Thank you!

Thank you for downloading one of our ANSYS whitepapers – we hope you enjoy it.

Have questions? Need more information? Please don’t hesitate to contact us! We have plenty more where this came from.
Automotive Hydraulic Pump Simulation

Automotive pumps take two forms: hydraulic and centrifugal. Hydraulic pumps are positive-displacement pumps, with flow rate a direct function of rotational velocity. In the automotive industry, hydraulic pumps include gear pumps, gerotor pumps, vane pumps and piston pumps. Vehicle applications include fuel pumps, super-charging pumps, power-steering pumps, automatic transmission pumps and many others. Hydraulic pumps must be designed to avoid leakage and cavitation. Additional objectives include minimizing fuel noise and maximizing volumetric efficiency.

Engineering R&D teams commonly use simulation technology, especially computational fluid dynamics (CFD), in pump design. But CFD simulation of hydraulic pumps can be challenging. The most basic challenge is that the flow volume or control volume continuously changes over time. Therefore, the CFD solver must be able to deform and recreate mesh on the fly. Furthermore, the clearance between moving and stationary parts is on the order of a few microns. Accurately modeling flow through these leakage paths is essential to accurately predicting volumetric efficiency. It can be difficult to provide sufficient mesh density within this gap.

The working fluid, which could be lube oil or fuel oil, usually contains dissolved gases. At extremely low pressures, this dissolved gas can separate; these multiphase effects must be included in the simulation to predict cavitation, for example. The fluid also experiences tremendous compression, so compressibility of the fluid must be taken into account. In some cases, the displacement of rigid thrust plates as a result of fluid forces must be considered. In other applications, fluid–structure interaction (FSI) is needed to model flexible thrust plates.

A Complete Toolset for Automotive Pump Simulation

ANSYS provides a complete set of tools for addressing these challenges. The ANSYS® Workbench™ platform enables engineers to easily define simulation workflows, including geometry preparation, meshing, solution and post-processing. The environment has the capacity to incorporate multiple physics such as CFD and structural dynamics. Data is automatically transferred across different components of the workflow, even across different physics, allowing users to easily perform FSI simulations. The design point capabilities in ANSYS Workbench automate the study of variations in design parameters, such as clearance gaps, and different operating conditions, such as rpm. Users can perform goal-driven optimization based on built-in design exploration capabilities.
Geometry and meshing tools like ANSYS DesignModeler™ and ANSYS Meshing make it possible to quickly create CFD models ready for simulation. Users can partially automate geometry and mesh creation for many types of pumps by defining a process that can then be re-used like a template on other similar models. Automatic geometry repair tools, virtual faces to assist high-quality meshing and work-sheet-based meshing procedures facilitate mesh creation workflow.

ANSYS CFD solutions incorporate all the building blocks required to perform highly accurate and fast CFD simulations. A rich suite of mesh motion and deformation algorithms provide flexibility to model different types of pump motion. Predefined standard subroutines enable engineers to model gerotors and vane pumps with a high-quality hexahedral mesh. Clearance gaps are resolved with sufficient numbers of mesh elements. Two different approaches address other pump types, like gear and crescent pumps: the simple mesh-sweep-based approach (popularly known as 2.5-D remeshing) and the newer mesh-replacement approach in which engineers can sew predefined meshes together at discrete points in time and interpolate the solution from one mesh to another. Piston pumps can benefit from mesh-deformation approaches like layering, in which a layered set of elements is added and deleted as the piston reciprocates in the chamber. Other approaches such as mesh smoothing and tetrahedral remeshing provide even more flexibility in modeling pump motion.

A number of capabilities help to ensure accurate results: physical models such as liquid compressibility, multiphase models that model dissolved gases to predict cavitation, temperature-dependent variation of saturation vapor pressure for cavitation modeling, and on-the-fly solution interpolation across meshes.

ANSYS CFD solutions support high-performance computing (HPC), substantially reducing simulation turnaround time by scaling the number of parallel computing solvers.

ANSYS CFD-Post software provides fast, user-friendly post-processing of solution data. A user can quickly generate animations of solution variables as the pump operates.

**Positive Displacement Pumps**

In positive displacement pumps, flow rate is a direct function of pump rpm. But as rpm increases, cavitation significantly reduces flow rate. CFD simulation can help to identify the threshold pump speed at which cavitation sets in along with the resulting loss in flow rate. An example of a gerotor pump is shown in Figures 1, 2 and 3.

![Structured hexahedral mesh in gerotor core](image1.png)

![Cavitation bubble in gerotor pump](image2.png)

![Comparison of gerotor pump simulated data with test data](image3.png)
The pump’s operating range varies from 1,000 rpm to 7,000 rpm. A high-quality hexahedral-element-dominant mesh is created in the gerotor core. A subroutine-based node-motion technique is deployed to allow mesh motion. The Schnerr–Sauer cavitation model is used. The Workbench design points feature is used to simulate eight different pump speeds for the same model. The threshold rpm and the subsequent loss in volumetric efficiency is accurately predicted. The eight operating conditions were simulated in less than 1.5 days on an 8 GB AMD 64-bit quad-core desktop.

**Vane Pump Case study**

To illustrate how CFD can be used to make design decisions, a vane pump simulation is shown beginning with Figure 4. As fluid passes from a vane pump’s rotating chambers into static inlet and outlet ports, it experiences a sudden change in available volume. This causes pressure ripples that generate noise. Two different port shapes, a baseline and another with a v-notch, are analyzed to compare the relative pressure-ripple effects at different pumps speeds. The results in Figure 6 show a 37.5 percent reduction in peak slide pressure with the v-notch. The v-notch also reduces pressure fluctuations at all frequencies in the outlet port.

It’s not easy to define the motion of each and every mesh node in a gear pump using a subroutine, since gear pump topology doesn’t easily allow creation of structured hexahedral elements. A more common approach is to perform two-dimensional remeshing of the facets of triangular wedge-shaped elements on one of the faces and sweep the updated mesh in the third direction. However, this approach, known as 2.5-D remeshing, does not resolve the tiny clearance gaps with a sufficient number of mesh elements.
A new approach addresses meshing the tiny clearance gaps with a sufficient number of elements. A series of 19 meshes, each representing one pitch of the gears, is created at discrete positions of the driving and driven gears. These meshes uniquely represent the motion of the entire 360 degree rotation of the gears.

A workflow automatically generates meshes with a well-resolved clearance region. A script places these meshes one by one in the CFD solver. The CFD solver interpolates the solution from each mesh to its successor, allowing mesh deformation using diffusion-based smoothing techniques. This approach accurately predicts torque, wall forces and leakage in gear pumps.
Gerotor Pump Leakage Case Study

Simulation of gerotor pumps typically requires prediction of torque, cavitation, pressure within the seal cavity and forces on the shaft. Technical challenges include narrow gaps and large deformation of the fluid volume. Alternative simulation approaches include the 2.5-D remeshing approach, the pure-hex approach with a user-defined function (UDF) for node movement, and the immersed solid method. The 2.5-D remeshing approach provides more elements in the core, but it is not good at predicting leakage. The pure-hex approach makes it possible to put more elements in the gaps between gears to accurately predict leakage.

The example shown in Figure 10 uses the pure-hex mesh approach to look at leakage in a gerotor pump. Four operating conditions, consisting of different speeds and pressures, are investigated. The results show minimal cavitation; however, the volumetric efficiency is low. Tip-to-tip clearance and leakage paths are seen to have a major impact on volumetric efficiency. The leakage paths are kept constant and tip-to-tip clearance is varied. Figure 11 shows that the simulation accurately predicts volumetric conditions under a wide range of operating conditions.

Conclusion

Unlike custom CFD solutions for pump design, ANSYS simulation tools offer the flexibility to develop a workflow system that can be used to model different design parameters and operating conditions for nearly any pump. The same toolset can be used to model multiple physics and to simulate nonpump components of the pump system. The powerful modeling capabilities of ANSYS simulation tools provide engineers with high-fidelity insights that lead to better pump designs.