# IC Engine Piston Cooling

FLUENT is used in this example to simulate the cooling of an engine piston with an oil jet. The CFD analysis demonstrates the feasibility of performing such a study, along with the benefits that can be realized. Simulations of this type culd be performed to test ways to improve the cooling, such as determining the optimum speed and orientation of the oil jet, for example.

As new engines continue to be developed, the power output continues to increase, and this results in increased pressure and thermal loads on many engine parts, such as the pistons. Unfortunately, the strength of aluminum alloys used for piston manufacturing nowadays decreases with temperature. For lightweight pistons, the reduction in strength with heating should be minimized to maintain the mechanical integrity of the part.

This drives the need for piston designs that use strong, lightweight materials that can sustain a harsh thermal environment through improved oil cooling. Piston cooling also reduces the likelihood of carbonization and pre-ignition, caused by hot spots on sharp edges of the piston crown. Pre-ignition is undesirable as it causes the air-fuel mixture to ignite before the spark occurs.

A sample geometry of a flat top piston, representative of a typical gasoline engine, was simulated in FLUENT. An oil jet impinges on the back side of the moving piston, and the resulting heat transfer to the piston is computed.

The numerical simulation makes use of several built-in FLUENT models. The transient piston motion is captured by the dynamic mesh model. The free surface of the jet is tracked using the volume of fluid (VOF) model. The piston is treated as a conducting wall, so the conjugate heat transfer to the piston takes into account its thermal mass. During the setup phase of the problem this means that a computational mesh needs to be created for the fluid volume (containing the air and oil) and the piston volume as well.

The dynamic mesh model in FLUENT is used for situations where the shape of the domain is changing with time due to the motion of one or more domain boundaries. The model needs only a starting volume mesh and the description of the wall motion to work.

The description of the wall motion, the piston in this case, is a function of the length of the connecting rod, the piston stroke, and crank shaft speed as entered by the user in the in-cylinder GUI (graphical user interface) panel.

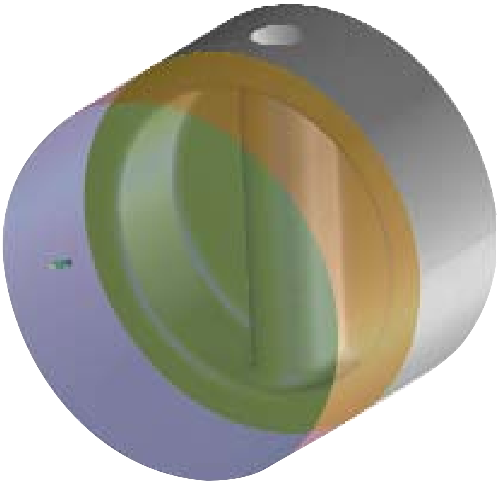

Figure 1: The piston geometry

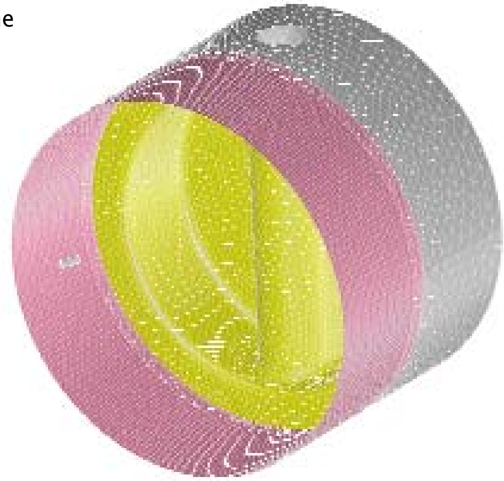

Figure 2: The surface mesh

FLUENT automatically computes the new position of the boundaries and updates the volume mesh at each time step.

The flow and thermal conditions simulated correspond to those of a sports car: the piston head temperature is initialized to 1300K to reflect the thermal conditions in the combustion chamber. The piston motion is set to a value that is typical of Formula 1 cars, and the oil jet, which enters through a tube, is given a speed of 20m/s, which is slower than the motion of the piston. The oil temperature is 400K. Turbulence is accounted for using the standard k-e model.

Figure 1 shows the geometry of the piston. Oil enters through a velocity inlet at the tip of a delivery tube (green, at left), and impinges on the underside of the piston (yellow). The piston wall is shown in gray, and a pressure outlet is shown in blue.

The surface mesh used is shown in Figure 2. The initial mesh size consists of 170,000 hybrid cells. The piston, whose volume is meshed with tetrahedral elements, moves back and forth as a rigid body.

The fluid volume underneath the piston is defined as a deforming zone, and a layering of hexahedral elements is used as the volume changes during the piston motion. The pressure outlet surface is removed from the image for better viewing.

In Figure 3, a series of images is used to illustrate the oil jet impingement on the piston underside, and the cooling effect that is achieved for an entire engine cycle. Shown are contours of temperature (with red the maximum and blue the minimum) along with the free surface of the oil jet. The changing fluid volume is shown in gray.

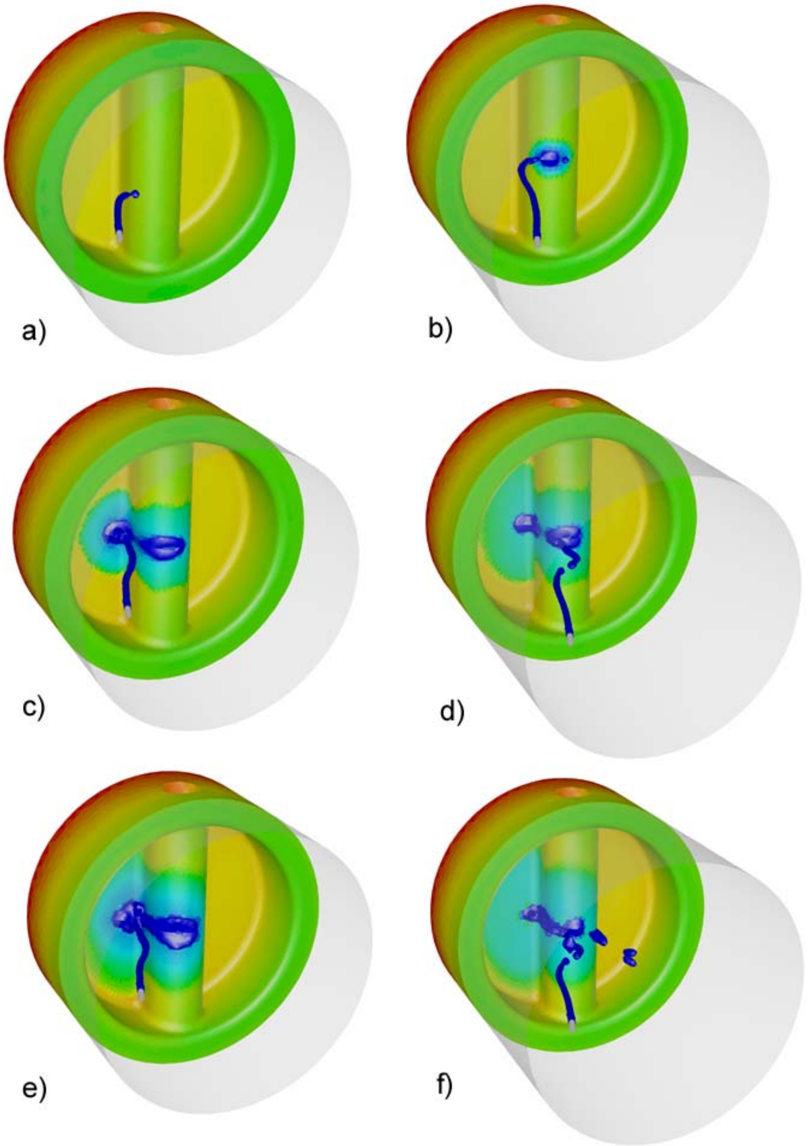

Figure 3: Temperature contours on the piston surface and the oil jet during two cycles of operation

The first image (a) is captured during piston motion towards top dead center (TDC). Both the piston and the jet travel in the same direction. The formation of a nodule at the tip of the jet indicates the development of instabilities in the liquid.

The second image (b) shows the piston moving towards bottom dead center (BDC), i.e. towards the jet. The jet impinges on the piston pin housing providing immediate local cooling, as is evidenced by the change of color on the piston. During this phase of the motion, the impingement location shifts slightly to the left.

The third image (c) shows the piston as it nears BDC and the increased wetted area on the piston underside. During the next stroke, the piston moves towards TDC again, i.e. away from the jet. Since the piston moves faster than the oil, the jet breaks up, as shown in the fourth image, (d).

The fifth image, (e), illustrates the increase in impingement area during the piston motion towards BDC, and further reduction in the piston underside temperatures.

The last image, (f), shows another round of jet breakup into droplets as the piston advances towards TDC. This oil splashing enhances the cooling.

A CFD analysis of engine piston cooling can provide useful information about the rate of piston cooling, detailed temperature distribution on the surfaces, as well as flow details on the dynamics of the oil jet impingement on the piston underside. The CFD results can help optimize the piston design to reduce thermal loads, and in the positioning of the oil delivery pipe so that cooling is enhanced. ■