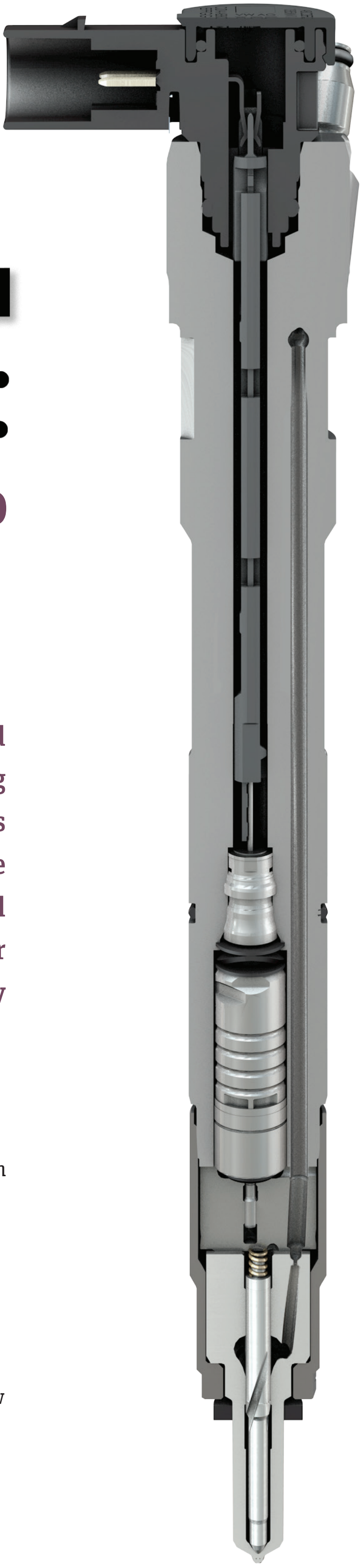
FUEL INJECTION:

Breaking Up is Hard to Do

By **Junmei Shi**, Simulation Team Leader, and
Pablo Lopez Aguado, Ph.D. Student,
Delphi Automotive Systems, Bascharage, Luxembourg

Improving internal combustion engine emissions and fuel economy performance requires better understanding of the process by which the fuel injection nozzle breaks up the liquid fuel and propels atomized droplets into the cylinder. Delphi engineers are using ANSYS computational fluid dynamics (CFD) software to design the fuel injector nozzle geometry to deliver droplets in just the right spray pattern to optimize engine performance.

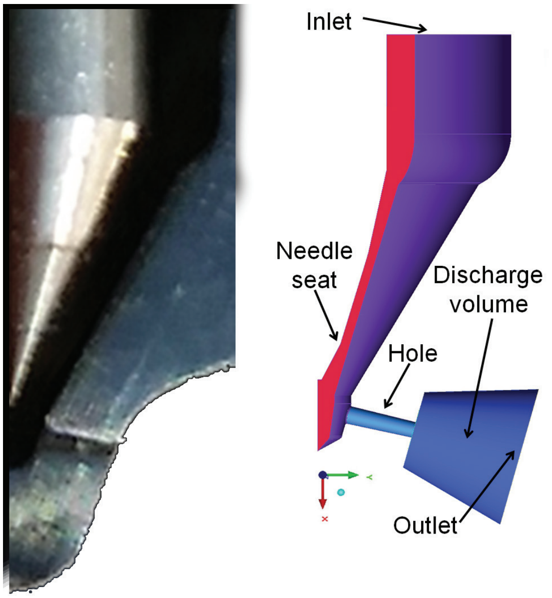


Improvements in clean internal combustion engine technology require controlling and optimizing the fuel–gas mixing, ignition and combustion processes. Engineers must translate the particular spray requirements of each engine into a detailed nozzle design. One of the big challenges to nozzle development is determining the fundamental physics of the primary breakup process and how this is impacted by nozzle geometry. Physical experiments have limitations in understanding the breakup process because there is no way to effectively measure turbulence and vortex structures inside tiny injection nozzles. Delphi Automotive Systems engineers use ANSYS Fluent CFD large eddy simulation (LES) to characterize the nozzle flow dynamics and breakup process. The nozzle flow and measured spray pattern predicted by simulation closely match experimental results, significantly advancing the fundamental understanding of fluid dynamics useful for optimizing fuel injector nozzle designs.

> **“Delphi Automotive Systems engineers use *ANSYS Fluent CFD* to characterize the nozzle flow dynamics and *breakup process.*”**

Traditional Fuel Injector Nozzle Design Methods

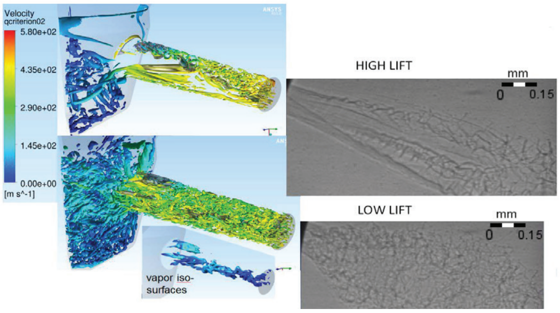
The performance of a fuel injection nozzle in breaking up the liquid fuel into spray droplets and the fuel–air mixing in the engine cylinder have a major impact on fuel economy and emissions. Both primary and secondary breakup phenomena occur simultaneously in the spray formation process. The primary breakup refers to the liquid jet deformation and big ligament formation phenomena. The ligaments further break up into droplets in the secondary breakup process. The primary breakup process involves highly complex multiphase and multiscale fluid dynamics phenomena, including turbulence and cavitation and their interaction inside the nozzle, along with aerodynamic interaction outside of the nozzle. The fuel injection engineering community has been working on this issue for more than 50 years, but have been hampered by lack of effective experimental and numerical diagnostic tools. Optical measurement techniques including phase-contrast X-ray imaging (PCX) and X-ray radiography have been developed for in-nozzle cavitation characterization, but so far there is no effective way to measure field turbulence inside the injection nozzle.



The nozzle flow path

Researchers have also worked with simulation to understand the breakup process. The level set interface tracking technique has been successfully employed with CFD to resolve the liquid–gas interface during droplet formation. But this technique requires direct numerical simulation (DNS) in which the Navier–Stokes equations are numerically solved without any turbulence model to deal with cavitating flows. DNS is still not feasible today since the computer power required is not available. An alternative to the level set technique is the volume of fluid (VOF) technique, which tracks the volume fraction in each cell rather than the interface itself. VOF is effective for in-nozzle flow analysis but is inaccurate for the prediction of jet breakup and droplet formation.

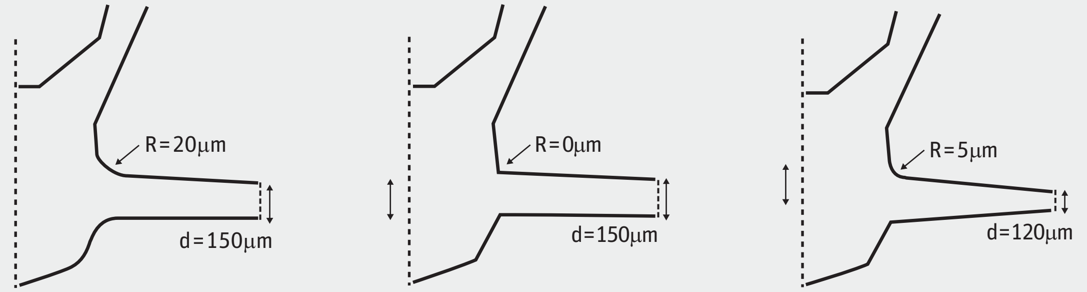
Because of these limitations in measurement and simulation techniques, fuel injector design still largely relies on a parametric optimization of the geometry following the build-and-test method. This process is inefficient and is sensitive to interactions between the many geometric parameters and to inaccuracies in the measurement system.



Strong correlation between simulated and measured nozzle spray for round injection hole nozzles at high and low needle valve lifts provides confidence in internal nozzle flow simulation.

LES Enables CFD Simulation of Flow Inside Nozzle

Delphi is working with Wayne State University and Argonne National Laboratory to achieve detailed characterization of the liquid–gas interface structures of the near-nozzle spray in the breakup process. In parallel, Delphi is leveraging the ANSYS Fluent LES turbulence modeling scheme in conjunction with VOF and



Rounded

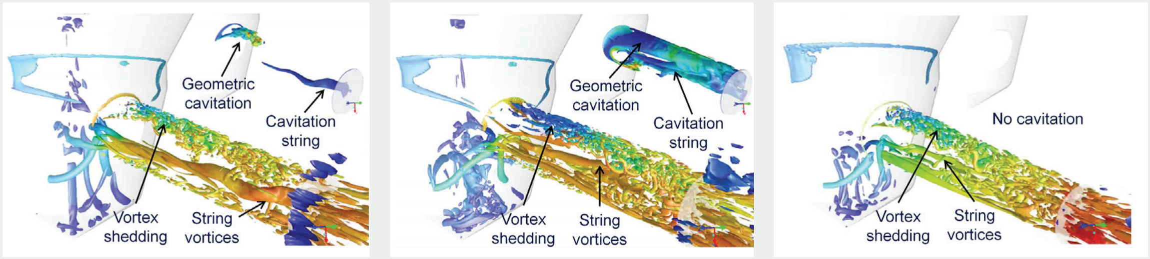
- Near-nozzle spray shows a transitional fluctuation between large-scale regular surface structures and smaller-scale, irregular structures.
- Weaker vortex shedding and cavitation
- String structures survive from time to time all the way to the injection hole outlet.

Sharp Edge

- The jet begins to break up closer to the nozzle hole exit, and the spray pattern is more stable with more fine-scale structures.
- Small-scale vortices accompanied by vortex shedding at the upper lip of the injection hole entrance and strings of vortex pairs rotating in opposite directions accompanied by string cavitation
- The intensity of shed vortices and cavitation are strong enough to break up the string structures inside the injection hole.

High-Performance Atomization

- Moderate, stable spray pattern with small-scale irregular surface structures
- Shed vortices interact with the string vortices, but no cavitation is observed.
- The shed vortices produce pulsating momentum, which leaves the injection hole exit and triggers jet deformation and wavy liquid–gas interface structures.
- Shed vortices also produce pulsating surface vortices, which are enhanced by liquid–gas interfacial interaction between liquid and surrounding gas, triggering droplet formation.



Comparison of LES simulation of the flow inside the nozzle helped engineers to understand how the different nozzle geometries produced contrasting results.

coupled VOF–level set techniques to simultaneously resolve the multiscale vortex dynamics in the nozzle and the liquid–gas interface of the near-nozzle spray during the primary breakup process. In LES, large eddies are resolved directly, while small eddies are modeled. Resolving only the large eddies makes it possible to use a much coarser mesh and larger time steps in LES when compared to DNS. Delphi uses this approach to simulate round- and sharp-edge–hole nozzles as well as its high-performance (HP) atomization hole nozzle, which uses a very high hole taper to increase the nozzle's hydraulic efficiency and the spray momentum rate.

The LES simulation of the flow inside the nozzle helped engineers to understand how the different nozzle geometries produced contrasting results as shown in the chart.

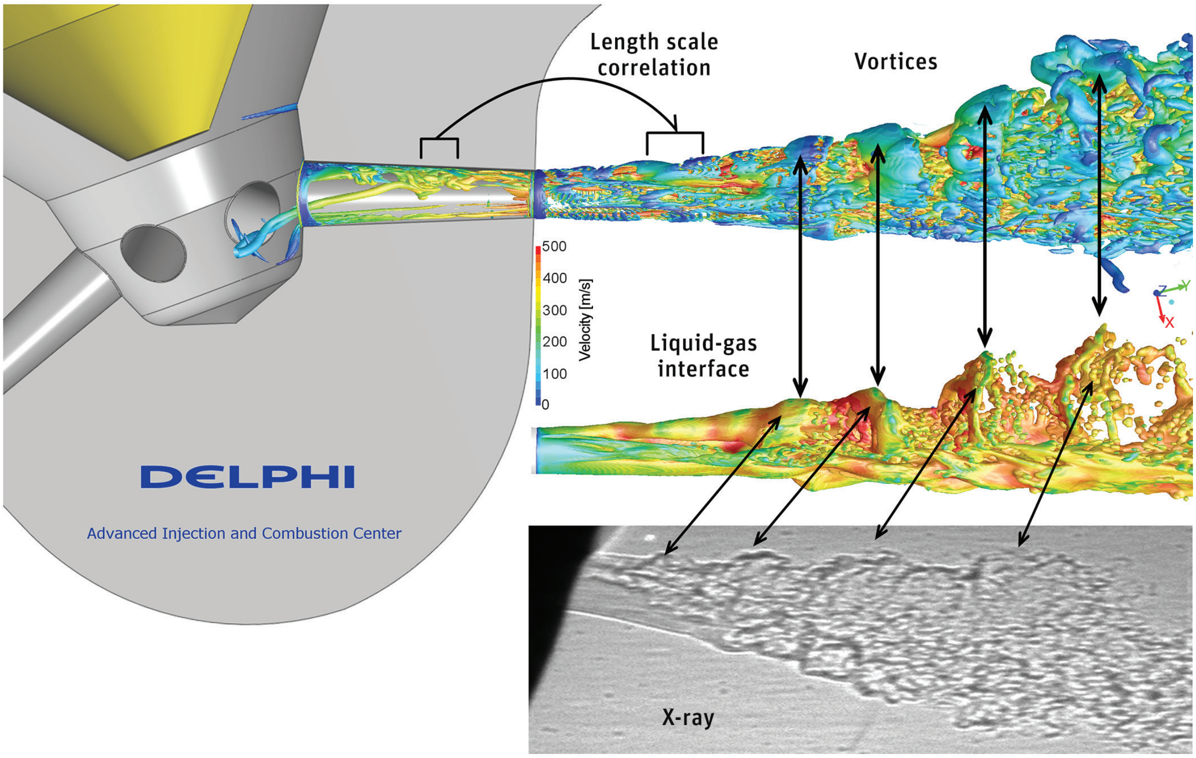
Nozzle Design Migrating to Simulation

For each case, good correlations were found between the predicted and the measured spray patterns, providing a high level of confidence in the accuracy of the flow patterns inside the nozzle. It was discovered that the process of the fluid flow entering the nozzle hole triggers vortex shedding, which further initiates liquid surface deformation and ligament formation in the primary spray breakup. This finding explains the influence of nozzle design parameters such as seat-sac, hole-inlet rounding, taper, needle shape and needle lift on the spray formation, and provides a new understanding of the primary breakup mechanisms in high-pressure fuel injection.

The fuel injection research community has been wrestling for over 50 years with the challenge of understanding the turbulence inside the nozzle and its

> “Delphi Automotive Systems engineers use ***ANSYS CFD*** to characterize the nozzle flow dynamics and breakup process.”

effect on spray simulation. CFD simulation with LES has substantially improved engineers’ understanding of the fundamental physics of the primary breakup process and the impact of the nozzle geometry on spray structure. Delphi engineers are moving forward to incorporate simulation in the design process for fuel injector nozzles in new engine models. Simulation will enable engineers to better understand the complex interaction of geometric parameters within the nozzle, which will allow a shift from a parametric to a knowledge-based optimization process. Fewer samples will be required for testing, and it is expected to reduce the time necessary to develop nozzles, leading to higher-performing engines with greater fuel economy and lower emissions. Λ



Simulated and measured nozzle spray for HP nozzles match well.

References

Shi, J.; Aguado Lopez, P.; Dober, G.; Guerrassi, N.; Bauer, B.; Lai, M.-C. Using LES and X-ray Imaging to Understand the Influence of Injection Hole Geometry on Diesel Spray Formation, Valencia: THIESEL 2016 Conference on Thermo- and Fluid Dynamic Processes in Direct Injection Engines, 2016.

Shi, J.; Aguado Lopez, P.; Guerrassi, N.; Dober, G. Understanding High-pressure Injection Primary Breakup by Using Large Eddy Simulation and X-ray Spray Imaging, *MTZ Worldwide*, **2017**, Issue 5, pp. 50–57, doi 10.1007/s38313-017-0039-4.

Injector Dynamics Assumptions and Their Impact on Performance
ansys.com/injector