# Trajectories of Solid and Gaseous Particles in a Hydraulic Reservoir

#### Vito Tic, Uni.-Dipl.-Ing.

OLMA d.d., Poljska pot 2, SI-1000 Ljubljana, Slovenia, E-mail: vito.tic@olma.si

#### Dr. Darko Lovrec

University of Maribor, Faculty of Mechanical Engineering, Smetanova 17, SI-2000 Maribor, Slovenia, E-Mail: d.lovrec@uni-mb.si

#### Abstract

Contaminants of hydraulic fluid are broadly defined as any substance that impairs the proper functioning of the fluid. Hydraulic fluid can be contaminated by air, particles, water, and foreign fluids. Fluid contamination can cause numerous problems in a hydraulic oil system including component damage, unacceptable noise, poor component response and severe fluid degradation. The paper is focused on two major contaminants which should be considered when designing a hydraulic reservoir: air and particle contamination.

Proper reservoir design can prevent the occurrence and help solve solid and air contamination in hydraulic fluid. Hydraulic reservoir should be designed in such a way to stabilize and direct the oil flow, so the oil has enough time to extract air bubbles and solid particles from the fluid. To see and understand flow patterns inside the reservoir, the advantage of using simulation techniques in the field of reservoir design will be explained.

The paper investigates trajectories of solid and gaseous particles in an hydraulic reservoir. Research is based on transient simulation using Ansys Workbench. Results obtained focuses on sedimentation of solid particles and elimination of gaseous particles in a hydraulic reservoir.

KEYWORDS: hydraulic reservoir, simulation, CFX, trajectories, air bubbles, solid particles

#### 1. Introduction

The 'reservoir' as the name suggests, is a tank which provides uninterrupted supply of fluid to the system, by storing the required quantity of fluid. The hydraulic fluid is considered the most important component in a hydraulic system or in other words its very heart. Since the reservoir holds the hydraulic fluid, its design is considered quite critical. The reservoir in addition to storing the hydraulic fluid performs various other important functions such as dissipating heat through its walls, conditioning of the fluid by helping settle down the contaminants, aiding in the escape of air and providing mounting support for the pump and various other components. /1/

The proper reservoir design for a hydraulic system is essential for the overall performance and for the individual components life. It also becomes the principle location where the fluid can be conditioned in order to enhance its suitability. Sludge, water and foreign matter such as metal particles, have a tendency to settle down in the reservoir, while the entrained air extracted from the oil is allowed to escape in the reservoir. This makes the construction and design of hydraulic reservoirs all the more crucial. /1/

Air as a contaminant may be introduced into hydraulic fluid through improper maintenance or as a result of system design. Besides more elastic response during system operation, the presence of air in a hydraulic system causes oil deterioration and degradation of lubrication, cavitation, erosion and noise generation. However, air elimination from a hydraulic fluid when the hydraulic circuit is in operation is a difficult technical problem.

On the other hand, particle contamination accelerates wear of hydraulic components. The rate at which damage occurs is dependent on the internal clearance of the components within the system, the size and quantity of particles present in the fluid, and system pressure.

Due to importance of mentioned problematic, it is necessary to know, how these contaminants move through the reservoir, and whether they are eliminated. Very helpful tool for predicting dynamical behaviour of contaminants inside reservoir is CFD simulation based on appropriate simulation model.

# 2. Simulation

Simulations of oil flow, containing air bubbles and solid particles, were performed using Ansys Workbench and were based on industrial 400 litre tank built according to "AB Normen Rexroth" (DN 400) /2/, with inner dimensions of 1492 x 712 x 390 mm as shown in **Figure 1**.



Figure 1: Simulations based on 400 L industrial tank

# 2.1. Simulation Model

A model of hydraulic fluid inside the tank was developed in SolidWorks:

As shown on Figure 2 there were three different simulation models:

- 1<sup>st</sup> model: plain vertical return line with two horizontal suction lines placed diagonally on the other side
- 2<sup>nd</sup> model: a longitudinal baffle was placed in the middle of the reservoir. The model uses the same plain vertical return line and two horizontal suction lines.
- 3<sup>rd</sup> model: vertical return line and two horizontal suction lines were placed the same as by the second model. A diffuser was placed on the return line, just below the oil surface, to help stabilize and direct the oil flow. To allow proper mixing of oil, the baffle was modified to only pass oil at bottom area (Figure 2).

As this is industrial practice, all return and suction pipes were cut at 45° angle.

To simplify the 3D model of the fluid inside the reservoir, all inactive hydraulic pipes that exist in actual, above described, were removed. The results of our previous

research (simulation of oil flow patterns inside the reservoir /3/) indicate, that the inactive tubes do not represent notable obstacles that would significantly change our simulation results.





In our previous work /3/ it was also discovered, that at given flow conditions, (described later in 2.2), oil surface may be considered completely horizontal with no level drop from return to suction line. In order to simply the model, the air above the oil surface was also neglected and simulation was made with degassing outlet condition at the top (instead of free surface flow).

# 2.1.1. Mesh

Surface and volume mesh were automatically created using Ansys CFX-Mesh with regard to additional settings. The mesh was refined in area of return and suction tubes in order to obtain more realistic simulation results. Since near solid wall's boundary layers affect velocity gradients, five inflation layers were created around the tubing. Meshing results are presented in **Table 1** and shown in **Figure 3**.



Figure 3: 2nd model – generated mesh

Table 1	Nodes	Elements
1 <sup>st</sup> Model	210498	630869
2 <sup>nd</sup> Model	254029	766012
3 <sup>rd</sup> Model	275710	821144

# 2.2. Simulation setup

The multiphase simulation of the research involves three homogenous materials: mineral oil (ISO VG 46), air (bubbles) and particles at constant temperature of 50 °C. The temperature influence and thus change of fluid properties due to change in temperature, were not calculated in this simulation.

At first, a steady state simulation was performed on all models. In order to achieve better convergence of the system, the steady state simulation was followed by the transient simulation with total time of 60 s in timesteps of 0,1 s.

## 2.2.1. Fluid and particle models

Mineral oil inside the fluid tank (ISO VG 46 grade) was modeled to be main continuous phase with molar mass evaluated at about 380 kg/kmol. /4/

Since simulation setup neglects the temperature effects, the density of oil was assumed to be constant value of 850 kg/m<sup>3</sup>. The viscosity at the given temperature is also constant and was evaluated to be 30 cSt (equivalent to ISO VG 46 viscosity at 50  $^{\circ}$ C).

One of the important parameters when simulating air bubble flow and small solid particle flow in continuous viscous fluid is the surface tension coefficient. The value was found in literature and was set to  $23 \ 10^{-3} \text{ N/m}$ . /5/

Air was modeled as dispersed fluid with three different specified mean diameters of 20, 100 and 500  $\mu$ m. In viscous fluid bigger air bubbles tend to rise more quickly since their lift force (minus the viscous drag force) is grater in comparison to lift force of smaller bubbles.

Most common solid particles found in used hydraulic oil are copper particles which are also used in our simulation (copper density is approx. 8940 kg/m<sup>3</sup> and is higher than of steel). Similar as air bubble, bigger copper particles are assumed to better resist the viscous fluid flow and fall more quickly than smaller ones.

#### 2.2.2. Fluid interphase drag

For low Mach number flows, the drag exerted on an immersed body by a moving fluid arises from two mechanisms only. The first is due to the viscous surface shear stress, so called skin friction. The second is due to the pressure distribution around the body, and is called the form drag. The total drag force is most conveniently expressed in terms of the dimensionless drag coefficient C<sub>D</sub>. The function may be determined experimentally, and is known as the drag curve. ANSYS CFX offers several different models for the drag curve, and also allows you to specify the drag coefficients directly.

Interphase drag between mineral oil and solid particles is modelled as Schiller Naumann drag model, where drag coefficient  $C_D$  equals /6/:

$$C_D = \frac{24}{Re} (1 + 0.15Re^{0.687}) \tag{1}$$

CFX modifies this to ensure the correct limiting behaviour in the inertial regime by taking:

$$C_D = max \left(\frac{24}{Re} (1+0.15Re^{0.687}), 0.44\right)$$
(2)

In this research it is supposed that the bubbles, in the dispersed phase, are single sized bubbles while break-up as well as coalescence are neglected. At sufficiently small particle Reynolds numbers (the viscous regime) fluid particles behave in the same manner as solid spherical particles. Hence the drag coefficient is well approximated by the Schiller-Naumann correlation described above. At larger particle Reynolds numbers, the inertial or distorted particle regime, surface tension effects become important. The fluid particles become, at first, approximately ellipsoidal in shape, and finally, spherical cap shaped. In this manner the Gracedrag-model is used, where drag coefficient  $C_D$  of a single bubble equals:

$$C_D(ellipse) = \frac{4}{3} \frac{gd}{u_T^2} \frac{\Delta\rho}{\rho_c}$$
(3)

where the terminal velocity  $U_T$  is given by:

$$U_T = \frac{\mu_c}{\rho_c d_p} M^{-0,149} (J - 0,857)$$
(4)

where *M* is Morton's number which can be further described in literature. /6/

For high bubble volume fractions, the Grace-model drag coefficient  $C_D$  may be modified using a simple power law correction:

$$C_D = r_c^p C_{D\infty} \tag{5}$$

Where  $C_{D^{\infty}}$  is the single bubble Grace drag coefficient and *p* is the volume fraction correction exponent which has negative value for small bubbles since they tend to rise more slowly at high void fraction, due to an increase in the effective mixture viscosity.

#### 2.2.3. Calculation model

The flow of main continuous phase (mineral oil) and the dispersed phase (air) was calculated with Eulerian-Eulerian model (with SST turbulence model), which is one of the two main multiphase models that has been implemented in Ansys CFX. The other one, which is the Lagrangian Particle Tracking Model, was used to calculate particle tracks within the main continuous phase.

#### 2.2.4. Boundary conditions

There are two pumps sucking 42 L/min of oil at the suction pipes, which cross-sections were defined as the system outlet. Each of the outlets was defined to have bulk mass flow rate of 0,60 kg/s.

The return pipe area represents system inlet and was defined to have the same bulk mass flow rate as the sum of the pump flows, that is 1,20 kg/s. The flow flowing into domain was defined to consist of:

- 94 % mineral oil volume fraction,
- 2 % air volume fraction with specified bubble diameter of 500  $\mu m,$
- 2 % air volume fraction with specified bubble diameter 100  $\mu$ m,
- 2 % air volume fraction with specified bubble diameter 20  $\mu$ m,
- 4 different sized groups of copper particles with specified diameter of 5, 25, 125 and 500 μm.

Although the particles of sizes 125 µm and larger are not likely to appear in hydraulic reservoir, they were also simulated to show what would happen in worst case scenario.

To simulate air bubble extraction from the surface, degassing outlet boundary condition was set at the top of the fluid domain, to enable air bubbles to escape from the domain.

# 3. Results

Different steady-state and transient simulation results were studied. Such simulation results are difficult to be presented on a static picture. Nevertheless, **Figure 4** shows horizontal plane at height of suction lines (100 mm from bottom), colored in the value of Air Volume Fraction from 0 (blue) to 1,00 % (red). It can be clearly seen from the figure, that the 1<sup>st</sup> model presented the worst case – it has extracted the least amount of air bubbles (of 500  $\mu$ m) from the oil. The 2<sup>nd</sup> model with the plain return line and the longitudinal baffle is much better solution, where much less air has gone to the second chamber of the reservoir. The best results were obtained by the third model, which uses diffuser on the return line together with the modified baffle.

And at the end, if we look at the figures – in comparison to  $1^{st}$  model there is approx. 4times less air (of 500 µm) being sucked in  $2^{nd}$  model. And further on, if we compare  $3^{rd}$ and  $2^{nd}$  model, the  $3^{rd}$  model is sucking even half of that in  $2^{nd}$  one.

Similar, a shade worse results were obtained by air bubbles in 100 and 20 µm size. This phenomena is due to smaller air bubbles rising more slowly since they are experiencing less lift force and more horizontal drag force from oil flow.

**Figure 5** shows copper particle position of sizes 5, 25, 125 and 500  $\mu$ m. Again, the worse performance can be seen on the 1<sup>st</sup> model where the particles of all sizes are very distributed inside the hydraulic tank. The exception are larger particles of size 125 and 500  $\mu$ m (which are very unlikely to be found in a reservoir) which tend to deposit at bottom around the return line at a certain circle. 1<sup>st</sup> model also shows a dead zone at bottom (in figure - upper right corner) where most of the particles are accumulated.

The performance of the second model is a little bit better, since there are fewer particles found near the suction lines. It can also be seen from the figure that the most of the larger particles will deposit in the first chamber.

Again, the best solution is represented by the  $3^{rd}$  model, which collected the most of the particles in the two steady areas of oil flow – the larger particles are accumulated just near the diffuser, and the rest of the particles are deposited in first chamber or in second chamber near the baffle (dead zone).



Figure 4: Air bubbles (500  $\mu$ m) Volume Fraction on horizontal plane at height of 100 mm, which is suction line height

# 1<sup>st</sup> model



2<sup>nd</sup> model



3<sup>rd</sup> model



Figure 5: Copper particle position on the bottom of reservoir: 500  $\mu$ m – orange, 125  $\mu$ m – purple, 25  $\mu$ m – green, 5  $\mu$ m – yellow

# 4. Conclusion

Although this research work neglects some effects in the hydraulic reservoir (temperature, free surface flow, bubble break-up,...) it gave us valuable information on what is happening inside the hydraulic tank.

One of the most interesting facts which have been revealed during the research is that the particles of size 5  $\mu$ m and 25  $\mu$ m (commonly found in hydraulic mineral oil) are very unlikely to deposit during the oil flow through the reservoir. Their mass and thus gravitational force is just too small to bring them to the bottom. Because oil viscous drag force is relatively much bigger than the gravitational force, the smaller particles tend to completely follow oil streamlines.

Similar results were obtained by simulating the air bubble extraction. It has been revealed that the smaller air bubbles of sizes 100  $\mu$ m and less are harder to extract, since they are rising much slower in viscous fluid.

# 5. Acknowledgement



Operation part financed by the European Union, European Social Fund. Operation implemented in the framework of the Operational Programme for Human Resources Development for the Period 2007-2013, Priority axis 1: Promoting entrepreneurship and adaptability, Main type of activity 1.1.: Experts and researchers for competitive enterprises.

## 6. Bibliographical References

- /1/ Doddannavar R., Barnard A.: Practical Hydraulic Systems, Elsevier Science & Technology Books, 2005
- /2/ N.N., AB Normen Rexroth

- /3/ Tic V., Lovrec D.: Design of Modern Hydraulic Tank Using Fluid Flow Simulation, 7th International Fluid Power Conference, 2010, Aachen, Germany
- /4/ Neale, M. J., Tribology Handbook, 2<sup>nd</sup> edition, Butterworth-Heinemann, 1995, Great Britain
- /5/ George E. Totten: Handbook of Hydraulic Fluid Technology, Union Carbide Corporation, 2000, Tarrytown, New York
- /6/ N.N., Ansys 13.0 Help, 2010

# 7. Symbols

C <sub>D</sub>	Drag coefficient	-
Re	Reynolds number	-
d	bubble diameter	mm
g	Gravitational force	Ν
Δρ	Reynolds averaged density difference between the phases	kg/m³
ρ <sub>c</sub>	Reynolds averaged density of continuous phase	kg/m³
Uτ	Terminal velocity	m/s
$\mu_c$	Dynamic viscosity of continuous phase	Pa s
rc	Volume fraction of continuous phase	-
р	Volume fraction correction exponent	-