# CFD SIMULATION OF PRIMARY ATOMIZATION FOR ENGINE APPLICATIONS.
# 1ST PAN AMERICAN CONGRESS ON COMPUTATIONAL MECHANICS - PANACM 2015 IN CONJUNCTION WITH THE XI ARGENTINE CONGRESS ON COMPUTATIONAL MECHANICS - MECOM 2015

## NORBERTO NIGRO[1], JUAN GIMENEZ[1], SANTIAGO MARQUEZ[1], HORACIO AGUERRE[1] AND DAMIAN RAMAJO[1]

[1]Centro de Investigación en Métodos Computacionales (CIMEC)
Colectora Ruta Nac 168 / Paraje El Pozo (3000) Santa Fe
nnigro@intec.unl.edu.ar and http://www.cimec.santafe-conicet.gov.ar

**Key words:** Computational Fluid Dynamics, Multiphase Flow, VOF, Atomization

**Abstract.**

Liquid atomization is an important process which found interest in several engineering applications such as aerospace propulsion systems, automotive engines, food processing, aerosols and ink-jet printing. Its numerical simulation allows to investigate physical processes of the atomization because our understanding on physical mechanisms of such phenomena is still not sufficient.

Our investigation group is doing its first steps in this research area and we report in this work our early results using two different approaches. On one hand a typical Eulerian collocated finite volume and on the other hand a Lagrangian based Particle Finite Element Method (PFEM) using both a volume of fluid (VOF) two phase flow model to capture the atomization phenomenon.

The final target of this research is the improvement of less detailed methodologies to be applicable in the internal combustion engine simulation using the results obtained here with an almost direct numerical simulation.

In order to validate these codes a very simple test has been chosen. This benchmark rests on the work of Menard and co-workers [**?**, **?**], which employ the Level Set Method (LSM) to track the interface added to the Ghost Fluid Method (GFM) to describe the interface discontinuites and manage the pressure, density and viscosity jumps. Also, the Level Set method is coupled with the Volume of Fluid method (VoF) to ensure mass conservation. The mesh used by Menard in [**?**] is a $2048 \times 256 \times 256$ Cartesian grid with regularly spaced nodes ($\Delta = 1.17[\mu m]$). The size of the domain is (2.1[mm], 0.3[mm], 0.3[mm]), where the first dimension is the streamwise direction and the other two, the

spanwise directions. At the injection level, the jet diameter $D$ is equal to 0.1[mm], while the liquid jet Reynolds number is equal to $Re = 4659$. Liquid surface instabilities close to the injector are visible. Their deformation leads to the formation of ligaments and droplets of various sizes. At the end of the domain, the liquid core has almost disappeared and a dense spray of droplets leaves the computational domain. The key of the quickly drop production is the use of a space-time correlated turbulent flow at the inlet: Menard uses a syntetized correlated turbulence with a method proposed by Klein [**?**]. In the work of Desjardins [**?**], the author uses a forerunner simulation to impose the inlet turbulent boundary condition, obtaining similar results to the above mentioned strategy. Both works have a relevant conclusion: by the end of the computational domain, the liquid core has been fully disintegrated. Another approach in the numerical characterization of jet atomization is reported by Shinjo in [**?**, **?**]. In his work, the author reports that the grid resolution used by Menard was coarse for the chosen Reynolds and Weber numbers, so this was not a direct numerical simulation in a true sense: the produced ligaments and droplets did not exhibit smooth shapes or wave dynamics driven by surface tension, but the overall liquid jet motion was captured in that simulation. Shinjo et.al had obtained their solution using different uniform meshes, with the finest mesh with around 6000 million of cells solving scales up to ($\Delta = 0.35[\mu m]$). The ligament instabilities are achieved far from the inlet: the main responsible is the plain velocity front at the inlet imposed by Shinjo instead of using a turbulent-induced flow [**?**].

Our initial simulations using the algorithms above mentioned (PFEM and `interFOAM`) show some similarities with both results, depending on the inlet condition imposed. It must be taken into account that in the most refined case simulated with OpenFOAM®(at full hardware capacity), the geometry was meshed with an uniform cartesian grid of $128 \times 128 \times 1024$ ($\Delta x \approx 2.3[\mu m]$), while the PFEM simulations had a $\Delta x \approx 7.5[\mu m]$ (7 millons of tetrahedral), far from the refinement degree used in the reference works.

In order to improve these results an adaptive refinement strategy with OpenFOAM®called `interDyMFoam` was firstly used with a base mesh of $16 \times 16 \times 128$ using 4 refinement levels reaching around 13 millons cells and a scale resolution of $\Delta x \approx 1.17[\mu m]$. These auspicious results serve not only for an initial comparison against the references but to understand the physical phenomena involved and their impact on the engine operation. The role of the turbulent velocity profile at the inlet was assessed using the so called `vortex method` implemented in the two codes used in this work. The following step was the addition of a new refinement level (5 instead of 4) reaching a scale resolution around $\Delta x \approx 0.6[\mu m]$. The droplet distribution convergence is tested in order to decide when the mesh is fine enough to accomplish this problem. The results show that the droplet formation and the like-mushroom shape are comparable with Shinjo but the minimum drop size is better described using a finer mesh reachable with an extra refinement level (6 instead of 5), but currently unaffordable with our available computation resources. Finally it may be concluded that both methodologies have the potential to solve this type of problems.