Internal fluid dynamics of diesel engine fuel injectors can have a major impact on engine performance. The flow inside the fuel injector influences the pattern with which fuel is sprayed into the engine, which, in turn, can impact combustion performance and emissions. Likewise, internal flow patterns affect fuel injector losses, and reducing these can improve fuel economy and engine performance. Cummins uses computational fluid dynamics (CFD) to simulate internal fluid dynamics of fuel injectors, paying particular attention to cavitation behavior, sac filling and pressure, and spray hole velocity and momentum. The robustness of the CFD software enables Cummins engineers to evaluate many design alternatives and iterate to an optimized fuel injector design with lower losses and a superior spray pattern, resulting in significant improvements in engine performance.

Fuel entering the injector body runs along the needle, passes through the seat, and enters the sac region before exiting the injector through the spray holes. In the computational domain depicted in Figure 1, symmetry assumptions are applied together with a conditional wall boundary condition to completely close the seat area at specific needle lifts.

The complexity of the operation of a fuel injector necessitates a sophisticated injection process flow. For a new design, the first step is a relatively simplistic steady-state analysis focusing on the fuel injector’s nozzle and plunger at specific needle lifts. This simulation is used to determine the force efficiency of the injector, a ratio of the integral pressure distribution along the needle tip and an ideal theoretical value. Losses are typically caused by pressure drop over the needle seat area as well as cavitation in the spray holes and sac. Force efficiency at high lift is typically over 90 percent, while force efficiency at low lift can range from 30 percent to 40 percent. The force efficiency is used as input to a one-dimensional analysis using software developed by Cummins that calculates the lift profile, the needle lift as a function of time (Figure 2). The lift profile, in turn, is used to drive a transient CFD run that provides a more realistic simulation of the complete fuel injection cycle, which, in turn, results in a new force efficiency being plugged back into the one-dimensional analysis.

Cummins uses ANSYS® Fluent® software to optimize fuel injector performance because it has correlated well with experimental results, such as providing excellent prediction of the area of damage to the fuel injector. Another advantage of Fluent is that it provides excellent scalability when run on high-performance computing (HPC) platforms, such as the 144-node cluster that Cummins used for the simulations described here.
Cummins Uses Simulation to Reduce Injector Losses and Improve Spray Pattern for Performance Gains

To reduce the computational intensity, the symmetry assumption is used to include only half a spray hole. The pressures at the inlet and outlet were determined based on experimental setups. The results of using a constant output pressure were compared to results with a simplified variable outlet pressure. A wall was used to close the needle during simulation based on the lift profile (one-dimensional analysis). The flow properties were set based on a kerosene-type calibration fluid to accentuate cavitation behavior. The analysis was run with both two-phase flow – liquid fuel and fuel vapor – and three-phase flow – liquid fuel, fuel vapor and air. The calculation of mass transfer from the liquid fuel to the fuel vapor was based on a cavitation model. In the initial conditions for the three-phase flow, the flow volume was filled with 100 percent air downstream of the sealing diameter.

The simulation calculates flow properties at the outlet, such as pressure, density, velocity, turbulence quantities and volume fractions. These properties can be used to optimize the targeting of the spray to reduce emissions and increase fuel economy.

Simulation highlighted differences between the opening and closing events. Higher velocity gradients resulting in increased vapor formation were observed during the closing event. High-momentum flow along the bottom of the spray hole during the closing event impacted spray targeting. While pressure distribution does not vary substantially between the opening and closing events, a strong hysteresis effect influences vapor formations and the velocity field, resulting in a different flow behavior for opening and closing events (Figure 3).

Cummins engineers used simulation to look at how decreasing pressure at the outlet after the needle valve closes influences the flow of air from the outlet into the sac. At time 1.98333, the pressure drop over the needle seat results in the sac pressure falling below outlet pressure, so an incoming pressure wave travels through the spray hole. The pressure wave reflects off the sac wall and needle at time 2.01667 and continues through a small gap in the needle valve. At time 2.08333, the pressure wave moves toward the bottom of the sac. Pressure variations were seen in the sac decay at time 2.28333. The sac pressure equaled the spray hole pressure at time 2.41667, and constant low pressure reached the outlet and throughout the sac and spray hole at time 2.59167.

The simulation highlighted the benefits of using three-phase flow and variable outlet pressure, despite their higher computational intensity, instead of two-phase flow and constant outlet pressure. During the valve opening process and for full lift, the sac pressure is considerably higher for two-phase flow, resulting in an unrealistically high force efficiency. With constant outlet pressure, more vapor is created in the sac late in the injector cycle when pressure is almost constant. Variable outlet pressure also results in a higher air concentration in the sac volume throughout the simulation, thus decreasing the outlet pressure results in a slightly higher liquid mass flow rate at the spray hole exit after needle closure.
Cummins engineers have used such simulation methods to optimize the design of the fuel injector line. They have essentially eliminated cavitation, resulting in a reduction in warranty returns and maintenance of consistent performance over the life of fuel injectors. Parasitic losses in the injector have been reduced, resulting in greater fuel economy and increased engine performance. Improvements in the spray pattern have helped to reduce emissions and provide further improvements in engine performance.

Figure 4. Conditions in the sac and spray hole late in injector cycle