

Large Eddy Simulation of Cavitating Multiphase Flows for Liquid Injection

Mathieu Caillou, Jérôme Hélie, Julien Reveillon, F.X. Demoulin

▶ **To cite this version:**

Mathieu Caillou, Jérôme Hélie, Julien Reveillon, F.X. Demoulin. Large Eddy Simulation of Cavitating Multiphase Flows for Liquid Injection. 9th International Symposium on Cavitation, EPFL, Dec 2015, Lausanne, Switzerland. pp.012081, 10.1088/1742-6596/656/1/012081 . hal-02144951

HAL Id: hal-02144951

https://normandie-univ.hal.science/hal-02144951

Submitted on 17 Jul 2019

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.

Journal of Physics: Conference Series

PAPER • **OPEN ACCESS**

Large Eddy Simulation of a Cavitating Multiphase Flow for Liquid Injection

To cite this article: M Cailloux *et al* 2015 *J. Phys.: Conf. Ser.* **656** 012081

View the article online for updates and enhancements.

Related content

- Large eddy simulation of turbulent cavitating flows
A Gnanaskandan and K Mahesh
- Quantifying variability of Large Eddy Simulations of very large wind farms
S J Andersen, B Witha, S-P Breton et al.
- Large Eddy Simulations on Vertical Axis Hydrokinetic Turbines and flow phenomena analysis
N Guillaud, G Balarac, E Goncalvès et al.

Recent citations

- An improved method for coupling the in-nozzle cavitation with Multi-fluid-quasi-VOF model for diesel spray
Hong Liu *et al*

IOP ebooks™

Bringing you innovative digital publishing with leading voices to create your essential collection of books in STEM research.

Start exploring the collection - download the first chapter of every title for free.

This content was downloaded from IP address 194.199.107.36 on 17/07/2019 at 15:18

Large Eddy Simulation of a Cavitating Multiphase Flow for Liquid Injection

M Cailloux[1], J Helie[2], J Reveillon[1] and F X Demoulin[1]

[1]CNRS UMR 6614 CORIA, Rouen, France
[2]Continental Automotive SAS, Toulouse, France

E-mail : demoulin@coria.fr

Abstract. This paper presents a numerical method for modelling a compressible multiphase flow that involves phase transition between liquid and vapour in the context of gasoline injection. A discontinuous compressible two fluid mixture based on the Volume of Fluid (VOF) implementation is employed to represent the phases of liquid, vapour and air. The mass transfer between phases is modelled by standard models such as Kunz or Schnerr-Sauer but including the presence of air in the gas phase. Turbulence is modelled using a Large Eddy Simulation (LES) approach to catch instationnarities and coherent structures. Eventually the modelling approach matches favourably experimental data concerning the effect of cavitation on atomisation process.

1. Introduction

In internal combustion engines, injectors are used to deliver the fuel to the engine in the most efficient way in order to trigger the combustion. This mixing between air and gasoline is improved by the formation of a spray with fine droplets at very high injection pressure. The injected liquid flows through the nozzle at very high speeds, usually in the order of hundreds meters per second, it is compressible and highly transient. In addition, the strong acceleration during the passage of the fluid throughout the nozzle, generate a pressure drop that can be less than the fuel vaporization pressure, triggering cavitation. Cavitation can help in promoting the primary atomization, however it also forces the liquid inside the injector nozzle to be distributed unevenly and results in unpredictable behavior of the spray shape. Therefore in order to obtain a precise control of the injection and for the improvement of engine performance, it is crucial to study the formation and development of cavitation.

OpenFOAM® [1] libraries used in this work include originally solvers for resolving a compressible multiphase flow as well as models for cavitation such as Kunz [2] and Schnerr-Sauer [3] for instance. However, both models hold generally for a flow composed only by the couple liquid/vapour, which does not represents the reality of fuel injection, where the ambient gas is not vapour but for instance air. The aim of this work is to complete the standard OpenFOAM® libraries for simulating multiphase flows with an appropriate physical cavitation model to represent the engine injectors.

The structure of the paper is as follow: first of all the modelling approaches are described together with their implementations. Then the models are validated against Sou's experimental data [4] for a liquid-vapour flow. The final part is dedicated to the extension to a gas mixture that can include air in addition to the vapour.

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd

2. Mathematical and physical model

2.1. Transport equations of volume fraction

The present method includes a transport equation for the volume fractions α_i ("i" stands for vapour, liquid or air), defined as the ratio between fluid i volume to the total volume. α_i ranges between 0 and 1, where 0 means that there is no fluid i in this cell and 1 corresponds to a cell full of this fluid.

Then we have the following equations for the volume fractions:

$$\begin{aligned} \partial_t(\alpha_l\rho_l) + \nabla.(\alpha_l\rho_l U) &= \boldsymbol{\dot{m}} \\ \partial_t(\alpha_v\rho_v) + \nabla.(\alpha_v\rho_v U) &= -\boldsymbol{\dot{m}} \\ \partial_t(\alpha_{nc}\rho_{nc}) + \nabla.(\alpha_{nc}\rho_{nc} U) &= 0 \end{aligned} \tag{1}$$

The source term added in bold for liquid and vapour equations traduces the exchanges between this two fluids induced by the cavitation. This term has been added to the compressible multiphase solver "compressibleMultiphaseInterFoam" of OpenFOAM®. The subscript *nc* is used to described the air phase which is a non cavitating fluid in this flow. The subscripts *l* and *v* stand for liquid and vapour respectively, leading to $\alpha_l + \alpha_v + \alpha_{nc} = 1$.

Bulk densities for liquid, vapour and air follow a linear relation to the pressure to include compressible phenomena:

$$\rho_l = \rho_{l0} + \psi_l p\ ,\ \ \rho_v = \rho_{v0} + \psi_v p,\ \ \rho_{nc} = \rho_{nc0} + \psi_{nc} p \tag{2}$$

ψ_i represents the compressibility coefficient of specie "i". Rewriting the equations (1) by substituting the densities with equations (2) leads to:

$$\begin{aligned} \partial_t(\alpha_l) + \nabla.(\alpha_l U) &= -\frac{\alpha_l\psi_l}{\rho_l} D_t p\ + \frac{\dot{m}}{\rho_l} \\ \partial_t(\alpha_v) + \nabla.(\alpha_v U) &= -\frac{\alpha_v\psi_v}{\rho_v} D_t p - \frac{\dot{m}}{\rho_v} \\ \partial_t(\alpha_{nc}) + \nabla.(\alpha_{nc} U) &= -\frac{\alpha_{nc}\psi_{nc}}{\rho_{nc}} D_t p \end{aligned} \tag{3}$$

Then, by combining these equations the velocity divergence is obtained:

$$\nabla.U = -\left(\frac{\alpha_l\psi_l}{\rho_l} + \frac{\alpha_v\psi_v}{\rho_v} + \frac{\alpha_{nc}\psi_{nc}}{\rho_{nc}}\right) D_t p + \dot{m}\left(\frac{1}{\rho_l} - \frac{1}{\rho_v}\right) \tag{4}$$

2.2. Cavitation model

In the literature, among existing models for the cavitation we start with approaches proposed by Kunz [2] and then Schnerr-Sauer [3] is considered (not presented in this abstract).

Kunz's approach decomposes the source term, $\boldsymbol{\dot{m}}$**,** into 2 parts. One part describes the liquid to vapour phase change (cavitation), which occurs when the local pressure is less than the saturation pressure: $\dot{m}^-$. On the other hand, when the pressure is higher than the saturation pressure, the vapour to liquid phase change is activated (condensation): $\dot{m}^+$.

$$\dot{m}^- = \frac{C_{\text{dest}}\rho_v\alpha_l \min(0, p-p_{\text{sat}})}{(0.5\rho_l U_\infty^2)t_\infty} \text{ and } \dot{m}^+ = \frac{C_{\text{prod}}\rho_g {\alpha_l}^2(1-\alpha_l)}{t_\infty} \tag{5}$$

C_{prod}, C_{dest}, U_∞ and t_∞ are constants to be specified.

3. Validation against Sou's experiments

3.1. Description of Sou's experiments

Work of Sou [4] aims to show different cavitation regimes, based on an experimental set up. He pointed out 4 main types of cavitation, depending on the cavitation number as well as the Reynolds numbers.

The cavitation number is defined as follow, where P_a is the atmospheric pressure, P_s the saturation vapour pressure and V_N the bulk liquid velocity in the nozzle.

$$\sigma = \frac{P_a - P_v}{0.5\rho_l V_N^2} \quad (6)$$

Sou's experiments point out that for cavitation number higher than 1.2, no cavitation bubbles are observed. Then for $0.75 \leq \sigma \leq 1.2$, cavitation cloud appeared in the upper half of the nozzle. For a cavitation number between 0.55 and 0.75, cavitation zone extended from the inlet to the injector exit. This is the super cavitation regime. In this condition, the liquid jet atomization is improved as droplets and ligaments appear quickly and the spray angle is enhanced. For lower cavitation numbers, we can observe the formation of hydraulic flip, in which air goes downstream at one edge of the nozzle up to the inlet.

3.2. Validation of the approach for a liquid-vapour flows

In the injection simulations presented in this section, it is considered that the liquid is injected to the chamber which is filled only with vapour. The cavitation phenomenon starts at the entrance of the injector nozzle where the composition of the ambient gas does not play any role. The simulations of Sou's experiments, based on the previous work of Lu [5], assuming only vapour give results presented on figure 1.

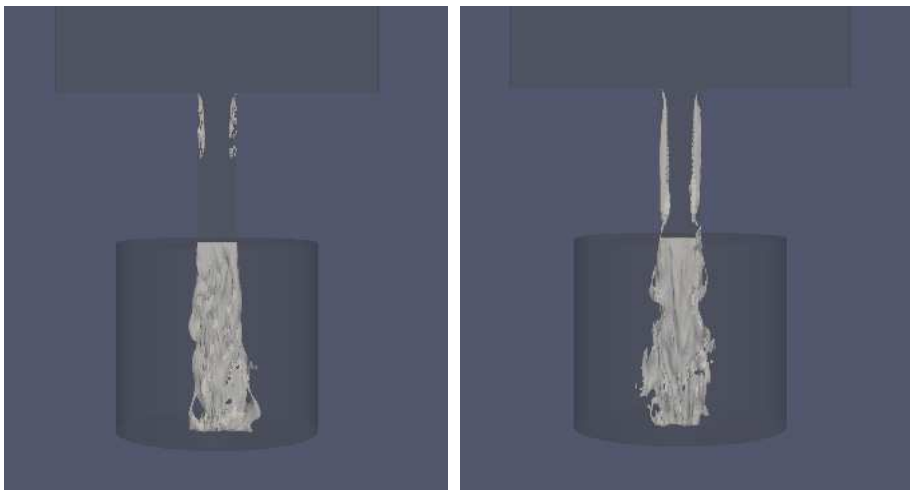


Figure 1. Simulations of the Sou's experiments. Cavitation number of 0.94 (left) and 0.65 (right)

The simulations are able to recover the cavitation regime proposed by Sou et al [4] more validations are proposed by Lu et al [5]. However, the ambient pressure is greater than the saturation pressure thus the model will consider condensation process acting on the spray jet because the simulated ambient gas is vapour instead of the actual air. The introduction of air in the composition of the gas phase has to be considered to avoid this bias.

4. Cavitation in multiphase flows

4.1. Method

Generally cavitation modelling considers one specie with two phases liquid and vapour. Application to atomization requires to consider for the gas phase other species such as air. As a consequence, under the assumption of thermodynamic equilibrium at the surface, this is the partial pressure of vapour that is equal to the saturation pressure instead of the total pressure in the gas. This is included in the mass transfer source term by replacing the total pressure by the vapour partial pressure. The second issue is related to the treatment of the liquid surface if any interface capturing method is considered. Numerical method should be revisited to carefully avoid any diffusion between liquid and gas but allowing for diffusion between different species of the gas phase. Here a splitting method has been used separating the treatment of the convection term from the diffusion term in the context of a VOF approach. More details regarding this aspect will be given during the presentation due to size restriction of this abstract.

4.2. Results and enhancements

The following picture shows results obtained for gas phase composed by vapour and air. A tank full of liquid at high pressure flows out to external environment composed of air. The nozzle geometry corresponds to a Sou's experiment. The Kunz's cavitation model is used in this case, see figure 2.

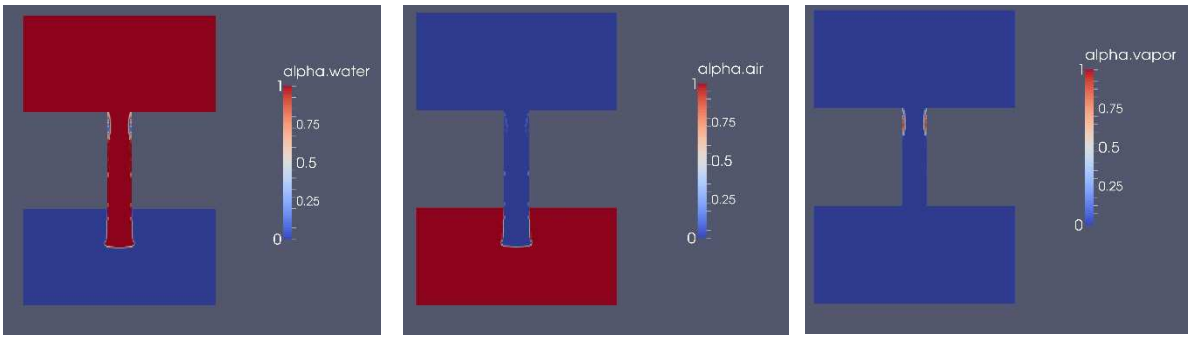


Figure 2. Simulation of cavitating injection: liquid, vapour and air volume fraction fields

We can see that now it is possible to have an injection of liquid into air with a production of vapour in the cavitation area. Additional comparisons between injection in vapour and in air will be presented.

5. Conclusions and perspectives

This work is an attempt to represent cavitation effect on atomisation processes with a solver based on interface capturing method for the two-phase flow, LES for the turbulence modelling and including the possibility various composition of the gas phase. Further improvements are expected to link the cavitation model to the vaporisation model. This should be done in particular by including the effect of the temperature. Additional improvement concerns the representation of subgrid scale wrinkling of the interface. A subgrid method should be applied, as the LES version of ELSA developed by Hecht et al [6] for the atomization.

References

[1] www.openfoam.org
[2] Kunz R, Boger D, Stinebring D , Chyczewski T, Lindau J, Gibeling H, Venkateswaran S and Govindan T 2000 *A preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction* Computers and Fluids 29 (8): 849-875
[3] Sauer J, Winkler G and Schnerr G H 2000 *Cavitation and condensation - common aspects of physical modeling and numerical approach* Chem. Engng & Technol 23: 663-666
[4] Sou A, Hosokawa S and Tomiyama A 2007 *Effects of cavitation in a nozzle on liquid jet atomization* IJHT 50 : 3575-3582
[5] Lu N, Demoulin F X, Reveillon J and Chesnel J 2014 *Large eddy simulation of cavitation and atomization in injector flows using OpenFOAM* (Bremen : ILASS)
[6] Hecht N and Demoulin F X 2014 *Towards general purpose LES model of injection and atomization* (Bremen : ILASS)