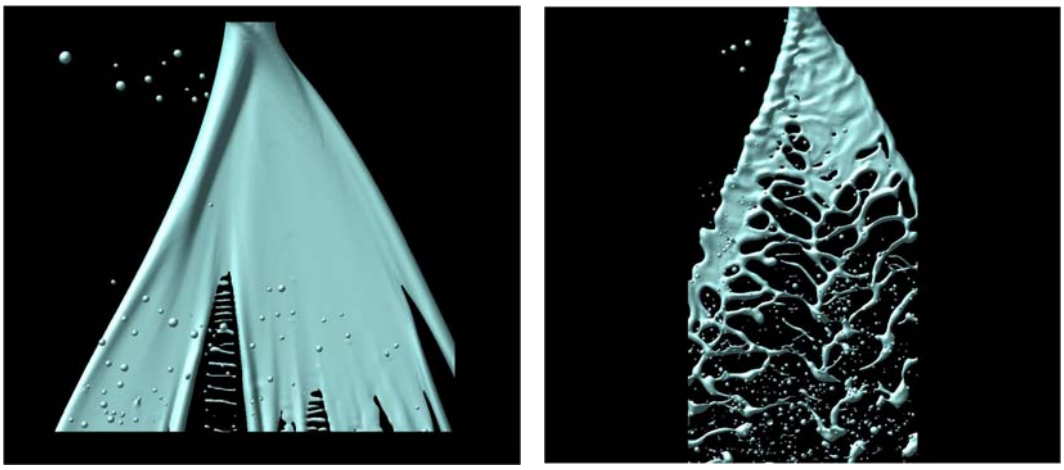
A Coupling between Internal Flow and Primary Breakup Simulations - Case of a Compound Nozzle

A. Berlemont[1], J. Cousin, M. Doring and T. Menard
CNRS UMR 6614 – CORIA Université et INSA de Rouen
Avenue de l'Université – BP 76801 Saint-Etienne du Rouvray, France

Abstract

In port-fuel gasoline engines, the injection pressure is kept low (usually less than 1 MPa) ; this results in the production of slow liquid jets at the nozzle exit. In such situations, as ambient pressures remain close to atmospheric conditions, the aerodynamic forces acting on the atomization process are negligible. Thus, the gasoline spray is mainly controlled by primary atomization where turbulence and capillary forces are of paramount importance. Low-pressure injectors with compound nozzle are commonly used in port-fuel spark-ignited engines to eject gasoline spray. The concept of compound nozzle is to enhance atomization by creating a secondary flow at the nozzle outlet as well as a relative strong turbulence level. This study focuses on the triple disk compound injectors which consist of the superposition of three disks that imposes drastic deflections to the internal flow. First, the secondary flow is characterized by a double vortex which induces a flattening of the cylindrical liquid jet as soon as it leaves the nozzle discharge orifice. Second, the turbulence level imposes initial perturbations on the liquid-gas interfaces. Recent experimental results demonstrated that the size of the droplets strongly depends on the non axial and turbulent kinetic energies of the issuing flow. These kinetic energies were deduced from internal flow simulations that also reported a dependence of the nozzle design.

This paper is intended to demonstrate the capability of a numerical approach to retrieve the involved primary breakup process in the case of a single triple disk compound injector.
The present study is based on a coupling between internal flow modeling and primary breakup simulations. First internal flow simulations are achieved thanks to the commercial code FLUENT 6.2. The computational domain was built with more than one million of cells and turbulence was modeled by using either a RANS approach or a Large Eddy Simulation (LES). This part of the study mainly focuses on the determination of the secondary flow and the turbulence level at the nozzle exit. The calculated issuing flow characteristics through the velocity profiles are then used as initial conditions for DNS simulations of the primary breakup.
A lot of topological changes occur (interface pinching or merging, droplet coalescence or secondary break-up) and the numerical method should describe the interface motion precisely, handle jump conditions at the interface without artificial smoothing, and respect mass conservation. We develop a 3D code, where interface tracking is ensured by Level Set method, Ghost Fluid Method is used to capture accurately sharp discontinuities, and coupling between Level Set and VOF methods is used for mass conservation. Specific care has been devoted to improve computing time with MPI parallelisation.
Methodologies of each simulation part are described and numerical results of the primary breakup are compared to in progress visualizations on a real size nozzle.



Zoom (0<z<8D) on jet break-up –left: without turbulence; right with turbulence

[1] Corresponding author: alain.Berlemont@coria.fr