CFD model of an external gear pump

Paulina Jędraszczyk and Wiesław Fiebig

Wrocław University of Technology

Abstract The paper presents a CFD model of an external gear pump. To investigate the fluid flow phenomena that occur in external gear pumps, one of the commercial CFD software packages was selected. The immersed solid method was used to simulate the flows induced by the rotation of gears. The article presents results of simulation studies carried out for various operating parameters of a chosen external gear pump. Prepared 2D CFD model was used to consider the influence of rotational speed and the pressure in the outlet channel on the cavitation intensity. Furthermore, the 3D simulation model was prepared to verify the pressure relief groves impact on the pressure distribution and velocity contours in the area of gears engagement, as well as mass flow rate and pressure pulsation on the outlet.

1 Introduction

Gear pumps belong to a group of positive displacement pumps. Their cost-effectiveness, simple construction and compactness makes them commonly used in both stationary and mobile hydraulic drives.

The volumetric efficiency of this pumps decrease because of fluid flow occurring in the axial gaps between sides of the gears and bearings surfaces as well as in radial gaps existing between the tooth heads and the casing [1-7]. Taking into account fluid volume changes in spaces between gear teeth, especially in the meshing zone, the cavitation phenomena can take place [4,7].

The design of gear pumps and motors is focused on more efficient units which is possible with advanced numerical simulation techniques. To investigate all of mentioned processes CFD model was prepared, which enables to study fluid flow phenomena during the pumps operation.

2 External gear pump 2D CFD model

The physical model chosen for the analysis is an external gear pump manufactured by Parker Hannifin Corporation. The PGP620 unit is made up of the three main elements: driving gear (active), driven gear (passive), housing.

Investigated pump operate at speeds from 500 to 3500 rpm and continuous pressures of up to 275 bar. Fig. 1 presents the geometrical model of the pump created using CATIA V5 software.

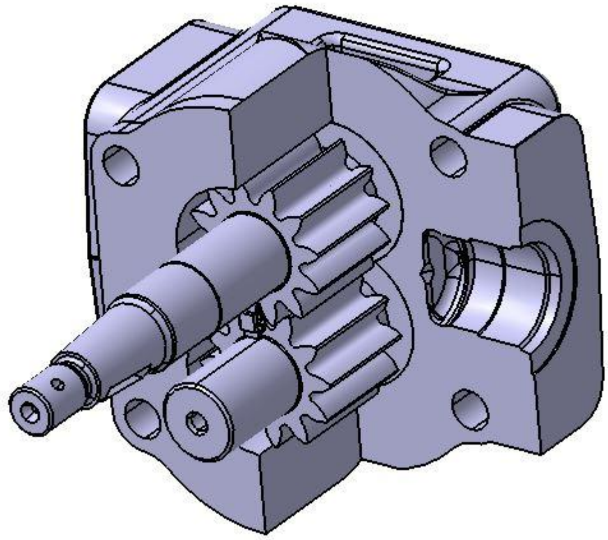

Fig. 1. 3D geometry model of an external gear pump created via CATIA V5

2.1 CFD model

Based on three-dimensional model geometry of the gear pump simplified two-dimensional model was created. Then it was discretized using hexahedral mesh with the thickness of one element. The mesh size was chosen to be sufficiently large to speed up calculations and small enough to visualize fluid flow phenomena that occur in the radial gap well as in meshing zone. For simulation the Immersed Solid Method was used, which is an option of ANSYS CFX program that allows simulation rigid motion of solid objects through fluid regions. Importantly, it avoids any mesh deformation or remeshing.

Fig. 2 shows two-dimensional model of the gear pump with selected domains and locations of boundary conditions. In order to obtain results of reasonable accuracy the simulation parameters were set according to the Table 1.

Hydraulic oil used as material for fluid domain has following properties:

- density – 950 kg/m^3
- kinematic viscosity – 37 mm^2/s.

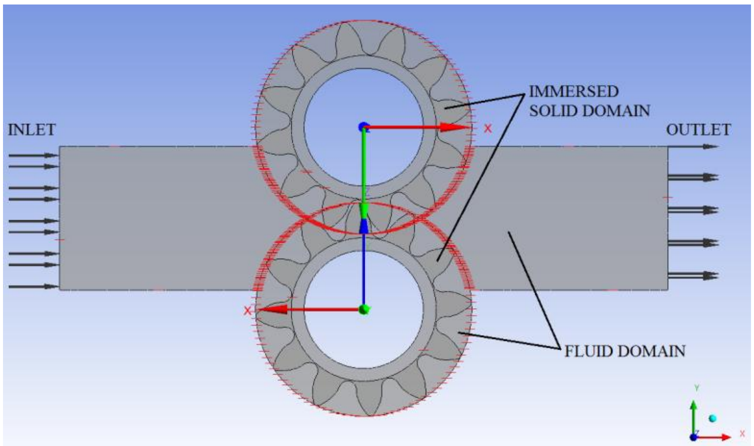

Fig. 2. Simplified 2D model of a PGP620 pump

Table 1. Computational setup for an analysis

Domains	
Fluid	stationary
Inlet	stationary
Outlet	stationary
Gears	rotation
Boundary conditions	
Inlet	pressure inlet
Outlet	pressure outlet
Housing	wall
Analysis	
Type	transient
Step time	10^-6s
Model	
Turbulence model	k-Epsilon

2.1 Comparison of experimental and simulation results

Results from simulation have been compared with experimental studies presented in [6] for defined positions of gears and equal operating parameters of the pump (rotational speed - 500 rev/min and discharge pressure - 0.55 MPa).

Fig. 3 shows velocity vectors plot from CFD simulation results and photo from high speed camera taken during the visualization investigations of the gear pump

[6]. In both cases strong disturbances in form of characteristic turbulence of the flow are visible. In this area, except of change of the flow direction, fluid velocity increase. At the same time the pressure in the turbulence zone decrease what is visible on the pressure distribution plot in Fig. 3a.

a) b)

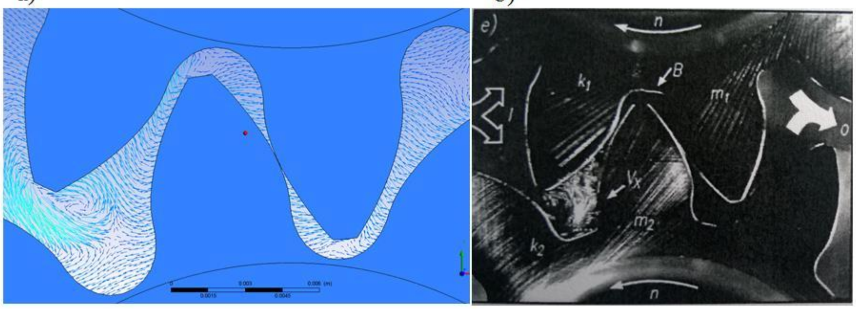

Fig. 3. Comparison of a) simulation research, b) visualization research

The solution obtained using CFD is in the agreement with results from experimental investigations. Therefore the CFD model can be used for further analysis of fluid flow phenomena in gear pumps.

3 Operating parameters and their influence on the flow

The fluid flow analysis was performed for different operating parameters of the pump such as gears rotational speed and discharge pressure while maintaining the same value of radial gap. All of the cases and sets of parameter considered during research are shown in Table 2. For each set of parameters numerical simulation was performed and then the pressure distribution and velocity vectors in the area of gears engagement were plotted. This enables the analysis of fluid flow processes that appear in this region as well as investigating the intensity of cavitation phenomenon.

Table 2. Analyzed set of parameters

Rotational speed	**Outlet pressure**	**Gap**
2000 rev/min	15 MPa	0.05 mm
	10 MPa	
	5 MPa	
1000 rev/min	5 MPa	
500 rev/min	5 Mpa	
	0,55 MPa	

3.1 The influence of rotational speed

Simulations for different rotational speed of gears, maintaining the discharge pressure equal 5 MPa, were performed. Pressure and velocity contours obtained for considered rotational speeds are shown in Fig. 4. In all cases the characteristic vortex appears, which intensity depends on the operating parameters of the pump. It can be noticed that with increasing rotational speed the fluid velocity increase leading to intensify observed vortex. At the same time, in the area of fluid flow disturbances negative pressure is present, what is shown in Fig. 5. Its value increase while increasing the gear rotational speed. For better visualization of the results the upper value for presented pressures was set to 0 Pa what enable to observe the whole range of negative pressure in this area.

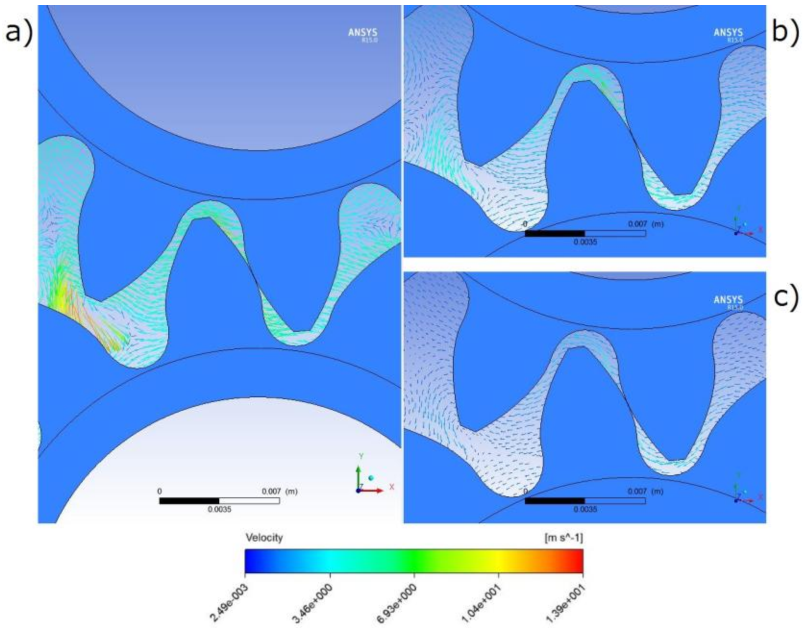

Fig. 4. Velocity vectors for different rotational speeds a) 2000 rev/min, b) 1000 rev/min, c) 500 rev/min

Taking into account results shown in Fig. 4 and 5 it can be concluded that cavitation phenomenon occurring in the zone of gear engagement increases with increasing of rotational speed of gears. This is due to the simultaneous increase in fluid velocity and local pressure drop.

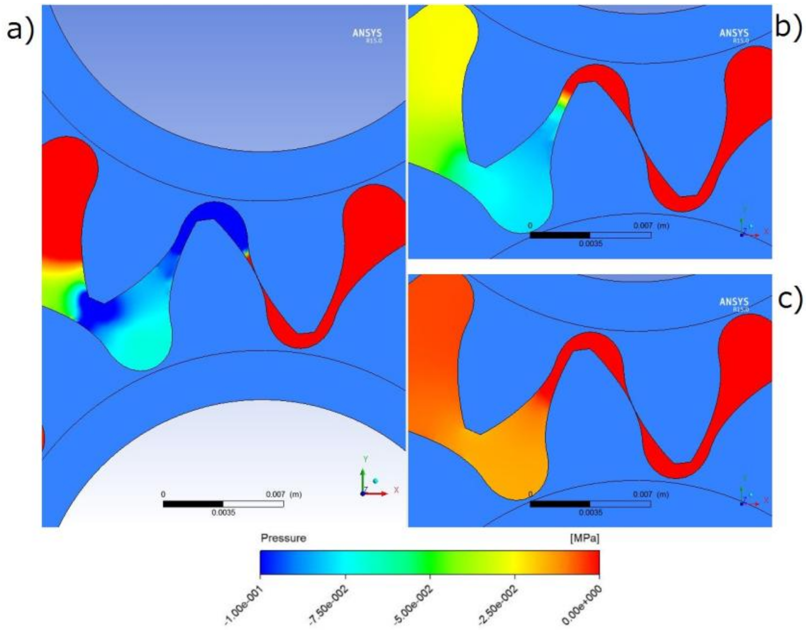

Fig. 5. Pressure distribution for different rotational speeds a) 2000rev/min, b) 1000 rev/min, c) 500 rev/min

3.2 The influence of discharge pressure

In the next step the influence of the pressure in the discharge channel on the flow in the gear meshing zone was investigated. Obtained results for discharge pressure equal 5 MPa, 10 MPa and 15MPa, in the form of pressure and velocity distribution contours, are shown in Fig. 6. It is visible, that the flow disturbances of the hydraulic fluid become less intensive while increasing the discharge pressure. Pressure distribution plots of the analyzed area shown in Fig. 7 confirm assumptions about decreasing cavitation intensity with higher discharge pressure.

The impact of the value of discharge pressure on fluid flow phenomena in the engagement area can observed on contours in Fig. 6 and 7. It has to be mentioned that in all considered cases both negative pressure values and velocities are lower than values observed for pump operating at different rotational speeds. This suggest less impact of the pressure in discharge channel on cavitation.

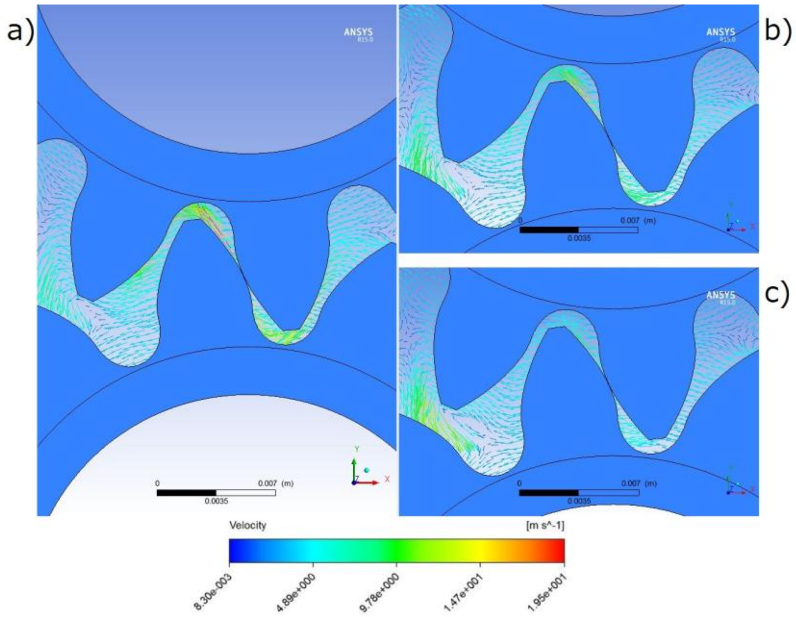

Fig. 6. Velocity vectors for different values of discharge pressure a) 15 MPa, b) 10 MPa, c) 5 MPa

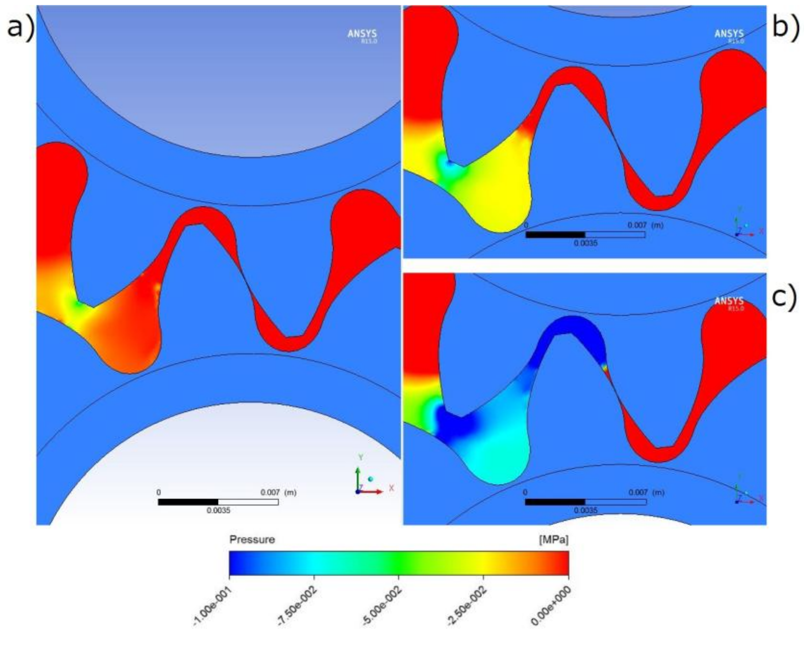

Fig. 7. Pressure distribution for different values of discharge pressure a) 15 MPa, b) 10 MPa, c) 5 MPa

4 External gear pump 3D CFD model

Two-dimensional CFD model presented in previous sections allows to investigate the influence of different pump operating parameters on cavitation intensity in the meshing zone. However, there are some other factors that affect the fluid flow in that area, for example pressure-relief grooves.

In order to investigate the influence of pressure-relief grooves on pressure and velocity distribution in the area of gears engagement 3D CFD model was prepared All boundary conditions and simulation parameters remain as in 2D model.

Simulations were performed for geometry of the pump with and without pressure-relief grooves.

Results from the 3D CFD model have been compared with experimental results [6]. Velocity vectors plotted for plane cutting the oil volume for geometry without pressure relief grooves were similar to the photo taken during the flow visualization [6]. Plane chosen for pressure and vector distribution contours analysis and volume rendering are shown in Fig. 8.

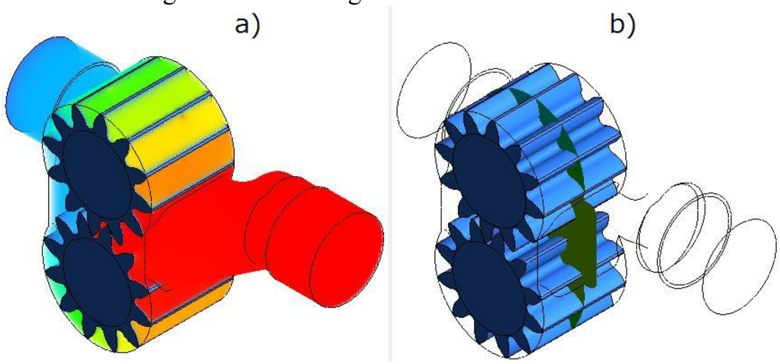

Fig. 8. Pressure distribution a) volume b) cutting plane

5 The influence of pressure relief grooves

Results obtained for discharge pressure of 5 MPa and rotational speed of gears equal 2000 rev/min are shown in Fig. 9 and Fig. 10. It is visible that pressures in the area of gears engagement for the gear pump without relief grooves are higher. Moreover, higher negative pressure values can be observed for the construction without pressure-relief grooves. The trapped volume between gears changes during the rotation of the gears. From the begin of the engagement of the second gear pair up to the central position of both gear pairs the trapped volume decrease what increases the pressure (Fig. 9). From the central position up to the end of the engagement of the previous gear pair the trapped volume increase. Due to changes of the trapped volume occurs the pressure overdue in the first phase and the under pressure in the second phase (Fig. 10) of the engagement area of two gear pairs.

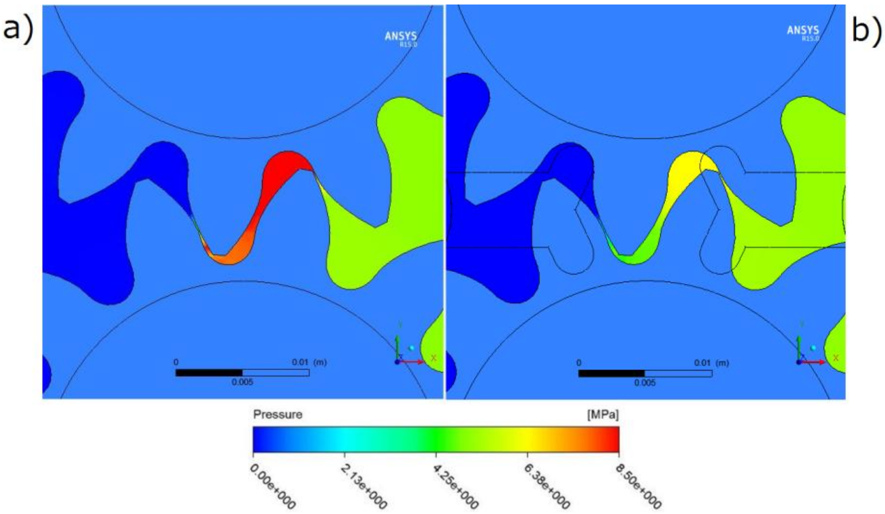

Fig. 9. Pressure contours for geometry a) without relief grooves, b) with relief grooves (p=5 MPa, n= 2000 rev/min)- trapped volume is decreasing

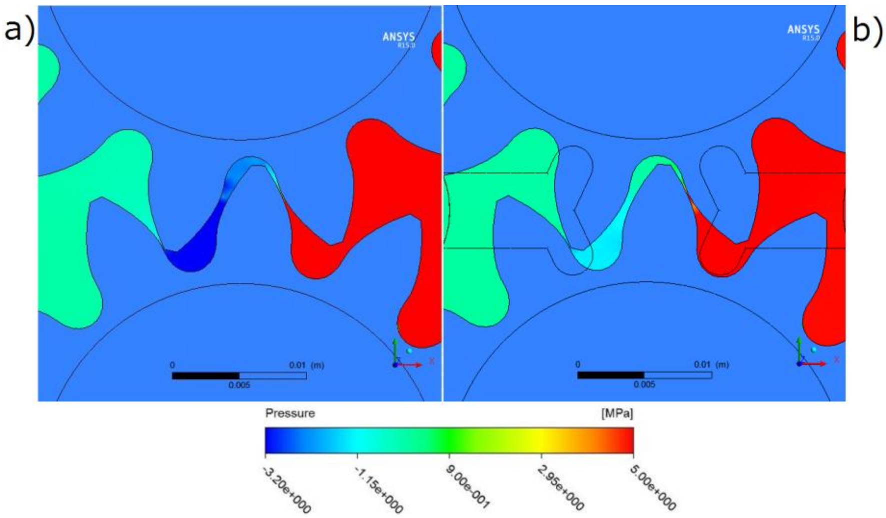

Fig. 10. Pressure contours for geometry a) without relief grooves, b) with relief grooves (p=5 MPa, n= 2000 rev/min)- trapped volume is increasing

Taking into account plots shown in Fig. 9 and Fig. 10 it can be concluded that pressure-relief grooves significantly reduce the high pressure and negative pressure occurring in trapped volume. It has to be mentioned that he pressure distribution slightly change according to the cutting plane due to asymmetric orientation of inlet and outlet channels.

6 Summary

The paper presents numerical simulation results for the fluid flow during the engagement of gears in an external gear pump. A comparison of results obtained by two different methods - traditional visualization method [6] and CFD technique – confirmed that the prepared CFD model reflects the basic phenomena that occurs during the fluid flow.

Simulations made for six different cases lead to the conclusion that operating parameters of the pump unit have a significant effect on the cavitation intensity. The resulting pressure distribution and vector plots in the area of gears engagement show that fluid flow turbulences and thereby the negative pressure increase with increasing rotational speed and decrease while reducing the outlet pressure. Furthermore, the comparison of the results reveals that changes in discharge pressure in the outlet channel affects the intensity of cavitation less than the rotational speed change.

3D CFD model allowed to investigate the impact of the pressure-relief grooves on pressure in the meshing zone. The pressure-relief grooves significantly reduce the high pressure and negative pressure occurring in trapped volume. In further investigations the pressure pulsation of the gear pump as well as the mass flow rate for different operating conditions and different geometry of the pressure relieve grooves will be analyzed.

References

[1] Ertürk N., Vernet A., Castilla R., Gamez-Montero P.J., Ferre J.A. (2011) Experimental analysis of the flow dynamics in the suction chamber of an external gear pump, International Journal of Mechanical Sciences, 02/2011

[2] Flores-Márquez A., Velázquez-Villegas F., Ascanio G.(2013) Numerical analysis of hydraulic gear micro motor, Journal of Mechanical Science and Technology 27 (5) (2013)

[3] Fiebig W. (2008) Schwingungs- und Geraeuschminderung in hydraulischen Systemen, VDI Fortschritt Berichte, No. 337, 2008

[4] Houzeaux G., Codina R. (2007) A finite element method for the solution of rotary pumps, Computers & Fluids, 36 (2007)

[5] Hyun K., Hazel M., Suresh P. (2007) Two-dimensional CFD analysis of a hydraulic gear pump, American Society for Engineering Education,

[6] Mochala M.,Intermittent (2009) CFD simulation of interlocked hydraulic pumps industrial use, basic conditions and prospect, FLUIDON Konferenz 2009 "Simulation im mechanischen Umfeld"

[7] Stryczek J. et al. (2014) Visualisation research of the flow processes in the outlet chamber - outlet bridge - inlet chamber zone of the gear pumps, Archives of Civil and Mechanical Engineering (2014), http://dx.doi.org/10.1016/j.ACME 2014.02.010