Intermittent CFD simulation of interlocked hydraulic pumps - industrial use, basic conditions and prospect

Dr.-Ing. Michael Moczala

TRW Automotive GmbH, Düsseldorf, Germany

Summary:

The modern commercial software solutions in CFD provide tools capable of modeling the flow effects inside of hydraulic gear pumps. However, for a useful numerical model some important geometrical simplifications are still necessary. In a real pump existing cavities between the meshing teeth and between the teeth and some not moving parts close transiently. For this important event, reliable and standardized numerical mechanisms are not available at the moment. Moreover, in case of helical gears, for instance, the deforming mesh approach is not capable to handle the complex 3D geometry. Also in the 2D case a stable setup requires high work effort during the mesh generation and the setup of the dynamic mesh parameters.

In general for this kind of application, the number of computational cells is very high and the deforming mesh algorithms are computationally intensive. Moreover, the handling of transient result data is also memory and time extensive. The usability of the results is due to the necessary simplifications in the model of low engineering significance.

Therefore, such approach can not be recommended nowadays for industrial applications, where the CFD expertise is implemented in a design process and where optimization loops have to be performed. In this case more efficient 1D analytical algorithms covered next to the transient flow effects also the mechanical phenomena should be used. In this case, the CFD methods are besides the measurement a powerful tool for parameterization and verification of the analytical model.

Keywords:

Computational Fluid Dynamics (CFD), Moving and Deforming Grids, Hydraulic Steering, Gear Pump

1 Introduction

Geared pumps are commonly used to feed the hydraulic fluid in automotive power steering systems. The Fig. 1 shows schematically an example of such apparatus. The pump is hydraulically strongly coupled with other components and influences the dynamic system behavior significantly. Due to a continuous development in vehicle manufacturing the requirements on the modern steering also rise steadily. Low noise emission is in this case as essential as high system stability and efficiency. Detailed knowledge and understanding of the physical effects and interactions inside the pump becomes to a key factor. On the other hand, the modern software and hardware solutions in Computational Fluid Dynamics are exerted to offer approaches to model the flow driven by the gear rotation in the pump.

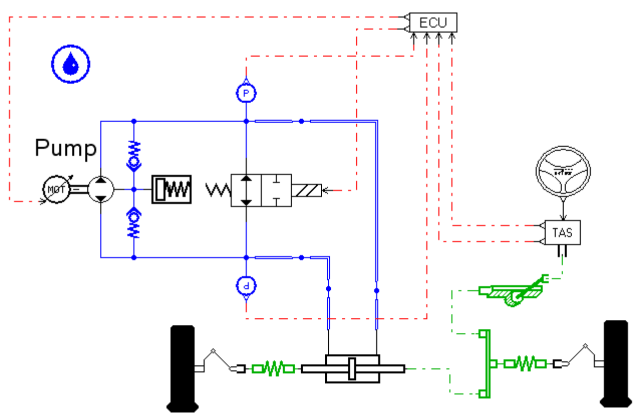


Fig. 1: Schematic diagram of an Electro Hydraulic Steering system

The aim of the present work was to asses the applicability of modern flow simulation software to answer questions related to gear pump engineering. Adequate solver parameters as well as essential geometrical simplifications, which afford a robust and efficient simulation run, have been researched on a 2D example. Some measurement results have been used to verify the numerical prediction and to evaluate the influence of the conducted assumptions. Restrictions and usability of a 3D simulation have been also investigated.

2 Investigated Geometry

Due to different operating principles two different types of hydraulic gear pumps have been considered: the external and the internal gear pump – see Fig. 2.

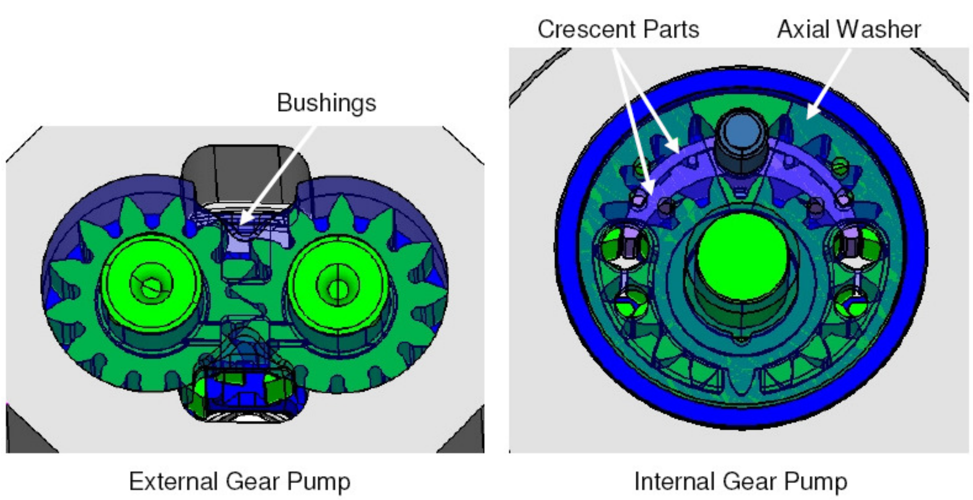


Fig. 2: Considered pump types

The external gear pump consists of two gears rotating in opposite sense with the same angular velocity. The driven gear is the so-called master. The slave gear is driven by the master due to the

tooth interaction. The circumferential sealing between the high and low pressure chambers results from the contact between the meshing gears and between the teeth tips and the pump housing. The housing contact is caused by the displacement of the gear axes due to different pressure levels in both chambers. In the present study the external gear pump has been investigated in 2D considering the transient gear rotation.
In the internal gear pump, the central placed pinion receives the torque from the motor and drives the geared ring. Both gears rotate in the same sense with different angular velocities. The circumferential sealing results from the adjusting of the gears and the crescent parts due to the pressure buildup. In the present study only steady state simulations has been done on the 3D model of this pump.

3 Numerical Method

The main objective of the present work was to evaluate the capability of modern commercial CFD software packages to perform reliable unsteady simulations of geared pumps. For the entire simulation work the standard tools provided by Fluent has been deployed.

3.1 Mesh

The computational meshes have been generated using the software Gambit 2.2.30. An example of a 2D configuration is displayed in Fig. 3. Some parts of the mesh experience remeshing during the simulation. These zones have been discretized using triangular cells. The remaining parts of the domain have been filled with quadrilateral elements.

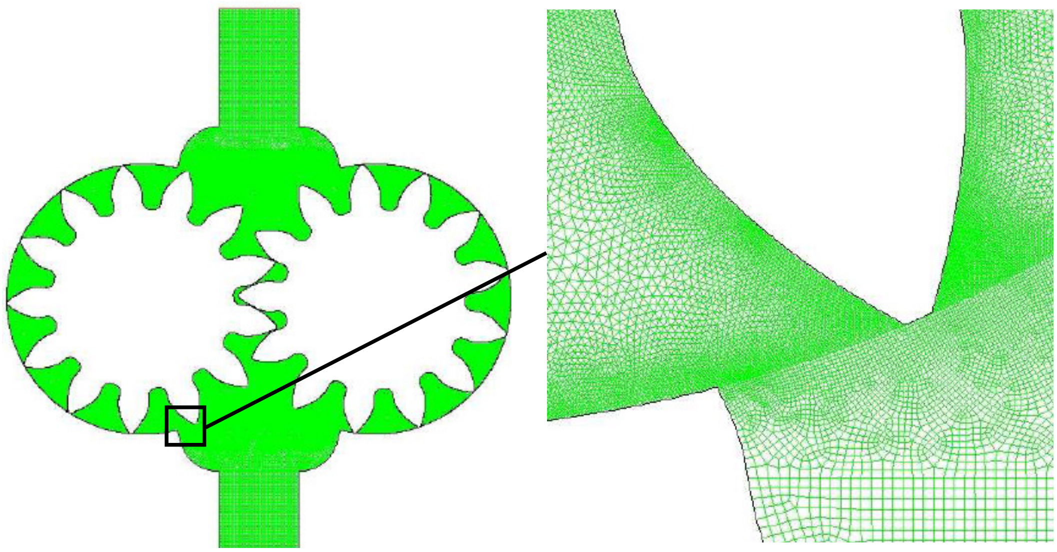


Fig. 3: Example of a 2D high density mesh

The size function parameters in the remeshed zone have to be chosen carefully. This zone contains the critical gaps between the gears and between the gears and the not moving walls. Size and distribution of cells in this region determine in connection with the remeshing parameters the time consumption and the stability of the simulation run.
The approximated mesh size for the unsteady 2D simulations was ca. 370 thousand cells. The steady 3D investigation has been performed on a mesh consisting of ca. two millions cells.

3.2 Solver and Boundary Conditions

For the flow simulation the version 6.2.16 of Fluent has been deployed. The unsteady and steady segregated (pressure based) solver with second order upwind discretization for pressure, momentum and turbulent quantities has been used. The pressure and velocity fields have been coupled under application of the PISO algorithm. In case of the moving mesh approach only the first order transient formulation is possible. Additionally to the default formulation also the Non-Iterative Time Advancement (NITA) has been examined. The standard k-ε model with Enhanced Wall Treatment has been used to reproduce the turbulent effects. This approach has shown the most stable convergence progression. Also due to stability reasons it has been resign to introduce the cavitation model. A Newtonian fluid with constant density and viscosity has been assumed.
At the inlet of the computational domain the total pressure and at the outlet the static pressure has been defined as boundary condition. At all solid walls the no-slip condition for viscous flows has been

imposed. The transient movement of the gear walls has been modeled using an appropriate User Defined Function (UDF) which sets the rotational speed of both wheels.
In the 3D case additionally the interface boundary condition between the zone connected to the gears and the remaining volume parts has been used. Also in the 3D configuration the moving wall boundary condition has been imposed on the appropriate surfaces.

4 2D Approach

The 2D models do not consider, for example, the important influence of the relief groves placed in the bushings of the external gear pump. In the internal gear pump, appropriate shaping of the axial washers provides the similar functionality. On the other hand, in case of application of the deforming mesh scheme for modeling of gear rotation, the capabilities of the numerical procedure are extremely exhausted. Therefore, a simplified 2D study is the essential first step for assessment of this numerical approach.
Due to the mentioned geometrical simplifications the presented 2D results do not reflect the true flow behavior in the pump. However, the analysis of the plausibility of these results is necessary for validation of the used numerical method.

4.1 Preliminary Studies

Divers questions concerning for instance the modeling of closing gaps or possibilities for speeding up of the transient simulation had raised before the presented investigation started. For some evidence several preliminary studies have been therefore performed.

4.1.1 Gap Size and its Spatial Discretization

Due to the torque exchange between the master and the slave gear there is a contact between the acting teeth. In case of the external gear pump the teeth tips come also in contact with the housing in the region near to the low pressure chamber. The pressure difference between the inlet and the outlet chamber imposes here some displacement of the gear axes. In case of the internal gear pump the teeth tips contact the crescent parts which moves also due to the pressure difference between the inlet and the outlet chamber.
Thus, in a real pump the gaps close transiently, but in the used CFD solver standardized numerical mechanisms are not available for modeling of this event. However, to obtain a simple numerical model a minimal gap size has been assumed. The clearance related to the teeth tips (tip clearance) has been maintained by an adequate definition of the meshed geometry – housing dimensions, position of the crescent parts and gear tip diameter. The minimal size of the gap between the meshing teeth (meshing clearance) depends on the initial angular position of the gears. As a consequence at places of mechanical contact a thin fluid layer separates the appropriate solid pars.
Therefore, as a preliminary study the influence of the gap size as well as of the minimal cell number over the high of the clearance has been performed. It has been found out, that the predicted time averaged volumetric efficiency of the pump is a good indication for this influence.

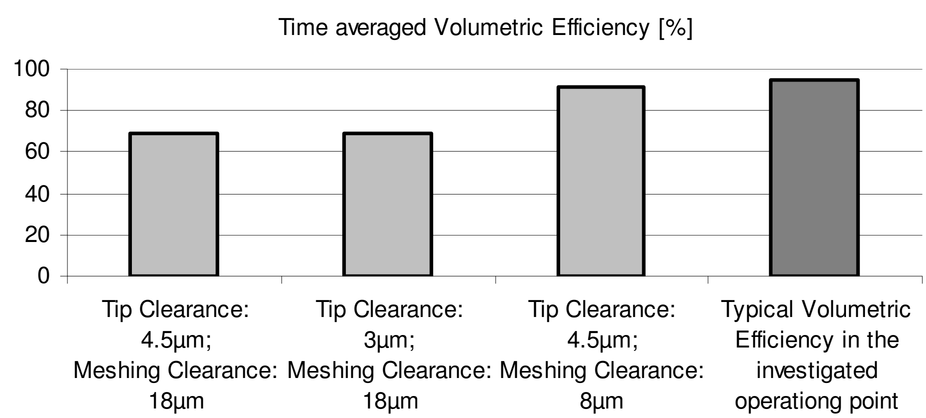


Fig. 4: Influence of the clearance size on the predicted time averaged Volumetric Efficiency

Several simulations have been carried out for different clearances. The diagram in Fig. 4 shows, that the size of the meshing clearance has a fundamental influence on the predicted volumetric efficiency. The considered gap sizes of 4.5µm for the tip clearance and 8µm for the meshing clearance represent the smallest values allowing a stable simulation run. The tip clearance of 3µm is in this regards a borderline value and its application do not improve the results significantly. Therefore, the combination of 4.5µm for the tip clearance and 8µm for the meshing clearance has been chosen for all following studies.

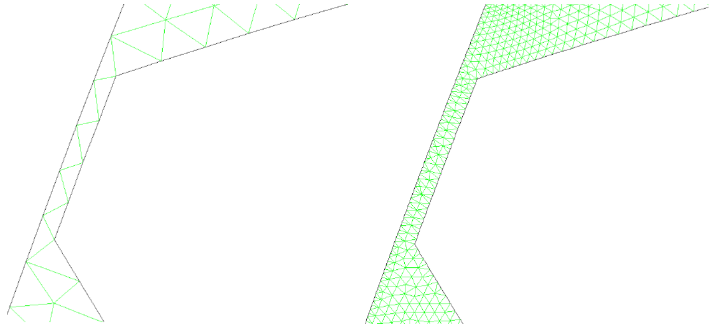


Fig. 5: Detail of the tip clearance with a coarse (one cell over the height) and a fine mesh

The investigation mentioned above has been performed for only one cell over the high of the clearance. However, the effect of the mesh density in the tip gap, see Fig. 5, has been also assessed. The outcome of this study was that the smaller the gap is the lower is also the influence of the cell number inside. Thus, meshing of the small clearance of 4.5µm with more then one cell over the height produces extremely high number of elements in the entire domain, but the improvement of the accuracy is insignificant. Therefore, for all further studies all the small transient gaps have been discretized with at least one cell over the gap height.

4.1.2 Remeshing Parameters

The experience with the dynamic mesh approach shows, that the remeshing parameters influence not only the convergence behavior but also the CPU effort of a simulation run. In case of improper parameter choice, the number of cells in the computational domain accumulates while the calculation advances. As a consequence, the CPU time needed for one time step increases steadily as the simulation proceeds. An example of an initial mesh and two resulting meshes after a test run is demonstrated in Fig. 6. To check the interaction of the mesh and the remeshing factors quickly, it is recommended to perform the test runs without solving of the flow related equations.

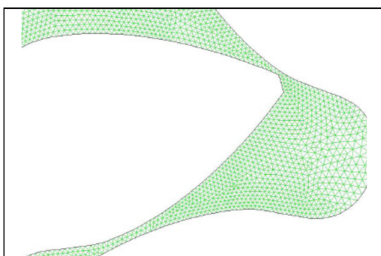
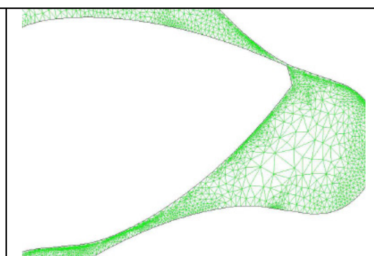
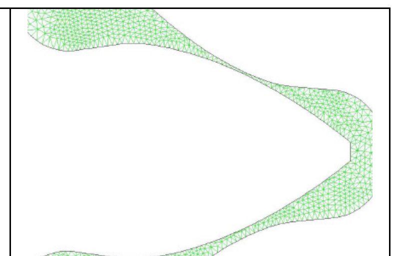
<table>
<tr><td></td><td></td><td></td></tr>
<tr><td rowspan="2">Initial mesh</td><td>Improper parameters</td><td>Well-defined parameters</td></tr>
<tr><td colspan="2">Mesh after a test run</td></tr>
</table>

Fig. 6: Influence of the remeshing parameters of the mesh evolution

The improper parameters drive the mesh to lose its quality along the simulation and broadly differences in the cell density develop.
The parameters which results in combination with the used mesh in satisfactory remeshing procedure are stored in Tab. 1. The method used for the mesh regeneration was “remeshing”. Additional enabling of “smoothing” option increases the calculation time significantly and do not improve the mesh quality in the investigated application. The surfaces of both gears have been defined as rigid bodies and the appropriate cell height has been set to $6 \cdot 10^{-5}$ m.

Tab. 1: Optimized dynamic mesh parameters

Must improve skewness	Maximum length scale	Minimum length scale	Maximum cell skewness	Size remesh interval
enabled	$1.6 \cdot 10^{-4}$ m	$3.2 \cdot 10^{-6}$ m	0.4	1

4.1.3 Time Discretization

The transient simulations have been performed using the fixed time stepping method. To comply with the applied remeshing procedure the time step size has to be related to the cell width. The general experience reveals that for a stable run the rotational displacement of the teeth tips during one time step has to be approximately equal to the edge length of the cells at the housing surface of the critical region. The time step size estimated in this way is very small, but contrariwise, the number of the sub-iterations necessary in this case for reaching a sufficient convergence was always lower then five.
Because of the small time step size and the computationally intensive dynamic mesh routine, the simulation runs are very time-consuming. A remarkable improvement provides here the so called Non-Iterative Time Advancement (NITA) which significantly reduces the time required to run the simulation. Application of NITA reduces in the investigated cases the simulation time by more then half compared to the standard iterative time advancement.
The change of the time advancement rearranges the solution method of the temporal discretized flow equations. To revise the influence of this modification on the simulation results, the standard simulation run and the run using NITA have been compared. Similar to the assessment of the gap size the volumetric efficiency shows also the strongest sensitivity on the change of the time advancement – see Fig. 7. The plot reveals that the highest amplitude difference is in the order of magnitude of less then 2% and the effect is therefore small. The high frequency fluctuations, which can be observed in the diagram, are created by the remeshing procedure.

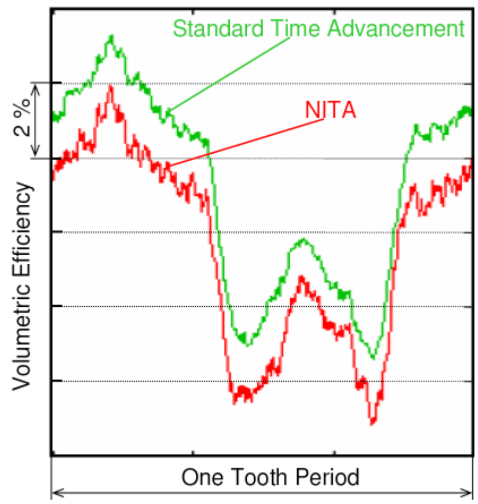


Fig. 7: Comparison of standard time advancement and NITA

4.2 Additional Results

Two effects caused by the flow phenomena are of particular interest in terms of the dynamic pump performance. The flow pulsation [1] influences the dynamic behavior of the entire hydraulic system. The squeezing and cavitation effects influence directly the solid components of the pump.

4.2.1 Flow Pulsation

Interlocked pumps do not produce ideal constant flow rate, but they generate the so called flow ripple. The frequency of this flow pulsation depends on the rotational speed of the pump actuation and on the teeth number. The transient flow generates on the resistances existing in the hydraulic system pressure pulsation. These pressure waves propagate in the entire system and affect its stability and acoustical behavior.
A direct measurement of the flow rate variation provoked by the pump is in general very difficult. On the opposite side, measurements of pressure pulsation in the hydraulic system are more convenient. Thus, the pressure pulsation measured in an appropriate test bench can be compared with a signal obtained from the CFD results.
A comparison of the simulated and measured pressure pulsations in the region just downstream of the pump is shown in Fig. 8. Both pressure signals show qualitatively similar characteristics. In general the amplitude of the signal obtained from the simulation is slightly higher. The measured pressure fluctuation contains some components of high frequency, while the signal from the simulation shows only the first and second order of the pump pulsation. A FFT analysis of both signals shows this behavior more clearly in the upper part of the diagram.

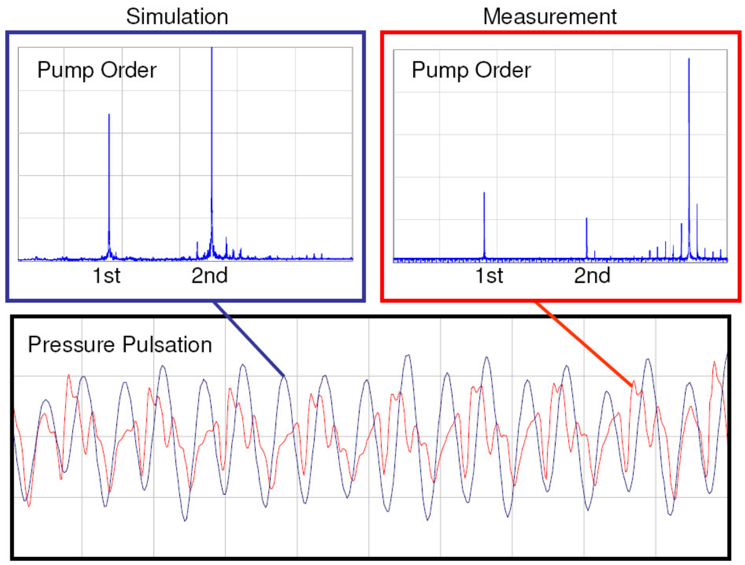


Fig. 8: Frequency analysis results and the related pressure signals

4.2.2 Squeezing and Cavitation Effects

Two trapped volumes arise between the teeth during the gear meshing. The Fig. 9 shows as example pressure distribution in the pump at on specific time step. The closed regions are also labeled here.
One of the regions experiences volume reduction and the trapped fluid is squeezed. High pressure builds up in a very short time and as a result, the gear shafts and bushings experience a load pick.
The second volume expands and the fluid components degases and evaporates. Pressure levels of only few Pascal arise inside the expanded trapped volume. After this volume again opens, the previously formed cavitation bubbles collapse and strong pressure waves develop and the affected solid parts experience again a strong mechanical load.
In a real pump special relief grooves are designed in the lateral pump bushings. The grooves connect the trapped volumes to the respective pump chamber and minimize the strong pressure changes due to squeezing and expansion of the volumes.

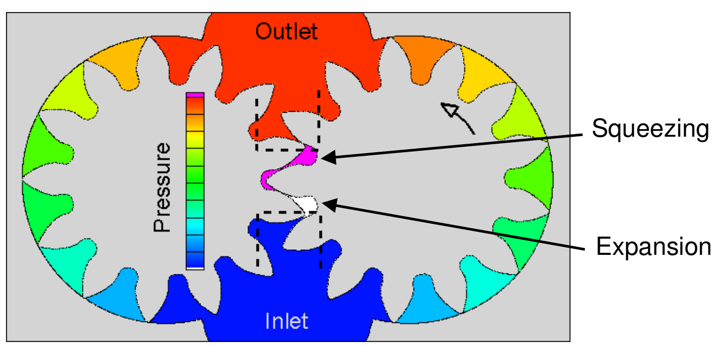


Fig. 9: Unsteady pressure distribution in the 2D case – momentary gear position

The simplified 2D consideration demonstrates the squeezing and expansion of the trapped volumes very clearly, but it does not account for the effect of the relief grooves.

5 3D Approach

Consideration of the 3D geometrical details of the pump design, like the relief grooves for example, is essential for proper prediction of the unsteady flow effects. However, the outcome of the preliminary 2D study on the deforming mesh approach reveals the high effort on computing power and preparation work for a single simulation run. The applied numerical methods as deforming mesh and NITA are useful, but in this particular application not sufficiently stable. Moreover, due to reduction of the cell number the remeshed volume has to be meshed using the Cooper scheme and the dynamic mesh approach is not capable to remesh quadrilateral faces over the cell height. Therefore, only spur gears can be rotated in the 3D case. Unsteady 3D simulations of geared pumps can not be nowadays recommended for industrial applications.

However, full 3D steady simulations can be useful deployed in the engineering process and also utilized for parameterization of analytical models. The Fig. 10 shows for example a steady-state prediction of pressure distribution inside the pump housing at one specific gear position. The results have been used for evaluation of the loads acting on the housing components. Forces and torques acting on the gears have been computed upon the predicted pressure distribution around the gears. Also the position and size of the relief grooves and holes has been optimized applying this approach. The steady simulation allows consideration of different clearance sizes. Also body contacts respectively closed gaps can be taken into account. The viscous flow effects in the clearances have been modeled properly under skilled application of the moving wall boundary condition.

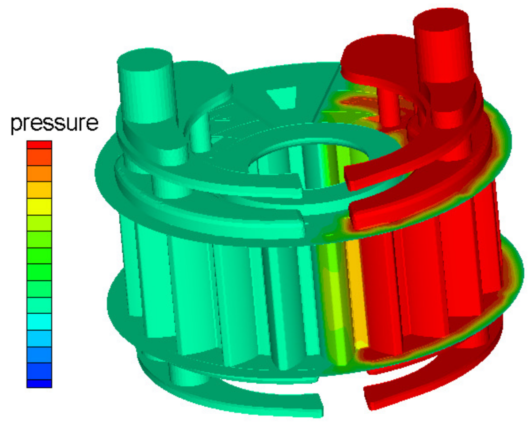


Fig. 10: Steady pressure distribution in the 3D case – exemplary gear position

6 Analytical Model

A different possibility to investigate the physical phenomena in geared pumps is deployment of analytical models. Such models are capable to consider the flow and the mechanical effects as well. Mathematical formulations necessary for the description of such fluid-structure-interaction are available already.

The main advantage of analytical models is they low CPU demand. In this case also extensive optimization runs under consideration of high number of variables can be performed quickly. The disadvantage is the high effort during the development and validation of the numerical formulations. Measurements and CFD simulations play at this place a very important role.

For the layout of external gear pumps an appropriate model has been developed already. It considers following effects: variable meshing stiffness, tooth profile errors, backlash between meshing teeth, lubricant squeeze, variable pressure distribution, hydrodynamic bearings and torsional stiffness and damping of the driving shaft. A detailed description of the model is contained in [3].

7 Conclusions

To allow further design optimization of gear pumps the availability of appropriate numerical models is essential. Current CFD approaches are based on some geometrical simplifications and its application is connected with high time effort. On the opposite side, the analytical methods are capable to provide comprehensive information about the modeled system and are more efficient in its application.

However, the CFD expertise plays besides the experimental methods an important role in the validation and parameterization of the analytical models. With the aid of numerical flow simulation it is possible to evaluate single effects such flow and pressure distribution in cavities, viscous forces and cavitation.

References

[1] Manring, N.D., Kasaragadda, S.B.: "The Theoretical Flow Ripple of an External Gear Pump", Transactions of the ASME Vol. 125, 2003, 396-404

[2] Inaguma, Y.: "Calculation of Theoretical Torque and Displacement in an Internal Gear Pump", JTEKT Engineering Journal English Edition No. 1001E, 2006, 70-76

[3] Mucchi, E.: "Dynamic analysis of external gear pumps by means of non linear models and experimental techniques", Thesis for the Degree of Doctor of Philosophy, 2007