Numerical Simulation of Oil Flow in a Power Steering Pump
Celso K. Takemori and Emilio E. Paladino
Engineering Simulation and Scientific Software – ESSS
Leonardo Lessa
DHB Componentes Automotivos

Abstract
This work presents the implementation of a computational model to simulate oil flow in a power steering pump. The simulations have the objective of visualize the flow pattern and to identify some possible flow induced noise sources by analyzing the pressure fluctuations. Another important issue investigated is the oil cavitation due to the presence of very low pressure regions. The mathematical model is based on the mass, momentum and energy conservation equations. Moving mesh capability had to be used because of the variations of the domain geometry with the pump rotation. The oil was considered as compressible liquid, because of the large pressure range operation. CFX-10.0 Solver was used to perform the simulations. Hybrid HEXA/TETRA/PRISM mesh was generated using ICEM-CFD 10.0. The results investigate possible cavitation (low pressure) regions, noise sources, pressure transients and recirculation zones. A comparative study between two geometries is presented, as well as the influence of geometric details used to improve performance and reduce pressure fluctuations and noise levels.

Introduction
Noise plays a fundamental role when an automotive manufacturer needs to choose a hydraulic steering pump. The quietness of the interior of automobiles is perceived by consumers as a measure of quality and comfort. Great strides have been achieved in isolating interiors from noise sources. In addition, as engines are being improved for quiet operation, the noise that comes from other components, like the steering system, is becoming more and more apparent and customers are now looking for cars with very low noise levels. To address this, automotive industry suppliers are spending large efforts in the noise reduction in several automotive systems and components and the noise level is one of the main requirements for the product acceptance by car manufacturers. One important noise source in the steering system is the positive displacement pump [1], used to pump the oil at very high pressure into the driving pistons of the steering system. This device operates between pressures of 1 and 120 bar, generating strong pressure fluctuations in suction and discharge regions.

The study presented in this work was focused on the investigation of pressure fluctuation which are the main noise sources, and on the influence of geometric details on these fluctuations. In addition, another point investigated was the low pressure regions present within the pump, which could give place to oil cavitation.

Computational model
The problem is governed by transient Navier-Stokes equations that represent continuity and momentum conservation for newtonian fluids. However, due to flow regime, a model for the turbulent closure is necessary. The turbulence model used was standard k-ε. In addition a compressible liquid was considered due to the high pressure ranges between inlets and outlets.
**Governing equations**

Continuity and momentum conservation equations are given by,

\[
\frac{\partial}{\partial t}(\rho) + \nabla \cdot (\rho \mathbf{U}) = 0 \tag{1}
\]

\[
\frac{\partial}{\partial t}(\rho \mathbf{U}) + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = -\nabla p + \nabla \cdot (\mathbf{T} + \mathbf{T}_{Turb}) \tag{2}
\]

where \( \rho \) is density, \( \mathbf{U} \) is velocity vector, \( p \) is pressure, \( \mu \) is dynamic viscosity, \( \mathbf{T}_{Turb} \) is turbulent stress tensor. The convective term \( \rho \mathbf{U} \mathbf{U} \) represents a differential balance of momentum, on a control volume. The right hand side of equation (2) represents the summation of external forces: pressure and shear stresses.

**Fluid model and properties**

Compressibility effects were modeled through an equation of state expressing density as a linear function of pressure as,

\[
\rho = 850 + 10^{-6} \cdot p \tag{3}
\]

where \( \rho \) is density given in \( \text{kg/m}^3 \) and \( p \) pressure in \( \text{Pa} \). It results in a sound speed of 1000 m/s and was calculated considering a volume variation of 1.4% within operational pressure range (0 and 120 bar relative). Dynamic viscosity was considered constant with a value of 11.0 cSt (@ temperature of 80°C).

**Turbulence model**

The standard k–\( \varepsilon \) turbulence model was used with a scalable wall function. This kind of wall function automatically detects if the use wall law will be necessary or not, depending on the grid refinement near walls.

The k–\( \varepsilon \) turbulence model is considered the standard model for industrial problems. This model has showed satisfactory in this case. Other models like SST, could be used, but with an increasing in computational cost and without assurance of better results. Improvements in using another turbulence models depends on each case and experimental data was not available, in order to assess the improvements introduced by a more sophisticated model.

![Figure 1 – Boundary conditions. Blue: Outlet, 120 bar. Red: Inlet, 0 bar.](image)
**Domain**
The model considered the full three-dimensional detailed geometry of the pump, including the deformation and relative movement in the rotor domain. This geometry is showed in Figure 1.

**Boundary conditions**
Prescribed pressure of 0 bar was used as boundary condition at inlets and 120 bar at outlets (see Figure 1). No slip conditions were prescribed at all walls and at no overlapping regions of the domain interface, showed in Figure 2. In these regions fluid is in contact with static surface, not considered in the model.

![Figure 2 – Non-overlapping interface regions (white)](image)

**Mesh generation**
The geometry was created using Unigraphics and imported into Ansys ICEM CFD 10.0 via STEP file format. Tetrahedral elements were used to discretize the complex geometry of inlets and outlets, as well as prism elements on walls to better capture boundary layer gradients. The mesh on rotor is entirely hexahedral, as shown in Figure 2, in order to have a better control of the mesh quality along the deformation process (bad angles, deformed elements etc.).

![Figure 3 – Mesh with hexahedral, tetrahedral and prismatic elements.](image)
**Numerical solution**

The governing equations were solved in conservative way, using Element Based Finite Volume Method and a coupled solver, which greatly improves the solution robustness [4], [5].

Other important resources used in this model were general grid interfaces and moving mesh (including deformation). As the problem includes relative movement between rotating parts (rotor and vanes) and stationary parts (inlets and outlets), general grid interfaces were necessary to consider that in a conservative and implicit way. In addition, as the thickness of cam annulus varies with rotation angle, it was necessary to prescribe the mesh deformation of the rotor domain, which was ruled by cam annulus.

The entire model was implemented in CFX-10.0 package, used in this study, and mesh movement was defined via FORTRAN User Routine. This routine reads a file containing the position of the external point of the rotor define by the cam annulus. This allows to easily evaluate different geometries of the cam annulus, in an easy way.

**Results**

Analysis was focused in two aspects: Pressure variations with time, which result in unwanted noise sources, and low pressure locations, which could originate cavitation problems. Two operational conditions were studied (3000 and 8000 rpm) and two geometries were analyzed at 3000 rpm. The focus at 8000 rpm is the cavitation problem (low pressure regions) as noise generated by the pump is irrelevant compared with the engine noise at this rotations.

Results present a comparative analysis for the same geometry at 3000 and 8000 rpm and for the two geometries studied.

**Case 1: Geometry #1, 3000 rpm**

The main reason for appearance of low pressure regions was the pressure relief of confined fluid when it discharges in the inlet (low pressure) region (Figure 5, point D).

The pressure relief also causes severe pressure fluctuations at inlets. At outlet regions, these variations are attenuated by a geometric detail called ‘v-notch’ (Figure 4) which allow a gradual pressure relief.

The effect of “v-notch” was one of the focus of this study and could be visualized for this case in Figure 5. The slope of curve at point A is lower than at point C. As a consequence, pressure fluctuations at point B are also lower than at D.
Figure 5 – Pressure variation at a point on a slide wall.

Figure 6 shows the FFT of pressure fluctuations at inlets and outlets. It could be seen that they have similar intensities. This suggests that noise sources have similar importance on both locations, but peak frequencies are a bit higher at inlets where there are no v-notch.

Figure 6 – Pressure as a function of frequency.

As we have a symmetric geometry, the movement of rotor induces an asymmetric flow (Figure 7), generating unbalanced forces on shaft. Figure 7b also shows an increasing in recirculation zones and pressure transients associated to that.

Figure 7 – Asymmetric flow due to symmetric geometry and rotation.
Case 2: Geometry #1, 8000 rpm

As expected, pressure fluctuations levels in this case are larger (about 10 times) than at 3000 rpm (Figure 8). Nevertheless, at this rotation, noise generated by engine is much more relevant.

![Figure 8 - Comparison of pressure fluctuations between 3000 rpm and 8000 rpm.](image)

As commented, the main issue at high rotations is cavitation. Figure 9 shows the iso-surface at inlets, delimiting regions of absolute pressures lower than 0.4 bar for (a) 3000 rpm case and (b) 8000 rpm case.

![Figure 9 - Low pressure zones at inlets.](image)

In frequency domain is possible to see that pressure levels are higher at inlet (Figure 10). From acoustic point of view, the related frequencies should be weighted using, for example, A-weighting\(^1\), reducing a lot these levels.

---

\(^1\) The objective of A-weighting is represent equal loudness curves, or simply represent the human hearing sensitivity.
Case 3: Geometry #2, 3000 rpm

Some modifications seeking for lower noise levels and cavitation problems were introduced. The v-notch helps to reduce pressure fluctuations. On geometry #1, this feature was present only at outlets. On geometry #2 a v-notch was placed also at inlet, in order to evaluate their influence on pressure fluctuations.

The use of the v-notch decreases significantly the pressure fluctuations at inlets. Pressure oscillations are strongly related with radiated noise.

Figure 10 – Pressure at inlet and outlet for 8000 rpm.

It is also possible to evaluate eventual cavitation problems by analyzing pressure transients at vanes (Figure 12). The solid curve, with v-notch, shows pressure levels found at a vane passing through inlet with v-notch, while the dashed curve refer to a vane passing through inlet without v-notch. Although the ‘sub-pressure’ levels are still high with v-notch, it is possible to reduce their values with geometric modifications like profile, length, depth etc.

Figure 11 – V-Notch influence on pressure fluctuations at inlets.
The effect of v-notch on low pressure zones could be visualized on Figure 13. The region delimited by red iso-surface, representing an absolute pressure of 0.6 bar, is smaller with v-notch (a) than without (b).

Another modification introduced in geometry #2 was the inclination of the outlet chamber, in order to avoid recirculation observed in geometry #1. With that modification, it was possible to reduce it, improving oil flow through the pump (figures are excluded because of confidentiality reasons).

**Conclusions**

Computational Fluid Dynamics has showed to be a valuable tool for the design of better positive displacement pumps. Important issues as noise sources and cavitation are related to the flow pattern, which can be fairly well predicted with this tool too expensive to be experimentally assessed.

It was possible to understand the influence of geometric details, like the “v-notch” and how it helps to smooth pressure variations.

Although cavitation was not modeled, it could be identified by regions with pressure lower than saturation point.

In this way, it is now possible to pre-evaluate designs more accurately at lower costs and times, which will considerably reduce turnaround time of a physical model test.
References


