not negligible, the cavitating flow is often treated as an isothermal flow by omitting the consumed heat of the local vaporization of the liquid [20]. The phase change generated by boiling is not the same as this isothermal situation. Pressure decreases, which can be generated by acoustic or hydraulic sources, is required for cavitation development. The pressure differential created by the pressure wave propagating through the liquid flow causes acoustic cavitation [21, 22]. Hydraulic power systems, such as high-pressure diesel engines, transmissions, and traction control systems, are prone to cavitation. When the lowered pressure is caused by the hydrodynamic motion of fluid flow, which is intimately tied to the geometric configuration of the flow route, hydraulic cavitation occurs [23]. The boundary layer
separation from the wall happens at the orifice inlet as a result of the sudden shift in both the flow passage and direction of high-velocity internal flow [24]. This separation causes a flow contraction, lowering the static pressure below the continuous fluid's vapor pressure. The decline in local pressure causes the liquid to vaporize, changing its phase. The attached cavitation and the vortex (traveling) cavitation are the two forms of cavitation that may be characterized based on their spatial interaction with the nozzle wall. The vapor zone of the connected one is immediately near to the solid border of the injector nozzle, whereas the bubble-filled region of the vortex/traveling cavitation is totally surrounded by the surrounding liquid [25]. To present, the majority of efforts have been focused on the research of the attached cavitation. Two-phase, unstable processes are involved in the production and collapse of bubbles in the wake zone. The increase in downstream freestream pressure causes large-scale bubble collapse, signifying the end of the wake zone and the cavitation event. In the last two decades, significant efforts have been made to build models that can forecast the emergence of cavitation erosion in fuel injection equipment. The phenomenon's complexity, both in terms of geometrical characteristics and operation circumstances, makes prediction difficult. Experiments on simplified geometries are thus critical for understanding the underlying physical processes as well as providing validation data for computational models [26]. The vast majority of numerical models in the literature have been validated using measurements taken in bigger injectors or simplified real-size nozzles operating at lower pressures. Numerical models based on multiphase computational fluid dynamics (CFD) can forecast the phase-change process and hydrodynamic processes that occur in cavitating flows, as well as offer relevant data on cavitation erosion. The dynamics of collapsing vapor cavities near to a solid surface were observed experimentally and used to construct a model. The study of steady and unsteady liquid jet flows, as well as their breakdown, is still continuing. Jets have a wide variety of applications in the investigation of many fundamental phenomena and physical processes. They may be found on length ranges ranging from the atomic to the cosmic [27]. They may be multi-phase in many applications, including several phase transitions, chemical reactions, and complicated flow phenomena. Government worries about human impacts on the environment, particularly global warming, are driving rigorous emission rules for engine makers [28]. The atomization of the injected liquid jet, which plays a vital role in the combustion process and, as a result, controls the formation of pollutants, is primarily responsible for the quality of air–fuel mixing. Engine makers are continually working to improve the fuel injection process in order to minimize exhaust gas emissions [29]. The optimization of air–fuel mixture is difficult due to the large range of engine operating conditions. On the one hand, there are practical applications such as the manufacture of diesel engine injectors, and on the other hand, there is a desire to understand the genesis of crucial atomization events and their impact on jet breakup processes [30, 31]. Combustion chambers in diesel engines are fueled by high-pressure fuel sprayed as a solid cone spray. This spray experiences a sequence of instabilities (longitudinal and transverse) that cause the liquid bulk to split into structures, which then disintegrate into droplets. The main breakdown process happens around the injection point and is the first step in the atomization process [1, 32-34]. In the scattered flow area, primary breakup mechanisms start the atomization process, govern the size of the liquid core, and set the stage for secondary breakup. Despite the fact that atomization is widely used and has a significant impact on engine combustion processes, heat release rate, and exhaust emissions, the characteristics of the spray produced (for example, droplet size and velocity distributions) are still difficult to predict due to the small length and time scales involved, particularly inside the jet [35]. When a high-speed liquid jet is injected into a pressured dense gas, there has been less agreement on the major processes of early breakup. Many interdependent events can cause extreme velocity changes, causing the flow inside the nozzle to become nonlinearly unstable [25, 36]. Turbulence caused by the nozzle geometry and the collapse of cavitating bubbles are examples of these occurrences. The initial breakdown is aided by turbulence, injection velocity changes, and drop shedding, in addition to turbulence. Separating and examining these diverse impacts experimentally is quite challenging. It is vital to understand the spray processes and then analyze the impacts of different parameters and engine operating conditions on fuel flow structures in order to create diesel engines with optimal fuel efficiency and minimal pollutant emissions. This is a difficult topic to research, both experimentally and statistically [37].

II. FINDINGS

Figure 1 shows velocity magnitude distribution alongside the whole nozzle geometry. In this study, we defined two semi-separate regions in order to avoid steep changes in boundary conditions in the inlet and outlet of the orifice. All of the presented cases defined in this study are simulated within 100 bar at the inlet and 40 bar at the outlet. As can be seen in figure 1 velocity changes color from blue region at the beginning to green, yellow and then red regions, which is signifying an increase in the amount of velocity when the flow enters the orifice area. Inside the orifice area, at the very beginning near the curved inlet augmented amount of velocity can be determined around the radial side of the orifice which is showing formation of cavitation. Along the orifice, the highest amount of velocity can be seen at the middle of it and the lowest amount of velocity is visualized near the wall regions which can be due to dissipation and cavitation and the mentioned conclusion is drawn from the manuscript of Sabetpour et al. [38] and Nezamirad et al [39].

Figure 1: Velocity magnitude contour in a Wink nozzle in cavitating flow.

Figure 2 shows formation of the radial velocity along the nozzle. It can be seen that at the left reservoir the velocity has symmetric behavior almost in the whole region. At the beginning of the orifice, velocity experience a very steep change and increase which is showing formation of cavitation. In the outlet of the orifice, a type of a spray behavior can be seen which is very efficient and is because of formation of cavitation inside the orifice.
Figure 2: Radial velocity contour in a Wink nozzle in cavitating flow.

Figure 3 shows formation of tangential velocity along the nozzle geometry. In the left reservoir, the characteristics of the flow is much more different than the right reservoir. Inside the left reservoir near the inlet of the orifice there are two regions that has the lowest values of the tangential velocity and those regions are the regions that cavitation has a high potential of occurrence.

Figure 3: Tangential velocity contour in a Wink nozzle in cavitating flow.

Figure 4 shows formation of vorticity magnitude inside the nozzle geometry. It can be seen that the inlet of the orifice vorticity increases noticeably which shows that the mentioned regions are susceptible toward formation of the cavitation phenomenon.

Figure 4: Vorticity magnitude contour in a Wink nozzle in cavitating flow.

III. CONCLUSIONS

In this study, we presented formation of the cavitation phenomenon through several contours which are velocity magnitude, axial velocity, tangential velocity, and vorticity magnitude. It was found that vorticity increases and tangential velocity decreases in the regions that have tendency of formation of the cavitation phenomenon.

IV. REFERENCES


33. Birkin, P.R., et al., Cavitation clusters in lipid systems - Ring-up, bubble population, and bifurcated streamer lifetime. Ultrasonics Sonochemistry, 2020. 69.


