Numerical Simulations of the Interactions between a Vortex and a Liquid Gas Interface

J. Cousin*, T. Ménard, A. Berlemont and J. Hélie†,
*CNRS UMR 6614 – CORIA Université et INSA de Rouen
Avenue de l'Université – BP 76801 Saint Etienne du Rouvray, France
† Continental Automotive SAS
1 Avenue Paul Ourliac – BP 1149 Toulouse Cedex 1, France

Abstract

The present study concerns the bidimensional direct numerical simulations between a vortex pair and a liquid-gas interface. The vortex pair is seen here as an energetic structure which is aimed to break the gas-liquid interface. By varying the vortex characteristics (size, energy,...), a series of simulations led to an energetic analysis which helps to know whether liquid flow structures such turbulent eddies are strong enough to produce small droplets.

Introduction

In the case of pressure atomizers, internal liquid flows and sprays are produced thanks to hydraulic energy brought by mechanical systems such pumps. It is well known that these systems are rather inefficient to produce small droplets. As a matter of fact, with these atomizing systems, the initial hydraulic energy is mainly converted into spray kinetic energy and the relative amount of spray surface energy is weak. For instance in the case of atomizers devoted to automotive port fuel injection, Triballier *et al* [1] defined an atomization efficiency, as the ratio between the surface energy and the brought mechanical energy, and measured efficiencies less than one per cent.

The objectives of this study are to get a better understanding of the propensity of energetic structures to deform a liquid-gas interface and to lead to its breakup in well defined conditions. This work consists in simulating the interactions between a liquid-gas interface with a vortex pair in a 2D configuration without any gravitational effects. This vortex pair is considered as an energetic structure which is aimed to break the gas-liquid interface. By varying the vortex characteristics (size, energy,...), a series of simulations led to an energetic analysis which helps to know whether liquid flow structures such turbulent eddies are strong enough to produce small droplets during the primary jet breakup process.

Numerical Approach

The academic DNS "ARCHER" program used here was developed to compute the Navier-Stokes equation for incompressible two-phase flows with arbitrary free surface, in 2 and 3 dimensions. Interface tracking is performed with a Level-Set method [2], the Ghost Fluid method [3] is used to capture accurately sharp discontinuities and both Level-Set and Volume Of Fluid methods are coupled [4] to ensure mass conservation. Detailed information about the implementation schemes and methods can be found in [5,6].

In this work, we numerically controlled the vortex pair by using Oseen's vortices. The mathematical form of such vortices expresses the tangential velocity as a function of the distance from the vortex center (r):

$$u_\theta = 1.56697 u_{\theta max}\left(\frac{l}{r}\right)\left[1-exp\left(-\frac{r^2}{l^2}\right)\right] \quad (1)$$

where $u_{\theta max}$ is the maximum velocity encountered inside the vortex and l is a vortex characteristic length. The constant in the equation (1) is derived from the original expression of the Oseen's vortex where the velocity profile is expressed as a function of the circulation.

The computational domain is meshed with a regular cartesian grid and is constituted of two separated fluids. When the computation starts, the liquid (μ_L and ρ_L are the dynamic viscosity and density respectively) fills the half right of the volume and the left part is filled of gas (μ_G and ρ_G are the dynamic viscosity and density respectively). At the initial computational time, the gas-liquid interface which is characterized by the surface tension coefficient σ is unperturbed and the vortex pair is injected within the liquid at a rather large distance from the interface.

* Corresponding author: cousin@coria.fr

In the present investigation, the physical phenomenon depends on six numbers without dimensions: two are length ratios, the liquid-gas density and the viscosity ratios as well as Reynolds and Weber numbers defined as:

$$Re = \frac{\rho_L u_{\theta max} l}{\mu_L} \text{ and } We = \frac{\rho_L u^2_{\theta max} l}{\sigma} \quad (2)$$

Results and Discussion

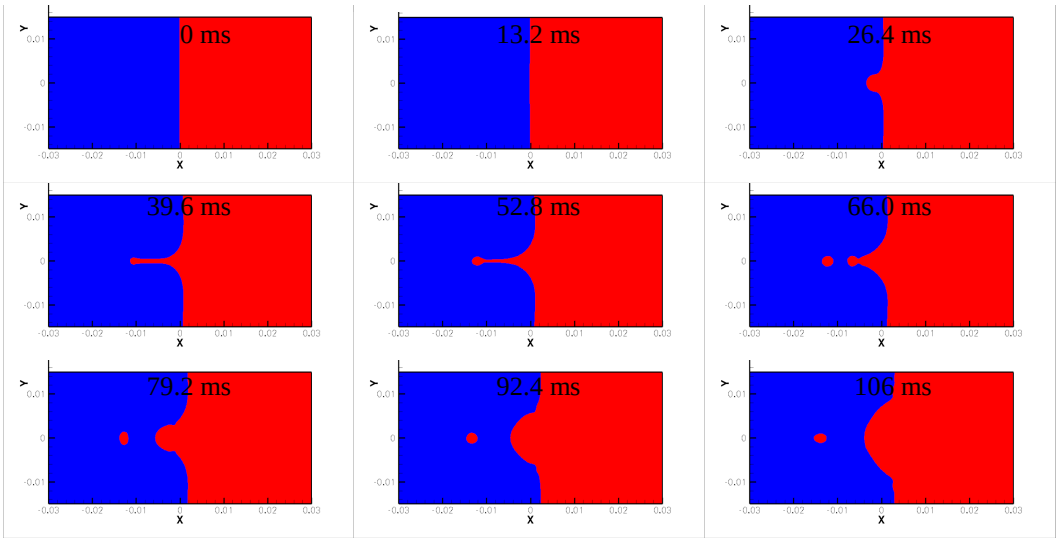

Figure 1. Temporal evolution of the liquid gas interface (water in air, *Re* = 880 and *We* = 13.89)

Figure 1 illustrates the temporal evolution of the water-air interface subjected to a vortex pair. It this case, the calculations report a breakup of the interface meaning that the vortex pair is strong enough to produce a droplet. By doing a series of numerical experiments, where Reynolds and Weber numbers only were varied, we showed that for a fixed Weber number, a critical Reynolds number has to be reached in order to observe a liquid detachment. All the numerical results were synthesized in a Reynolds-Weber numbers diagram where a critical line which locates the liquid detachment has been found as observed in Fig. 2. Finally, the study is extended to situations where density and viscosity ratios are varied.

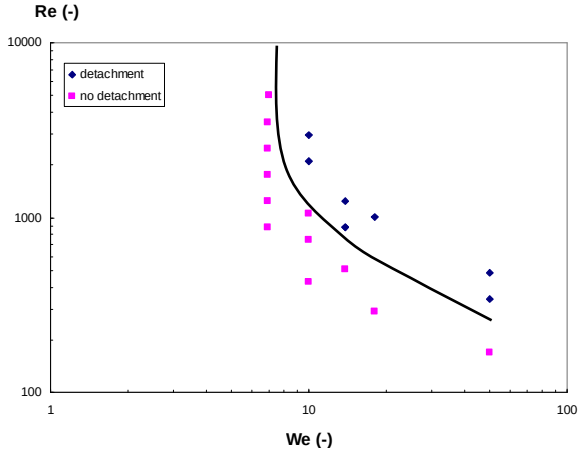

Figure 2. Critical line in a *Re*-*We* diagram (each point corresponds to different values of the couple ($u_{\theta max}$, σ))

References

[1] Triballier, K., Cousin, J., Dumouchel, C. ILASS Europe, Zaragoza, Spain, September 2002.
[2] Stanley Osher and James A. Sethian, *Journal of Computational Physics* 79 (1), 12 (1988)
[3] Fedkiw R. , Aslam T. , Merriman B. , Osher S . *J. Comp. Phys*. 152:457-492 (1999)
[4] Mark Sussman and Elbridge Gerry Puckett, *Journal of Computational Physics* **162** (2), 301 (2000)
[5] Menard, T., Tanguy, S. and Berlemont A., R.D., *International Journal of Multiphase Flow* 33 (5):510-524 (2007).
[6] Tanguy, S., Berlemont A. *International Journal of Multiphase Flow* 31:1015-1035 (2005).