EX164

Automotive Water Pump

The multiple reference frames model in FLUENT is used in this example to study an automotive water pump. This steady-state model is popular for turbomachinery applications where rotating components play an important role in governing the flow. Results for pressure rise across the pump are in very good agreement with experimental measurements.

Courtesy of Tesma Engine Technologies

The multiple reference frames (MRF) model in FLUENT has been used by Tesma Engine Technologies to study the flow through an automotive water pump. Predictions for the pressure rise across the pump were found to be within 7% of experimental measurements. These results suggest that the MRF model is well suited for use in turbomachinery simulations involving rotating components.

Automotive water pumps are used to circulate coolant through the engine. Most water pumps, like the one simulated here, are centrifugal pumps consisting of inlets, outlets, a housing, a shaft, a rotating impeller, a volute, and optionally, a shroud. The rotating impeller found inside the pump poses a challenge to CFD analysis because the blades move relative to other stationary pump components. The technique used in this example to capture the impeller behavior makes use of the MRF model. This steady-state model allows for multiple sub-domains (or fluid zones), each of which can rotate independently. The flow in each rotating zone is solved in the rotating frame of the zone itself, while the flow outside the rotating zones is solved in the stationary frame. Each fluid zone interacts with adjacent fluid zones by a steady transfer of information across pre-defined interfaces. The model is popular because it is less computationally demanding than sliding or moving mesh techniques, which require a transient calculation.

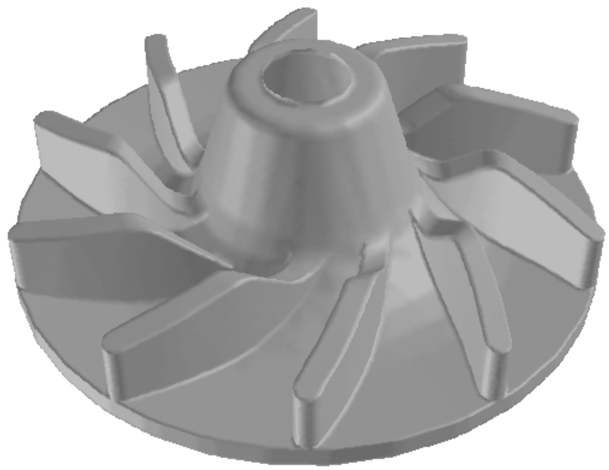

Figure 2: The impeller detail

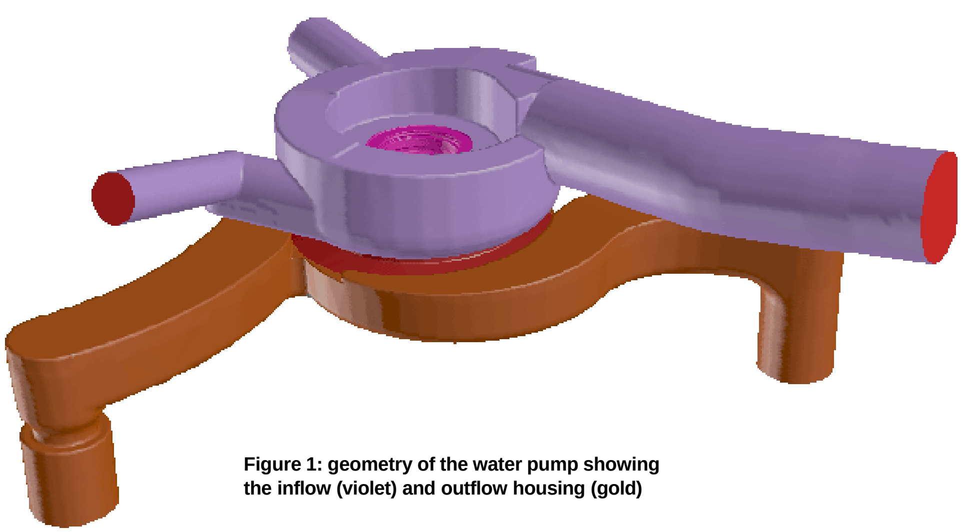

Figure 1: geometry of the water pump showing the inflow (violet) and outflow housing (gold)

In the simulation, mass flow inlet and pressure outlet boundary conditions were imposed at the flow boundaries. The standard k-ε turbulence model was used. The impeller was assigned a rotational speed of 6,000 rpm.

FLUENT WWW.FLUENT.COM

Figure 1 shows the water pump geometry provided by Tesma Engine Technologies for the simulation. Three water inlet channels are shown in violet. These lead to the circular pump inflow housing, which is positioned above the pump and outflow housing, shown in gold. The ten-blade impeller is positioned in the central region of the gold outflow housing. Two outflow channels are shown, each of which terminates with a 90 degree bend prior to connecting to the outgoing cooling lines.

The impeller detail is shown in Figure 2. It consists of a round plate to which ten slightly curved and tapered vanes are mounted. When the impeller turns, the coolant is drawn in from above and expelled radially outward by centrifugal force.

Due to the complexity of the geometry, a hybrid mesh was generated. As shown in Figure 3, the inlet ports were meshed using hexahedral elements, due in part to the simple nature of this part of the geometry. The remainder of the domain was meshed using tetrahedral elements. A total of about 1.6 million cells were used, which is at the high end for water pump applications.

Figure 4 shows pressure contours on the pump and outflow housing. Velocity vectors on a flat surface slicing through the impeller are also shown. The vectors show the swirling flow induced by the pump and the subsequent channeling of the water into the two exit passages. The pressure contours indicate that there is a large pressure loss at the left-hand exit due to a constriction inside the piping. Indeed, the overall pressure rise across water pumps is one of the most important parameters for designers. In this simulation, the pressure rise predicted by FLUENT agrees with the experimental data to within 7%. Engineers at Tesma (and other automotive suppliers) have reported that predictions provided by FLUENT are consistently in good agreement with the experimental data.

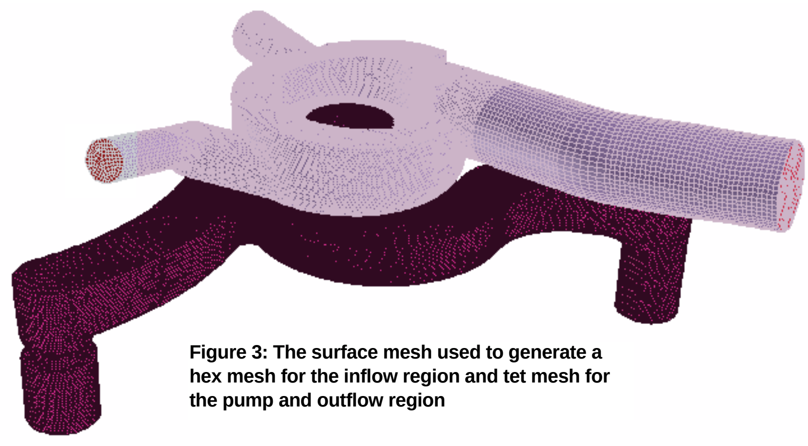

Figure 3: The surface mesh used to generate a hex mesh for the inflow region and tet mesh for the pump and outflow region

In summary, the MRF model in FLUENT has been used to predict the pressure rise across an automotive water pump. Results are in very good agreement with experimental measurements carried out at Tesma Engine Technologies. Pressure contours on the outflow housing helped to identify a significant pressure loss near a constriction in the piping. This helped the engineers to understand why the pump was performing below its designed pressure specifications. The information provided by this and subsequent simulations will help them with design modifications that will be used to improve the pump performance in other models.

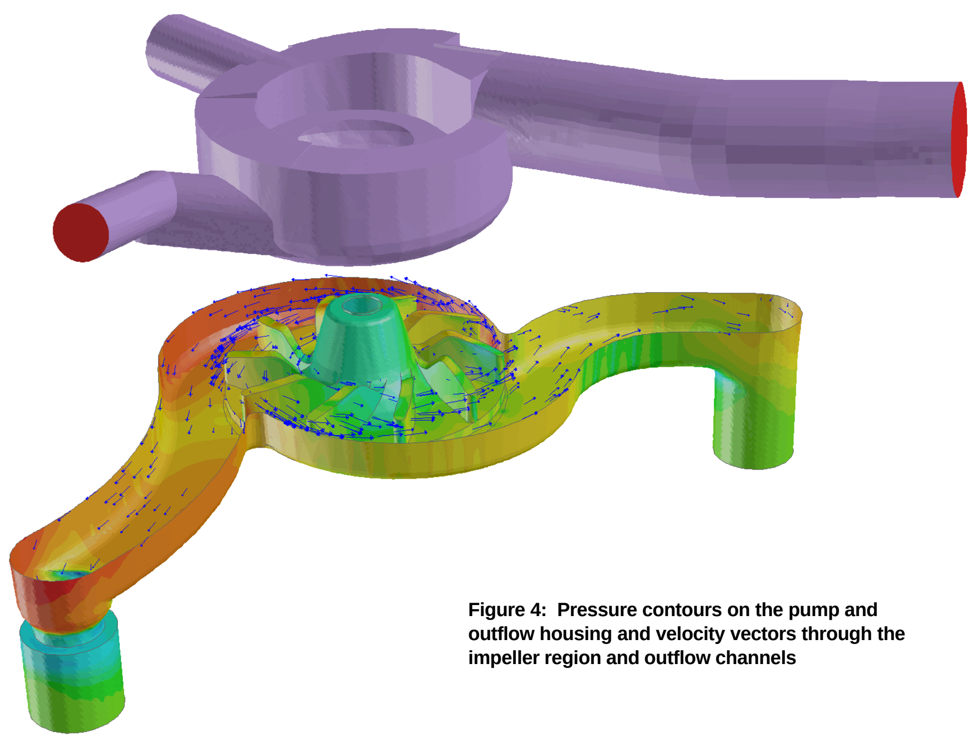

Figure 4: Pressure contours on the pump and outflow housing and velocity vectors through the impeller region and outflow channels