Fuel Injection Gets New Direction

CFD helps analyze fuel–air mixing in modern gasoline engines.

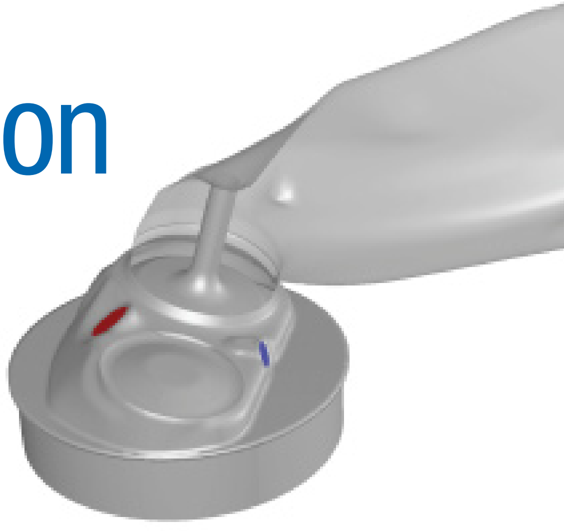
By Chris DeMinco, Delphi Automotive Systems, New York, U.S.A.
Yong Yi, ANSYS, Inc.

Gasoline direct injection (GDI) engine technology has the potential to provide significant improvements in fuel efficiency while maintaining higher power output compared to port injection engines that are currently in mainstream usage. In direct injection, fuel is sprayed directly into the combustion chamber of each cylinder, whereas in multi-port injection, fuel is sprayed into the intake port from a location upstream from the intake valve.

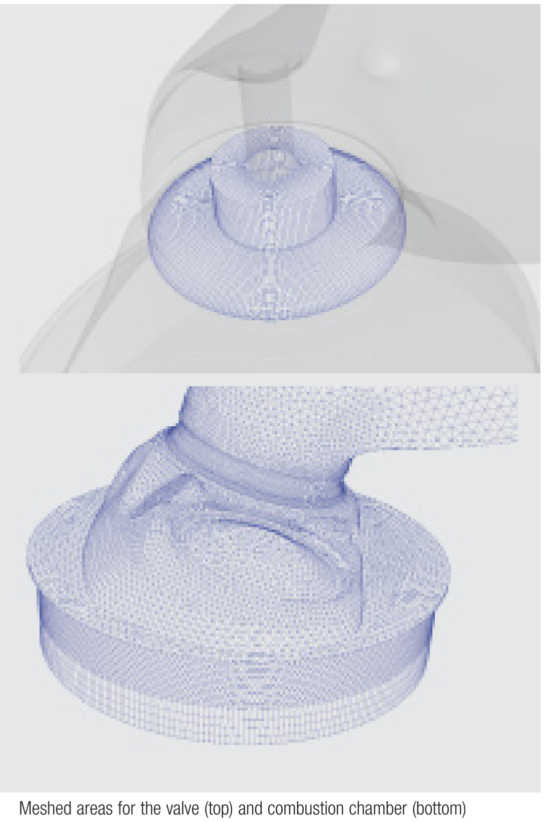
One benefit of using direct injection is that it produces improvements to fuel efficiency by offering more precise control over the fuel delivery process (amount and timing). One of the primary challenges with GDI engine design is to understand the behavior of the fuel injection and mixing processes, since the fuel spray process must be optimized to minimize evaporation time and distributed in a desired manner, under a variety of operating conditions and with a number of injection strategies.

To learn more about injection processes, a study was performed on a two-valve, four-stroke GDI engine provided by Delphi Automotive Systems based in Rochester, New York, in the United States. The study focused on understanding the injected fuel spray interaction with in-cylinder air motion. The effects of injection conditions on the ignition performance were considered by examining the mixture distribution in the combustion chamber and the equivalence ratio, which is the actual fuel-to-air ratio divided by the stoichiometric ratio at the spark plug location.

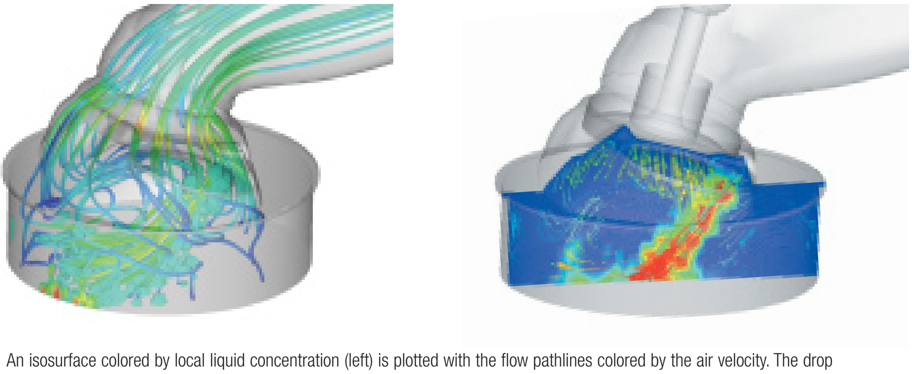
In order to simulate the engine's moving valve and piston accurately, a hybrid mesh was created using both GAMBIT and TGrid software products. The research team divided the mesh into two regions. The first region was a tetrahedral mesh in the upper portion of the combustion chamber near the valves. The second was composed of a hexahedral mesh used in the lower portion of the cylinder above the piston and in the port volumes, including the gap between the valve and valve seat. A layering model was used to simulate the valve motion: when the valve opens, extra cells are added to the grid volume between the valve and the valve seat, and when the valve closes, hexahedral elements are removed from the volumes between the valve and its seat. The team used a similar strategy to model piston motion.



The geometry of the engine: the injector is located near the valves at the end of the engine cylinder as represented by the blue dot, while the spark plug is on the opposite side of the injector, located 3 mm inside the red dot.



Meshed areas for the valve (top) and combustion chamber (bottom)



An isosurface colored by local liquid concentration (left) is plotted with the flow pathlines colored by the air velocity. The drop size distribution (right) is plotted with velocity vectors of air on a plane cutting through the centers of the spark plug and the injector. Both plots show that the spray strongly changes the direction of the air flow.

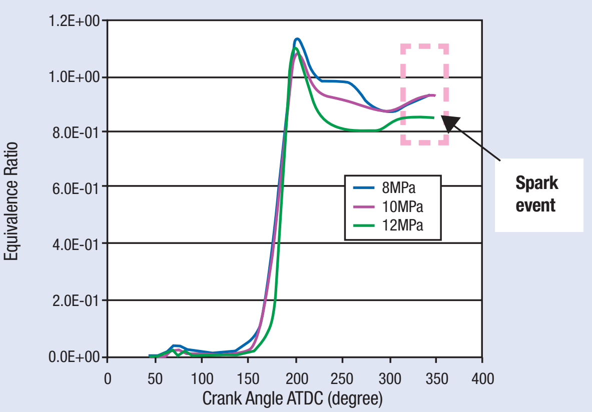
Once the meshing was complete, the motion of the intake valve and the piston was simulated using the dynamic mesh model in the FLUENT computational fluid dynamics (CFD) software package. The injector under analysis has six holes distributed in a U-shape and is located in the engine cylinder opposite the spark plug. However, the detailed structures of the injector and the spark plug were not modeled. Instead, the injector spray was simulated using the discrete phase model (DPM) and spray sub-models within the FLUENT package.

A uniform initial drop size distribution was computed from the measured discharge coefficient at the nozzle exit. The injection velocity was computed from the injection pressure and the discharge coefficient. The atomization and evaporation of the fuel jet was simulated using the variety of spray sub-models in FLUENT, including the drag, breakup, collision and coalescence, wall-impingement and evaporation models.

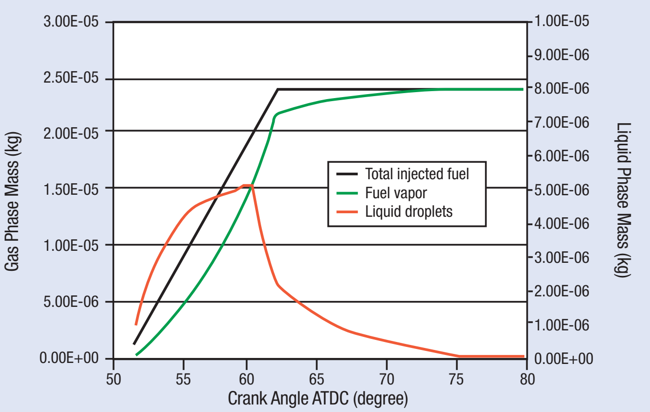
Since the fuel injection timing overlaps with part of the time during which the intake valve is open, the injected fuel will interact with the intake air flow. The FLUENT simulation showed that the fuel jet acts like a curtain and prevents a portion of the intake air from getting into the region of the cylinder below the spray. It also creates significant recirculation zones above and below the fuel spray. These flow patterns significantly impact the liquid evaporation and fuel–air mixing processes. As the liquid fuel spray evaporates, the fuel vapor mixes with air. The quality of the mixture, or the mass fraction that vaporizes, can be evaluated by examining the distribution of the equivalence ratio of the fuel–air mixture in the combustion chamber, especially near the spark location.

From the droplet distribution, the research team noted that many liquid droplets impinge on the piston and cylinder walls. This indicates the high possibility that a film of fuel forms on those surfaces. Due to the heat transfer between the wall film, the wall and its ambient gas, the wall film eventually disappeared. However, because of the weaker heat transfer behaviors and smaller liquid surface area for evaporation, it took much longer for the wall film to evaporate compared to the droplets in the free stream. Thus, the wall film has been shown to have a strong impact on the overall evaporation and mixing processes.

The spray wall impingement is an important factor in resolving accuracy in predictions of mixture quality. CFD models have provided a helpful tool for studying this phenomenon. The models also are proving to be an important asset in the advancement of GDI technology. ■



Equivalence ratio at spark plug location for injection pressure study. The dashed box indicates the possible spark timing. It is seen that all three injections form slightly lean, but still ignitable, mixture around the spark. The mixtures are moving to their stoichiometric ratios at about 180 degrees of crankshaft rotation, which indicates that later spark timing may be more favorable for a stable ignition.



Evaporation history of the fuel–air mixture. The mass of the wall film increases very rapidly at the start of injection since the piston is very close to the injector at that point. The wall film mass reaches the maximum value at a crank angle of 60 degrees after top dead center (ATDC), or about 10 degrees after the start of injection, after which the wall film starts to reduce.