EX238

Gerotor Pump

In this example, an automotive assist gerotor pump is analyzed using FLUENT 6.2. The goals of the CFD analysis are to visualize the flow through the pump and validate the methodology used. The model predictions are shown to be in good agreement with experimental measurements. Predictions of forces and torques on certain pump components are also examined to ensure that excess wear can be avoided.

Gerotor pumps are commonly used for cooling, lubrication, and filtration systems, pumping liquids such as oil, transmission fluid, and fuel. They provide high volumetric efficiency and smooth pumping action, and they work well with a wide range of fluid viscosities. The pump operates on a simple principle: the working fluid is drawn into and squeezed out of a cavity formed by two rotating, intermeshing gears. The rotating inner gear has fewer teeth than the outer gear, and as it rotates relative to the outer gear, one portion of the cavity opens and is filled by the liquid. As rotation continues, that portion of the cavity closes and the liquid is squeezed out, and an adjacent portion opens. In theory, the pumping action is a function of the gear rotational speed only, and a constant discharge is delivered regardless of the operating pressure. In practice, however, leakage through the gaps formed by the meshing gears increases as the pressure increases.

A CFD simulation of the gerotor pump can be used to better understand its performance. In particular, such a simulation can offer a mechanism for visualizing the flow through the pump, and for predicting the leakage at a given operating pressure. The gears are driven by a rotating shaft from an electric motor, and CFD can predict the torque that the electric motor needs to deliver. In addition, the pressure in the seal cavity can be studied, in hopes that the seal life can be prolonged. The forces acting on the shaft can also be predicted. These are important because they directly influence the load on the bearing that supports the shaft.

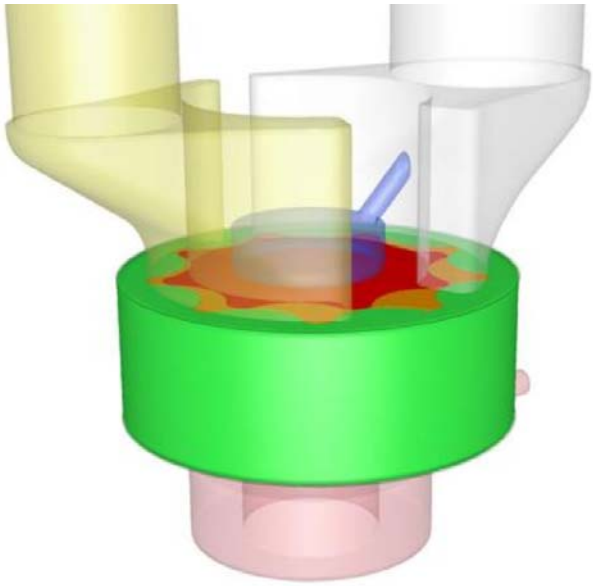


Figure 1: The pump geometry

Using FLUENT, the transient gerotor motion is simulated with the dynamic mesh model. This model is useful whenever the shape of a domain changes with time as a result of the motion of the domain boundaries. The dynamic mesh model needs only a starting volume mesh and the description of the motion of the moving boundaries. At each new time step, a new volume mesh is generated automatically by FLUENT, based on the new positions of the boundaries.

Because the gear region is an extrudable geometry, the 2.5D meshing strategy is used. This novel meshing technique is essentially an extrusion of a 2D triangular mesh along the normal axis of the dynamic zone. At each

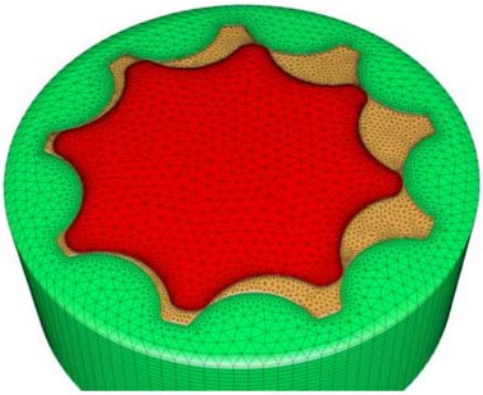


Figure 2:Surface mesh on the fluid in the gerotor region

Copyright © 2005 Fluent Inc.

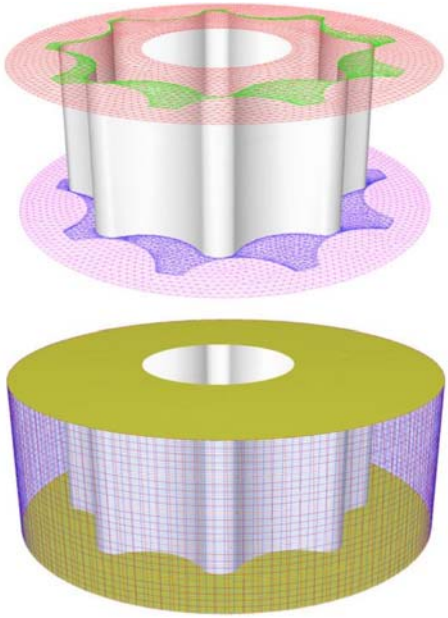


Figure 3: Non-conformal mesh interfaces in the gerotor region

time step, the triangular surface mesh is recreated and smoothed on one side, and the changes are then extruded to the opposite side. The technique requires less time than regular volumetric meshing for the equivalent volume.

The pump consists of an inner gear with 8 teeth rotating at 4000 rpm, and an outer gear with 9 teeth rotating at 3555 rpm. A realistic minimum clearance between the two gears is used. The working fluid is oil with a temperature-dependent viscosity, and the standard k-epsilon model is used for turbulence.

Figure 1 shows the geometry of the gerotor pump. Oil enters through the pressure inlet (gray), squeezes through the gap (orange) formed by the inner (red) and outer (green) gears and is pushed into the outlet passage (yellow). The seal cavity (pink) and shaft relief volume (blue) are also modeled. The CFD model includes fluid volumes for clearances formed by the rotor and the top, bottom, and side surfaces of the pump housing.

The mesh detail of the fluid volume (orange) formed by the inner (red) and outer (green) rotor is shown in Figure 2. The hybrid mesh for the whole pump model has about 800,000 cells.

Figure 3 (a and b) shows non-conformal interfaces around the gerotor section. Figure 3a shows non-conformal interfaces between the fluid gap and the top and bottom clearances, while Figure 3b shows one between the outer wall of the larger gear and the side clearance. The ability of the solver to handle non-conformal interfaces makes moving mesh problems easier to set up and solve.

The static pressure distribution on the surface of the fluid volume formed by the two gears is shown in Figure 4. The pressure distribution in the fluid is used for predicting cavitation (which was not included in this simulation), and for assessing the pressure ripple effect (or pressure change between the rotating gears).

Figure 5 shows contours of static pressure on selected surfaces of interest, such as the shaft, seal cavity, relief valve, and gerotor. The pressure distribution on these surfaces can be used to calculate the forces acting on the pump components.

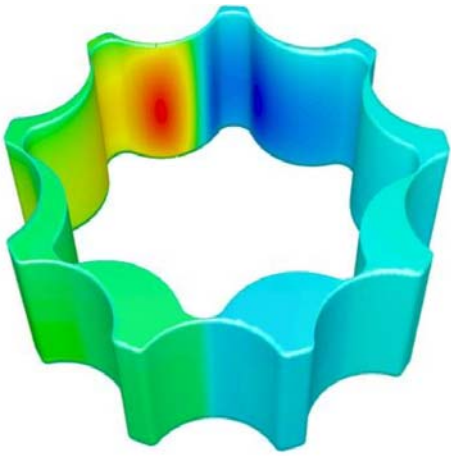


Figure 4: Static pressure on the fluid surface in the gerotor region

Based on the CFD results, the flow rate through the pump and torque were computed. These predictions were found to agree within 8% and 7.5% of experimental measurements, respectively. The CFD simulation revealed that pressure forces on the seal were exceeding the allowable limit. With this information, the original pump design was modified. Plans for future work are to analyze the pump performance for various oil viscosities, including cold start conditions, and to include cavitation in the model.

Courtesy of EMP Inc.

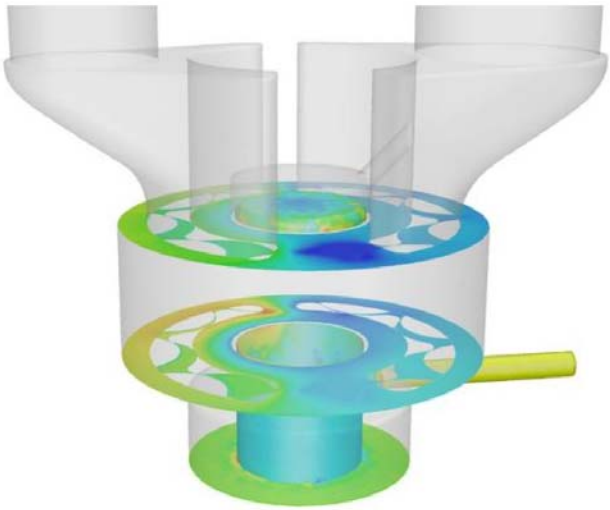


Figure 5: Contours of static pressure on selected surfaces inside the pump

Copyright © 2005 Fluent Inc.