

# Diesel Internal Combustion Engine

*A flow analysis of an internal combustion engine, which was done for a diesel engine company using the moving and deforming (MDM) model in FLUENT, is described in this example. The complex geometry, coupled with the complex nature of the motion, in which the valves and piston move, make this a challenging problem to solve. The MDM model is well suited to the task, however, because it works in an unstructured mesh environment, and has three algorithms for rebuilding the mesh with each new position of the moving parts.*

In this example, FLUENT's moving and deforming mesh (MDM) model is used to simulate the performance of an internal combustion (IC) engine. A cold flow analysis is performed for this purpose. Cold flow simulations for IC engines can provide valuable design information to engineers. These simulations allow for the effect on volume efficiency and/or swirl and tumble characteristics to be predicted based on changes in port and combustion chamber design, valve lift timing, or other parameters.

An analysis of an IC engine presents some of the most challenging problems faced by engineers in the automotive industry, because the simulations need to address the moving parts in an appropriate manner. The MDM model in FLUENT 6 is uniquely suited to this task, because as the parts move into a new position at each timestep of the calculation, the grid is automatically rebuilt. To make the

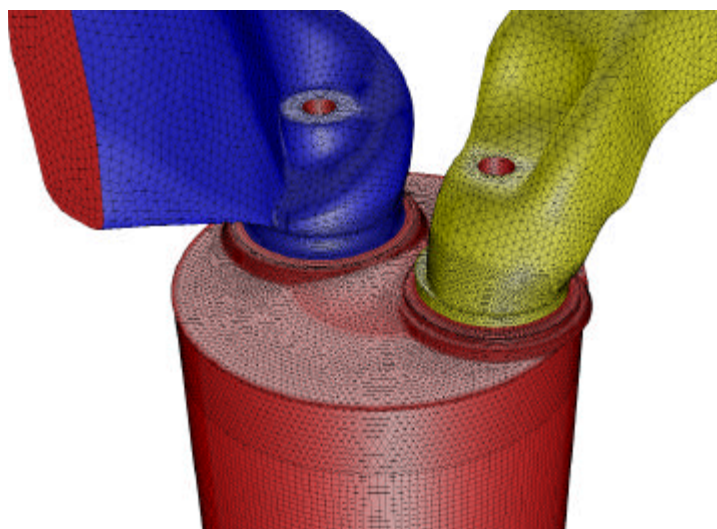


Figure 1: The cylinder (red) and intake (blue) and exhaust (yellow) runners of the diesel engine

model more flexible, it is based on an unstructured mesh framework. This allows an unprecedented capability to perform simulations of internal combustion engines, with the speed and ease-of-use traditionally associated with FLUENT products.

The MDM model in FLUENT 6 requires the user to provide an initial mesh along with a specification of the motion of the moving parts. In the case of the IC engine, the prescribed motions of the valves and piston are required. The solver automatically moves the components accordingly, and

then reconstructs the mesh, employing one, or a combination of two or three of the three available remeshing techniques or schemes: dynamic layering, spring smoothing, and local re-meshing.

The geometry of the diesel engine components, along with a surface grid, is shown in Figure 1. The cylinder, with the piston out of view in the bottom-most location, is shown in

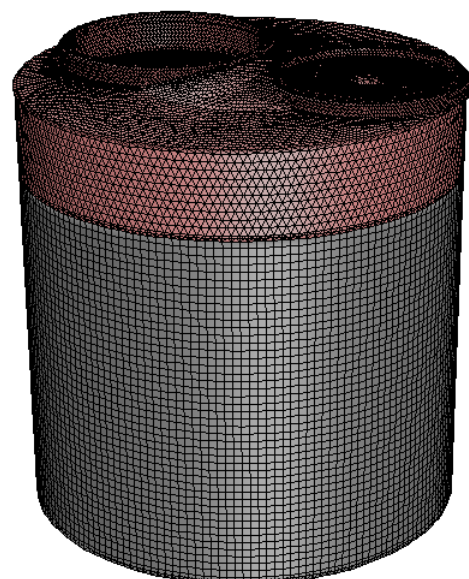
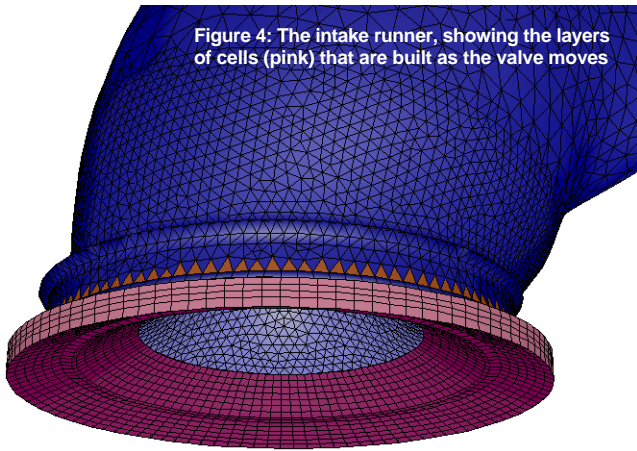


Figure 2: The cylinder, with the piston at bottom dead center, showing the cell layering (gray) that results from the piston motion



6.0's mesh preview feature can be invoked. The mesh preview feature allows the user to cycle only the mesh through its full range of motion without the calculation of flow physics. This provides a quick visualization and mesh integrity check prior to the

red. The intake runner (blue) is on the left, and the exhaust runner (yellow) is on the right. To define the mesh motion, the rigid body motion of the intake and exhaust valves, as well as that of the piston, are described using profiles or user-defined functions (UDFs). These profiles or UDFs prescribe the geometric translation of the moving wall zones as a function of time or engine crank angle.

submission of the more CPU intensive flow calculation. Due to the inherent cost associated with transient simulations, the overall mesh count throughout the cycle is of concern. In this case the resulting mesh varied from 262,000 elements at top dead center to 375,000 elements at bottom dead center.

In the cold flow calculation that was performed, pressure boundary

conditions were supplied at the intake and exhaust runner boundaries. It was assumed that the air was compressible. The standard k-ε model was employed. As an example of the results obtained, a series of snapshots of velocity magnitude contours on a plane cut through the center of the combustion chamber is shown for a complete cycle in Figure 4.

In summary, the MDM model in FLUENT model has been used to simulate transient IC engine performance. The model is easy to use, which makes the set-up and solution process straightforward. The model provides an unprecedented opportunity for engineers to examine the transient flow patterns, including combustion, inside the cylinder during the engine cycle.

To minimize the cell count and ensure proper resolution of the flow, dynamic layering is used as the valves and piston move. In Figure 2, layers of prisms (grey) are built as the piston drops to the bottom dead center position. In the remainder of the domain, including the intake and exhaust runners, a tetrahedral mesh is utilized, and in these regions, FLUENT does both spring smoothing and local re-meshing to dynamically alter the mesh throughout the engine cycle. In Figure 3, the layers of mesh, built as the valve begins its motion, are shown in pink.

Once the mesh motion and schemes are defined, FLUENT

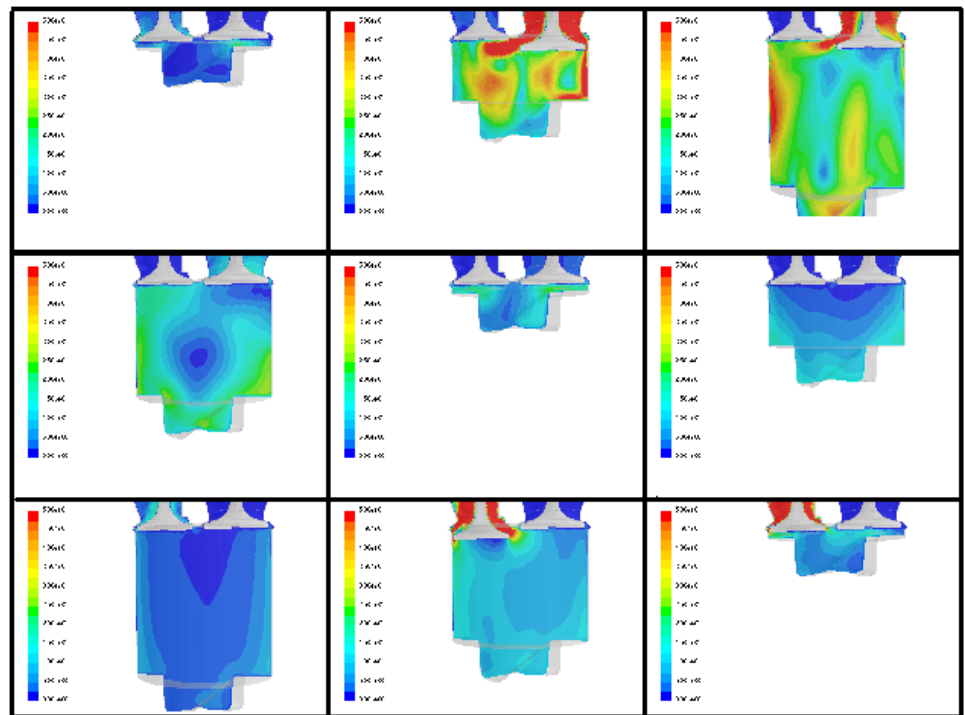


Figure 5: Contours of velocity magnitude on a slice through the center of the cylinder at nine times during the engine cycle