Cavitating Diesel Fuel Injector

The dynamic mesh and cavitation models in FLUENT are used to simulate a diesel fuel injector in this example. The transient approach tracks the motion of the internal plunger during a typical cycle of operation. The fuel injector operates at high pressure, and under these conditions, cavitation is shown to develop in the spray channels.

Fuel injectors are devices that supply a desired quantity of fuel at a controlled time to an internal combustion engine. For diesel engines, the fuel delivery timing is such that the combustion process in the engine occurs during fuel mixing and evaporation.

To enable spray penetration into the air at high compression density, and to obtain a fine spray for rapid mixing and reduced emissions, a high injection pressure is required. Diesel engine fuel injectors often operate at injection pressures of well over 2000 atm.

There are two key factors for judging the performance of a fuel injector. The first is its ability to control the amount of liquid fuel delivered to the engine. The amount of fuel delivered is controlled by the length of time the injector is kept open.

The opening and closing of the injector is typically achieved by the axial translation of a needle or plunger device, housed by the injector. The second parameter is the spray quality. The ability of the injector to achieve proper spatial and temporal spray characteristics is important because it impacts the mixing of fuel and air, as well as the engine emission levels.

High operating pressures in diesel injectors can, however, cause cavitation in the liquid fuel, leading to degradation in injector performance and structural damage.

Because of the small size of typical fuel injectors, experimental measurements of these phenomena are difficult

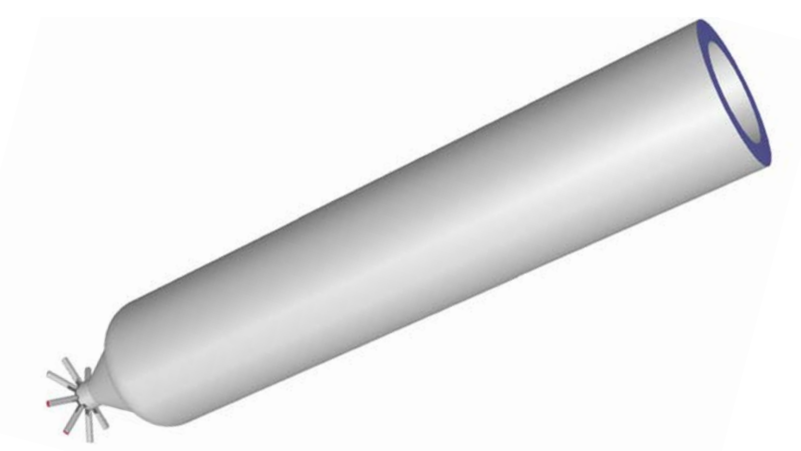

Figure 1: The fuel injector geometry

Figure 2: The hexahedral surface mesh

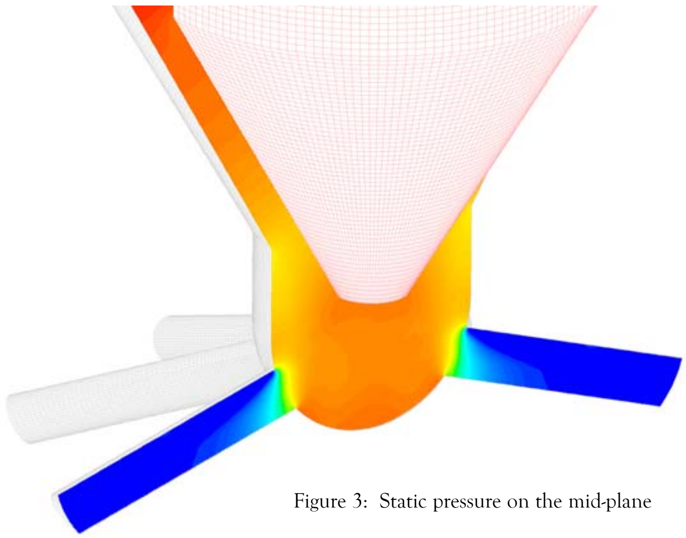

Figure 3: Static pressure on the mid-plane

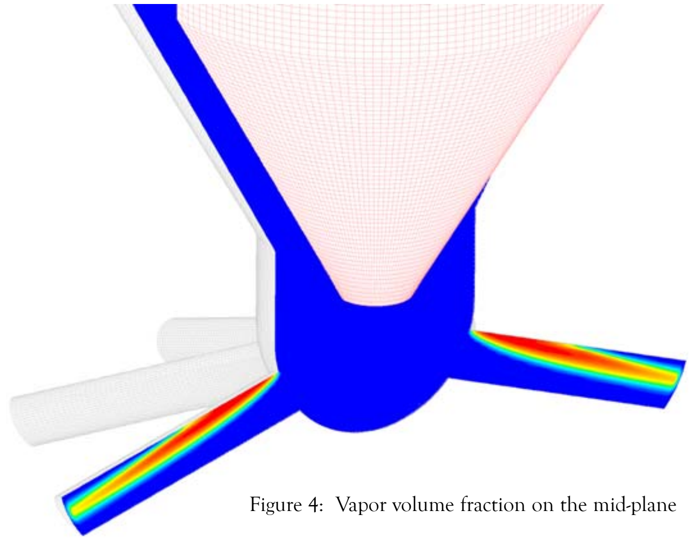

Figure 4: Vapor volume fraction on the mid-plane

to make. It is therefore of great benefit to numerically study highly cavitating injectors and predict the flow behavior at such operating conditions.

Toward this end, a fuel injector with eight spray holes was modeled in FLUENT. Instead of modeling 1/8 of the device, the full 360 degree geometry was simulated in order to asses flow variations, if any, at individual spray holes.

The inlet and outlets were modeled as pressure boundaries with 2400 atm of gauge pressure at the inlet and 0 at the outlets. The working fluid is a mixture of diesel fuel and vapor. The plunger location vs. time was prescribed, and the plunger motion was captured using the built-in dynamic mesh model.

The cavitation model was used as well, and took into account the presence of non-condensable gases in the fuel. While compressibility could have been included, in this simulation it was not. The realizable k-e model was used for turbulence.

The geometry of the fuel injector is shown in Figure 1. Diesel fuel enters through the inlet at the upper right (shown in blue), and exits through the eight spray holes (shown in red).

The mesh used consists entirely of hexahedral cells. The mesh size at the beginning of the transient simulation was about 800k cells. Figure 2 shows the mesh on one of the symmetry planes (gray) and on the plunger (red).

In Figure 3, the distribution of static pressure on a plane passing midway through two spray holes of the injector is shown. The injector is opening, so the plunger (shown in red) is moving upwards. The regions of low pressure are where cavitation is likely to occur.

This is supported in Figure 4, where the diesel vapor fraction distribution on the mid-plane is shown. The figure shows that the diesel vapors are carried outside the fuel injector through the outlet channels, which is indicative of heavily cavitating conditions.

Figure 5 shows contours of vapor volume fraction displayed on the spray hole channel surfaces at one instant in time. The pattern shows that the flow solution is fairly symmetric among all spray holes. Such flow symmetry was observed for other flow times throughout the simulation as well.

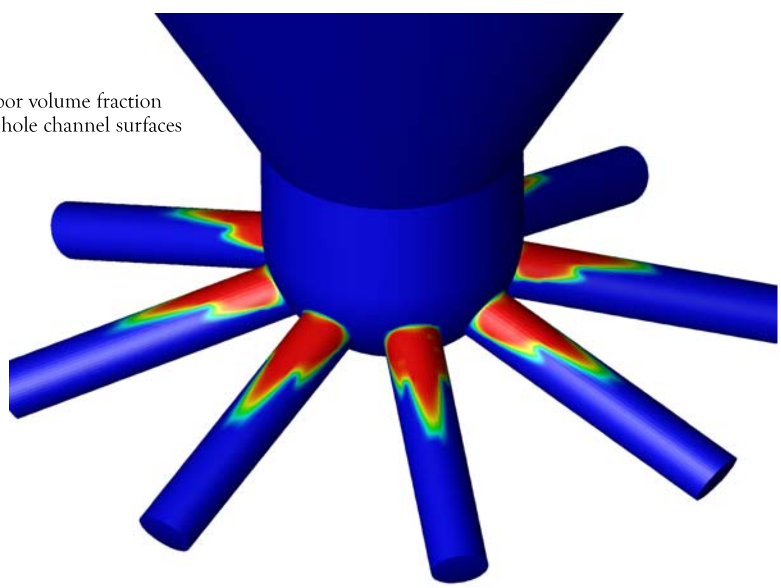

Figure 5: Vapor volume fraction on the spray hole channel surfaces

In summary, a transient simulation of a highly cavitating diesel fuel injector has been performed using the cavitation model in FLUENT. The results for diesel vapor distribution show that cavitation occurs mostly within the spray hole channels, with flow features being fairly uniform among the spray holes.

Numerical modeling of cavitation is an economical method of evaluating various injector designs through a range of operating conditions, since actual test measurements for these devices are difficult to conduct.