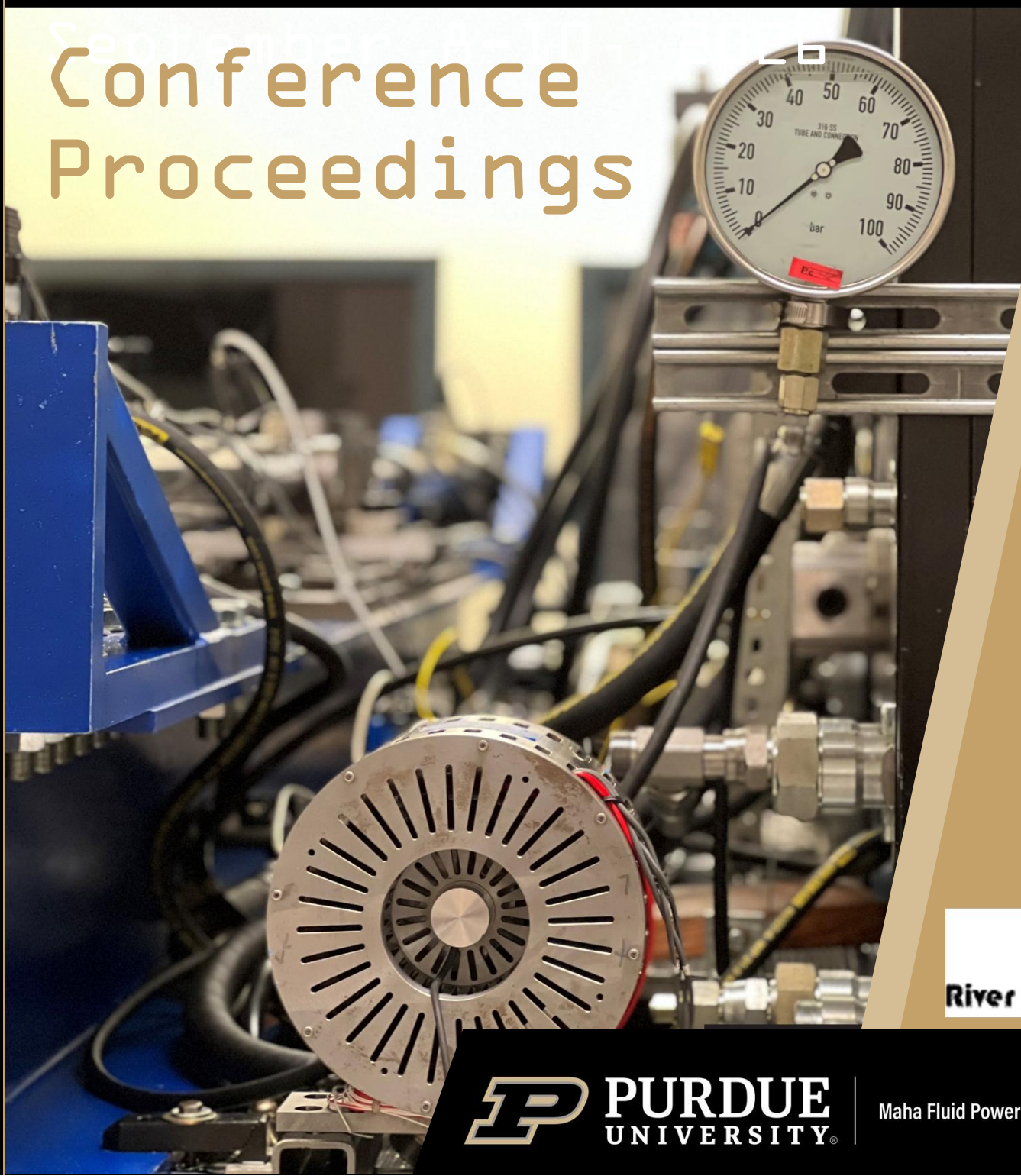


2026 International Maha Fluid Power Conference

West Lafayette, IN, USA

Conference
Proceedings



2026 International Maha Fluid Power Conference Proceedings

Table of contents

Title	Page No.
Using Nondimensional Analysis to Design and Model Hydraulic Control Systems Noah Manring	1
How Fluid Power Research Can Contribute to a More Sustainable Construction Industry Olivier Reinertz, Christian Bertram, Katharina Schmitz	10
Novel Poppet Valve for Digital Hydraulic Transformer Elton Bishop	25
Unsteady CFD Analysis of Alternating Flow Patterns in a Restrictive Orifice of a Pressure Regulator Valve Naoki Hirose, Seiei Masuda, Yasushi Matsunaga, Fumio Shimizu, Kazuhiro Tanaka, Takeshi Yamaguchi	43
New Sensing Method for Cartridge Valves Massimiliano Ruggeri	63
Bridging 1D System Modeling and 3D CFD for Hydraulic Slack Adjuster Design: A Decoupled Simulation Approach Vignesh Jegannathan, Chris Imholte, Ashutosh Pandey, Bipin Kashid	80
1D System Modeling and 3D CFD Co-Simulation Methodology for Analyzing Transient Behavior in Hydraulic Valves Priyanka Viswanath, Nathan Rosenbaum, Chris Imholte, Ashutosh Pandey, Bipin Kashid	100
Analysis of Efficiency and Axial Compensation in Continuous-Contact Helical Gear Pumps through Multiphysics Modeling and Experimental Validation Parth Tawarawala, Harsh Kulkarni, Dinghao Pan, Andrea Vacca	118
CFT Cartridge Flow Meter: Enabling Continuous In-Circuit Hydraulic Diagnostics Dale G. Dietel	133
Ultra-Compact High-Speed Axial Piston Pumps for Electrified Mobile Machinery Roman Ivantysyn, Jürgen Weber	142
When Orifice Flow Must Consider Extensional Effects Austin Zapata, Andrea Vacca	165
Design and Optimization of Fluid Supply Flow Paths for a Balanced Twin-Lip Vane Pump	177

Jinglun Qu, Andrea Vacca, Sri Krishna Uppaluri

Cylinder Block Deformation as Indicator for Cylinder Pressure in Axial Piston Units: A Novel Method Using Strain Gauges

Karl Stübig, Luca Löding, Frank Geuer, Lars Leonhard, Matthias Liermann, Sumeru Srinivasamurthy, Carsten Fiebing, Robert Rahmfeld 192

Full 3D CFD and FEM Validation of Surrogate-Optimized Hydraulic Manifolds Designed via the MOThEA Framework

Samuele Nizzardo, Andrea Vacca, Alberto Benato 206

Experimental Characterization and Numerical Modelling of an Electric Vehicle Thermal Management System Integrating a Screw-Type e-Pump

Stefano Li Veli, Luca Romagnuolo, Valerio Di Cosmo, Emma Frosina, Adolfo Senatore 221

Numerical analysis of the influence of ball-joint clearance on the elastohydrodynamic performance of slipper interfaces in axial piston motors

Julien Ntede Florent, Ishan Suvarna, Lizhi Shang, Andrea De Pascale, Davide Moro, Alberto Gatti 236

Using Nondimensional Analysis to Model, Design, and Test Hydraulic Control Systems

Noah D. Manring

University of Missouri, ManringN@missouri.edu

Abstract

This paper presents the merits for modelling and designing hydraulic control systems using nondimensional analysis. It is shown in this paper that nondimensional analysis provides physical insight into the most important features of the control system, facilitating model reduction, providing a basis for machine-design sensitivity analysis, and reducing laboratory experiments. In engineering applications, nondimensional analysis is commonly used for fluid-thermal systems that depend on nondimensional groups such as the Reynold Number, the Mach Number, the Nusselt Number, among others. Even in dynamical systems we use the nondimensional damping ratio to describe the time response of a second order system; however, it is the author's opinion that nondimensional analysis presents a powerful technique for studying hydraulic control systems that should be exploited more fully in our industry. In this paper, the author will present three case studies to show the value of using nondimensional analysis for modelling a hydraulic control system, designing hydraulic machinery, and reducing laboratory experiments. By following these examples, it is believed that efficiency will be improved in bringing products to market, and the quality and performance of our products will be advanced.

Keywords. Hydraulic control systems, nondimensional analysis

1. INTRODUCTION

Most engineering work is done using dimensions, because we ultimately need dimensions to build physical systems. For example, we need to know if the piston diameter should be 25.4 mm or if that is too small or too large. So, in the end, we will always revert to a dimensional value of a parameter when building hydraulic control systems. However, the physical world really doesn't care if we measure our parameters in pounds, newtons, or stones. And in fact, the physics of a problem are often clouded by those dimensions. For that reason, scientific analysis is usually done in nondimensional form. The thesis of this paper states that nondimensional analysis is also useful for engineering applications; and in this paper we illustrate this fact for the modelling, designing, and testing of hydraulic control systems.

This paper is more of a didactic summary rather than a research paper. As such, the references in this document are used to draw examples from the author's past work to show how nondimensional analysis may be used to improve our understanding of hydraulic control systems. The illustrations are germane to modelling the dynamics of the system, designing machine parameters, and testing operating efficiency.

2. MODELLING HYDRAULIC CONTROL SYSTEMS

Hydraulic control systems are often used as an example to teach courses on dynamic systems and control where a strong focus is placed on mathematical modelling. Many examples are presented in such a class to show that dynamical systems may be simplified if their fundamental equations are nondimensionalized. This method of modelling applies to hydraulic systems as well. **Figure 1** presents a dynamical system that we will use to illustrate the benefits of modelling hydraulic control systems using nondimensional analysis.

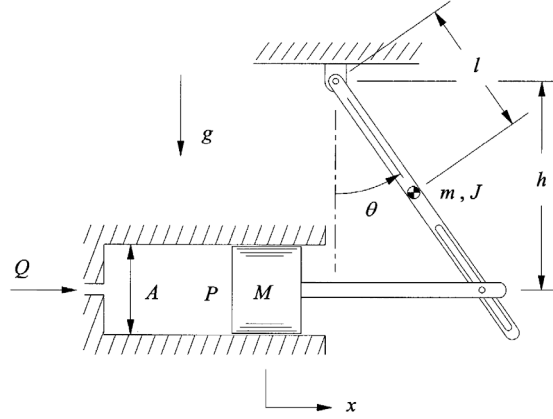


Figure 1. Example dynamic system used to illustrate nondimensional modelling [1]

We begin by presenting the model in its dimensional form. Using standard Newtonian mechanics and the conservation of fluid mass, it may be shown that the dynamic equations for this system are given by

$$M \frac{d^2 x}{dt^2} = P A - c \frac{dx}{dt} - f \quad (1)$$

$$(J + m l^2) \frac{d^2 \theta}{dt^2} = -m g l \sin(\theta) - b \frac{d\theta}{dt} + f h \quad (2)$$

$$\frac{V}{\beta} \frac{dP}{dt} = Q - K P - \frac{dV}{dt} \quad , \quad (3)$$

In these equations the symbols c and b represent coefficients of viscous damping, K is the coefficient for low Reynolds number leakage, β is the fluid bulk modulus, f is the reaction force at the sliding pin-joint, and all other symbols are self-explanatory. The volume of fluid in the piston chamber is given by

$$V = V_o + A x \quad (4)$$

where V_o is the volume of fluid in the piston chamber when $x = 0$. The nonlinearities of Eq. (1) tend to complicate things. If we assume that θ doesn't depart too far from zero, a Maclaurin series may be written to show that:

$$\sin(\theta) = \theta = \frac{x}{h} . \quad (5)$$

Substituting this result back into Eq. (1) produces the following simplified form of the dynamic equations

$$M_{eff} \frac{d^2 x}{dt^2} + C_{eff} \frac{dx}{dt} + K_{eff} x = P A \quad \text{and} \quad \frac{V_o}{\beta} \frac{dP}{dt} + K P = Q - A \frac{dx}{dt} \quad (6)$$

where

$$M_{eff} = M + \frac{J + m l^2}{h^2} , \quad C_{eff} = c + \frac{b}{h^2} \quad \text{and} \quad K_{eff} = \frac{m g l}{h^2} . \quad (7)$$

Equation (6) is the simplest, dimensional form of the dynamic equations that we can write. This equation represents a third-order dynamical system that depends upon 12 dimensional parameters. In other words, we can adjust the dynamical characteristics of this system by changing up to 12 things, but physically speaking things are not that complicated. If we nondimensionalize Eq. (6) the dynamics may be shown to depend on only 3 nondimensional groups. To do this we will make the following definitions where symbols with subscripts are steady-state values for each parameter, symbols with carets are nondimensional and of order one, and τ is a characteristic time constant of the system:

$$x = h \hat{x} , \quad P = P_o \hat{P} , \quad Q = Q_o \hat{Q} , \quad t = \tau \hat{t} \quad (8)$$

Substitution of these definitions into Eq. (6) produces the following equations:

$$\begin{aligned} \frac{M_{eff}}{\tau^2 A P_o} \frac{d^2 \hat{x}}{d\hat{t}^2} + \frac{C_{eff}}{\tau A P_o} \frac{d\hat{x}}{d\hat{t}} + \frac{K_{eff}}{A P_o} \hat{x} &= \hat{P} \\ \frac{V_o}{\beta} \frac{P_o}{\tau Q_o} \frac{d\hat{P}}{d\hat{t}} + \frac{K P_o}{Q_o} \hat{P} &= \hat{Q} - \frac{A h}{\tau Q_o} \frac{d\hat{x}}{d\hat{t}} . \end{aligned} \quad (9)$$

From steady-state analysis of this equation, where all time derivatives vanish, it can be shown that

$$Q_o = K P_o \quad \text{and} \quad P_o = \frac{K_{eff} h}{A} . \quad (10)$$

Substituting these results back into Eq. (9) produces the following form:

$$\begin{aligned} \frac{M_{eff}}{\tau^2 K_{eff}} \frac{d^2 \hat{x}}{d\hat{t}^2} + \frac{C_{eff}}{\tau K_{eff}} \frac{d\hat{x}}{d\hat{t}} + \hat{x} &= \hat{P} \\ \frac{V_o}{\beta} \frac{1}{\tau K} \frac{d\hat{P}}{d\hat{t}} + \hat{P} &= \hat{Q} - \frac{A^2}{\tau K K_{eff}} \frac{d\hat{x}}{d\hat{t}} . \end{aligned} \quad (11)$$

If we define the characteristic time as

$$\tau = \frac{A^2}{K K_{eff}} \quad (12)$$

and substitute this into Eq. (11) the simplest form of the nondimensional equations becomes

$$\xi_1 \frac{d^2 \hat{x}}{d\hat{t}^2} + \xi_2 \frac{d\hat{x}}{d\hat{t}} + \hat{x} = \hat{P} , \quad \xi_3 \frac{d\hat{P}}{d\hat{t}} + \hat{P} = \hat{Q} - \frac{d\hat{x}}{d\hat{t}} \quad (13)$$

where the 3 nondimensional coefficients / groups are given by

$$\xi_1 = \frac{M_{eff} K_{eff} K^2}{A^4} , \quad \xi_2 = \frac{C_{eff} K}{A^2} \quad \text{and} \quad \xi_3 = \frac{V_o K_{eff}}{\beta A^2} . \quad (14)$$

We have just reduced the dynamical complexity of the governing equations from depending on 12 dimensional parameters, to depending on only 3 nondimensional groups! That is powerful in and of itself, but there is more to be gleaned from these results. Since hydraulic control systems are known for having a high effort-to-inertia ratio, and since leakage coefficients need to be small for high efficiency, it may generally be shown that $\xi_1 \ll 1$ and its associated term in the equations may be ignored without loss of accuracy. Similarly, if the viscous damping and leakage is small compared to the actuator size it can be shown that $\xi_2 \ll 1$ and its associated term may also be neglected. Finally, if the fluid bulk modulus is large then $\xi_3 \ll 1$ and the compressibility term in the pressure equation may also be ignored. In other words, our nondimensional analysis has provided enough insight to show what physically matters in the equations under typical conditions. Whereas we thought we had a complex third-order dynamical system, in reality we have a fairly simple first-order dynamical system described by the following equations:

$$\frac{d\hat{x}}{d\hat{t}} + \hat{x} = \hat{Q} \quad \text{and} \quad \hat{P} = \hat{x} . \quad (15)$$

These equations can be solved by hand for a given input $\hat{Q}$. Once Eq. (15) is solved for $\hat{x}$ and $\hat{P}$, these values can be re-dimensionalized using the definitions in Eq. (8). Time will also need to be made dimensional by using the time constant in Eq. (12) and that is very straightforward.

Since Eq. (15) represents a linear, first-order system, we know that the settling time for the transient response is four times the time constant given in Eq. (12). More explicitly, the settling time is given by

$$t_s = 4 \times \frac{A^2}{K K_{eff}} \quad (16)$$

This equation shows that the settling time is dominated by the actuator size. A large actuator will take longer to reach steady state compared to a small actuator.

3. DESIGNING HYDRAULIC MACHINERY

When designing hydraulic machinery, one of the more difficult tasks is to decide which parameter to adjust to make performance improvements. Conceivably, adjusting some parameters may have a minimal effect on the desired outcome. Adjusting other parameters may have a large effect. In this section we show how to use nondimensional analysis to identify the most important parameter to adjust when trying to increase the fill speed of an axial piston pump.

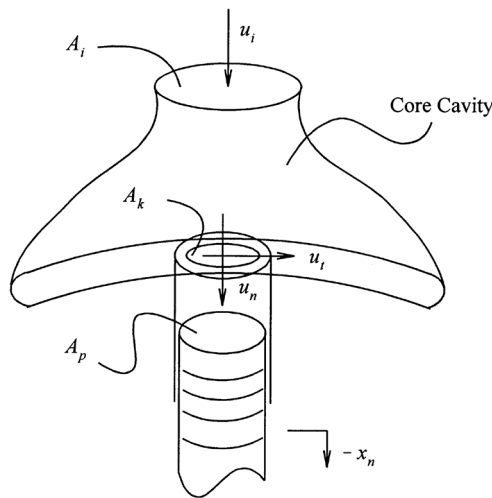


Figure 2. A schematic showing a piston chamber filling with fluid [2]

Figure 2 shows a picture of a single piston within an axial piston pump that is being filled with fluid as the piston withdraws from the piston chamber. The core cavity shows the intake port of the pump. Using Bernoulli principles, it may be shown that the piston will stop filling (and fluid will start to vaporize) when the pump speed exceeds the following limit [2]:

$$\omega < \sqrt{\frac{2 P_i}{\rho r^2 \left\{ 1 + \left[\left(\frac{A_p}{A_k} \right)^2 - \left(\frac{N A_p}{\pi A_i} \right)^2 \right] \tan^2(\alpha) \right\}}} . \quad (17)$$

In this equation P_i is the absolute inlet pressure of the pump, ρ is the fluid density, r is the piston pitch radius, A_p is the cross-sectional area of the piston, A_k is the kidney area on the cylinder block, A_i is the inlet area of the core cavity, N is the total number of pistons in the machine, and α is the swash plate angle of the pump.

To consider the sensitivity of ω to various design changes, we use a first order Taylor series to study design perturbations about a nominal design condition. Using Eq. (17), we will consider the perturbation of three parameters for illustration. We will perturb the inlet pressure P_i , the piston pitch radius r , and the kidney area A_k . Other parameters could also be studied. The resulting Taylor series is then:

$$\omega = \omega_o + \left. \frac{\partial \omega}{\partial P_i} \right|_o (P_i - P_{i_o}) + \left. \frac{\partial \omega}{\partial r} \right|_o (r - r_o) + \left. \frac{\partial \omega}{\partial A_k} \right|_o (A_k - A_{k_o}) \quad (18)$$

where the subscript “o” indicates that the parameters are evaluated at the nominal design conditions. Equation (18) may be rewritten as follows

$$\frac{\omega}{\omega_o} = 1 + \underbrace{\left. \frac{\partial \omega}{\partial P_i} \right|_o \frac{P_{i_o}}{\omega_o} \left(\frac{P_i}{P_{i_o}} - 1 \right)}_{S_{P_i}} + \underbrace{\left. \frac{\partial \omega}{\partial r} \right|_o \frac{r_o}{\omega_o} \left(\frac{r}{r_o} - 1 \right)}_{S_r} + \underbrace{\left. \frac{\partial \omega}{\partial A_k} \right|_o \frac{A_{k_o}}{\omega_o} \left(\frac{A_k}{A_{k_o}} - 1 \right)}_{S_{A_k}} \quad (19)$$

where S_{P_i} , S_r , and S_{A_k} are nondimensional sensitivity coefficients. Since these coefficients are nondimensional the coefficient with the largest nondimensional magnitude will have the greatest impact on adjusting the fill speed for the pump. It may be shown that these coefficients are given by

$$S_r = \left. \frac{\partial \omega}{\partial r} \right|_o \frac{r_o}{\omega_o} = -1 \quad , \quad S_{P_i} = \left. \frac{\partial \omega}{\partial P_i} \right|_o \frac{P_{i_o}}{\omega_o} = \frac{1}{2} \quad , \quad \text{and} \quad (20)$$

$$S_{A_k} = \left. \frac{\partial \omega}{\partial A_k} \right|_o \frac{A_{k_o}}{\omega_o} = \frac{1}{1 + \left(\frac{A_k}{A_p \tan(\alpha)} \right)^2 - \left(\frac{N A_k}{\pi A_i} \right)^2} = 0.233 \quad .$$

These results are numerically calculated for a 45 cc/rev machine with the following design

parameters: $N = 9$, $A_p = 2.28 \text{ cm}^2$, $A_k = 1.52 \text{ cm}^2$, $A_i = 4.56 \text{ cm}^2$, $\alpha = 18 \text{ deg}$, $r = 3.38 \text{ cm}$, $\rho = 850 \text{ kg/m}^3$, and $P_i = 1 \text{ atm}$. As shown in Eq. (20) the most sensitive thing we can adjust to increase the fill speed of the pump is to reduce the piston pitch radius, r . Adjustments to this parameter are twice as impactful as changes to the inlet pressure, and four times as impactful to changes in the kidney area. These insights are helpful for design guidance, and are not obvious without the use of nondimensional analysis.

4. TESTING HYDRAULIC PUMPS

A common experiment for hydraulic control systems is to measure the efficiency of a hydraulic pump. This is often done by measuring relevant parameters of the machine over a wide range of operating conditions, usually producing hundreds of data points and multiple charts [2]. Using nondimensional analysis, it may be shown that a full understanding of pump efficiency may be obtained by taking only 10 measurements and creating 1 chart.

The *Buckingham Pi Theorem* states that if a physical problem is described by a relevant set of n dimensional parameters which use m independent, fundamental dimensions (e.g., time, length, mass) then a complete set of $n - m$ nondimensional groups may be used to completely describe the problem [3].

Consider our interest in measuring the *overall* efficiency of a hydraulic pump. It turns out that this problem depends on 5 dimensional parameters: discharge pressure P , discharge flow rate Q , shaft torque T , shaft speed ω , and absolute fluid viscosity μ . These parameters are comprised of 3 fundamental dimensions: time, length, and mass. This means that the overall efficiency problem for the pump should be completely described by 2 nondimensional groups. From the relevant set of dimensional parameters, we select these groups to be the overall efficiency and the Hersey number:

$$\eta = \frac{PQ}{T\omega}, \quad H = \frac{\mu\omega}{P}. \quad (21)$$

The Hersey number is often found in fluid mechanics textbooks. Similarly, if we are interested in measuring the *mechanical* efficiency of a hydraulic pump there are 5 dimensional parameters that describe this problem: discharge pressure P , volumetric displacement V_d , shaft torque T , shaft speed ω , and absolute fluid viscosity μ . The 3 fundamental dimensions of this problem are the same as before which means that 2 nondimensional groups may be used to fully describe this problem:

$$\eta_m = \frac{V_d P}{T}, \quad H = \frac{\mu\omega}{P}. \quad (22)$$

Similarly, it may be shown that the *volumetric* efficiency may be described with the following nondimensional groups:

$$\eta_v = \frac{Q}{V_d \omega} , \quad H = \frac{\mu \omega}{P} . \quad (23)$$

In other words, all efficiency numbers may be plotted on a single chart as they vary with the Hersey number. If a full range of the Hersey number is tested, this chart should describe the efficiency of the pump for the entire operating range of the machine.

The Appendix presents experimental data that was taken while measuring the efficiency of a 145 cc/rev axial piston, swashplate type hydrostatic pump. **Figure 2** shows the efficiency data plotted against the Hersey number. Only 10 data points are used, and the efficiency plot is complete for all operating conditions of the machine. The claim of this nondimensional approach is that only 10 data points and 1 chart is needed for a full description of the pump efficiency across the entire operating range of the machine. This is a significant reduction in engineering effort, and it is also a very useful way to present efficiency data for new product literature.

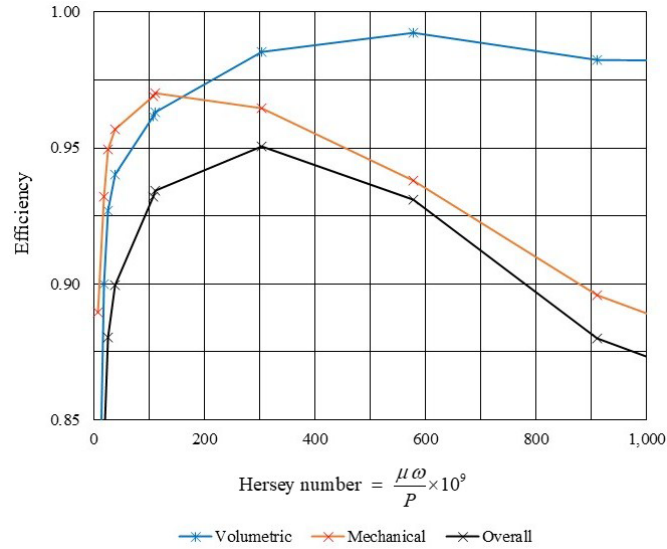


Figure 2. A complete map of efficiency for a 145 cc/rev pump

5. CONCLUSION

The following conclusions are supported by the analysis, discussion, and results as presented in this paper:

1. That nondimensional analysis can reduce the complexity of dynamical systems, and that physical insights may be gained which enable the order reduction of the dynamical model.

2. That nondimensional analysis may be used to study the sensitivity of changing machine design parameters, and that this approach elucidates the most important variables to be changed for improved operation.
3. That nondimensional analysis may be used to streamline the experiments that are conducted for the efficiency of hydraulic machinery, and that this approach may be used to reduce experimental effort.

As noted in the introduction of this paper, it is important to always re-dimensionalize the analysis in order to build an actual hydraulic control system. This re-dimensionalization is straightforward using the definitions that were made to conduct the nondimensional analysis.

6. APPENDIX

The following table includes the test data that was used to generate **Fig. 2**:

#	$H \times 10^6$	T [Nm]	ω [rpm]	P [MPa]	Q [lpm]	Temp [C]	μ [cP]	η_v	η_m	η
1	7	911	500	34.64	57.65	112.43	4.66	0.784	0.889	0.697
2	19	863	1,525	35.41	198.30	110.97	4.92	0.900	0.932	0.839
3	26	749	998	30.22	132.18	107.30	5.64	0.927	0.950	0.880
4	38	735	1,999	30.06	272.87	109.03	5.29	0.940	0.957	0.900
5	108	480	1,001	20.11	141.39	63.49	28.57	0.962	0.969	0.932
6	112	243	500	9.90	70.85	53.65	41.14	0.963	0.970	0.934
7	305	133	999	5.09	144.32	51.64	44.31	0.985	0.965	0.951
8	578	244	1,482	9.79	216.24	56.90	36.47	0.992	0.938	0.931
9	911	147	2,178	5.11	319.64	52.88	42.32	0.982	0.896	0.880
10	1,345	143	1,985	5.02	291.37	51.94	43.81	0.982	0.863	0.847

7. REFERENCES

- [1] Manring, N.D., and R.C. Fales. 2020. Hydraulic Control Systems, 2nd ed. John Wiley, Hoboken, New Jersey
- [2] Manring, N.D. 2013. Fluid Power Pumps and Motors – Analysis, Design, and Control. McGraw Hill, New York
- [3] Wheeler A. and A.R. Ganji. 1996. Introduction to Engineering Experimentation. Prentice Hall, Upper Saddle River, New Jersey

How Fluid Power Research Can Contribute to a More Sustainable Construction Industry

Prof. Dr. Olivier Reinertz, Christian Bertram, M. Sc., Prof. Dr. Katharina Schmitz

*RWTH Aachen University, Institute for Fluid Power Drives and Systems (ifas),
post@ifas.rwth-aachen.de*

Abstract

The construction industry is facing severe challenges in near future. The reasons are widespread. First, the growing earth population leads to a continuously rising demand for infrastructure, living and industrial space, forcing the construction industry to raise productivity. On the other hand, the building and construction sector is responsible for approx. 34 % of the global anthropogenic CO₂ emissions, while this share will grow further with enhanced construction activity. Significant parts of these emissions result from the production of building materials. Cement production alone generates approx. 6 % of the global emissions, due to its enormous used quantities and high temperature requirements in production. Paradigm changes in raw material selection, building designs, and material use are therefore unavoidable. Additionally, the drive systems of mobile construction machines contribute to the sector's emissions. While construction principles need to change, the requirement for earth moving and mining and thus for more efficient and emission-free working machines persists. Finally, construction is a labor-intensive branch, facing severe problems with recruitment due to the shrinking number of young people in many industrialized countries and the direct concurrence with more attractive, less physically demanding jobs.

The paper shows how fluid power can contribute to an ecologically and socially more sustainable construction industry by means of several examples from current research at ifas and beyond. Herein, both dimensions of sustainability are simultaneously optimized by automation of construction processes, e.g. by means of partly automated mobile machines with additional intelligence, to allow real time surveillance of the building progress, fluid power enabled automated prefabrication of lightweight assemblies, which require less concrete and labor on site, and enhanced switching schemes and fluid selection for mobile hydraulics to ensure increased energetic efficiencies.

Keywords. Fluid Power Research, Construction Industry, Sustainability.

1. INTRODUCTION

The world is facing a strongly growing population, which is forecasted to peak only in the year 2084 with 10.3 billion people living on earth [1]. This implies that over the next 60 years, living space, production facilities, transportation, etc. need to grow by approx. 25 % of the existing infrastructure only to maintain the current average standard of living. Meanwhile, the global birth rate had its maximum already mid of the last decade and is shrinking [1], which will lead to a significantly reduced number of young craftsmen in a few decades in many countries. Ultimately, higher demands on productivity, automation, and reduced manual workload in construction result. This is especially the case for China, Europe, India, and Japan while persistent population growth, and thus a stable number of workers is forecasted for the United States over the century [1].

In parallel, the limitation of climate change, which is attributed by most scientists and governments to the anthropogenic greenhouse gas emissions, results in higher demands, awareness, and legislative restrictions on these emissions. Starting with the UN conference in Stockholm 1972, possible pathways to minimize greenhouse gas emissions aiming to limit climate change to a temperature increase of 1.5 °C above pre-industrial levels end of the century have been discussed. With the Paris Agreement, a binding international treaty was achieved in 2015, which paves a way to the temperature goal by defining quantitative reductions in greenhouse gas emissions over time to achieve net zero emissions in 2050. Meanwhile, recent studies show, that despite huge efforts to reduce greenhouse gas emissions, economic growth has offset these reductions and emissions still not declined. Peak emissions are forecasted for the current period and a temperature increase of 2 °C will probably result. [2]

The combined challenges of construction industry resulting from fast-growing population, which requires intensive construction activities, reduced availability of workers, demanding for highest productivity and automation, and limitations in greenhouse gas emissions, can only be addressed by innovative solutions on all involved fields of research. Therefore, a comprehensive understanding of sustainability is required, not only including closed material cycles and strongly reduced climate gas emissions during production and service life but also the social and demographic dimension.

In the following, the role of fluid power to solve the before mentioned challenges in construction industry is discussed.

2. CLIMATE IMPACT AND FUTURE OF CONSTRUCTION INDUSTRY

In 2024, the building and construction sector accounted for about 34 % of global energy-related CO₂ emissions [3]. In addition to building operations, such as heating, ventilation, air conditioning, and lighting, the production of raw materials and the operation of construction sites are significant drivers of carbon emissions in this sector. Furthermore, the sector's immense resource consumption negatively impacts its sustainability.

In 2024, 3,950 Mt of cement were produced [4]. Including emissions from associated industrial processes, cement production accounted for 2,324 of 38,153 Mt total CO₂ emissions in this year, corresponding to a share of 6.1 % [4]. Approx. 60 % of these emissions are attributed to the heated limestone, while the remaining 40 % is linked to the

operation of the cement kiln and other production processes [5]. While global net-zero emissions are targeted by 2050, cement demand is projected to reach 3,535 Mt accounting for only 48 Mt of CO₂ emissions for this year, as shown in Figure 2.1 [4]. A significant reduction in CO₂ emissions from production is therefore necessary: from 588.4 kg of CO₂ per ton of cement in 2024 to 13.6 kg of CO₂/t in 2050 to achieve global net-zero emissions.

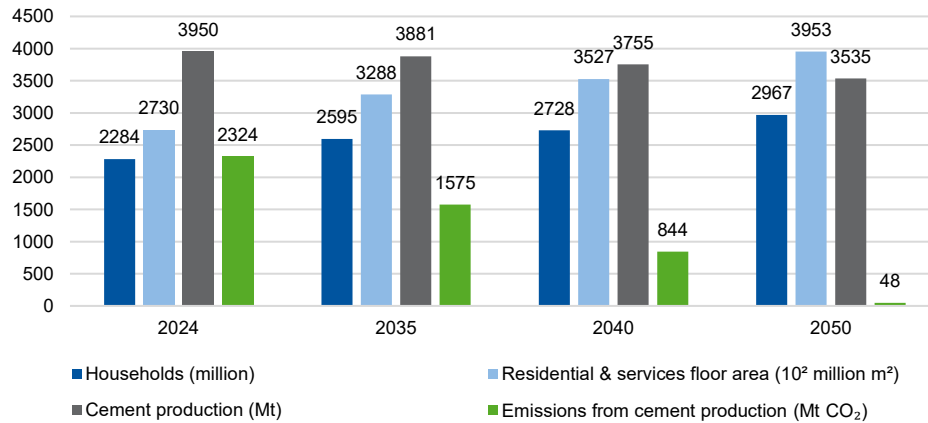


Figure 2.1. Trends in household and residential & services floor area demand compared with cement production and correlated CO₂ emissions, based on actual values (2024) and targets of the ‘Net-Zero Emissions by 2050’ scenario [4]

To reach this goal, while maintaining or even improving performance, the proportion of Portland clinker must be reduced or substituted with lower-emission materials [5]. Emissions can be further reduced by implementing process-related changes, such as improving energy efficiency and operating the cement kiln with hydrogen or electricity.

In addition to cement, steel is an important raw material for the construction sector. Although not all the steel produced worldwide – 1,883 Mt accounting for 2,750 Mt of CO₂ emissions in 2024 – is used in the construction sector, this resource still contributes significantly to the industry's emissions [4]. Additionally, approx. 400 Mt of CO₂ emissions were produced by mobile hydraulic machinery in the construction sector in 2022, corresponding to approx. 1 % of total CO₂ emissions [6].

Consequently, the climate targets in the ‘Net-Zero Emissions by 2050’ scenario can only be achieved through higher design and construction efficiency (approx. 22 % of the total contribution) as well as low-emission materials and manufacturing processes (approx. 31 % of the total contribution). Additional measures, such as carbon capture during cement production, are also necessary [5].

It is obvious that switching hydraulic construction machinery from combustion engines to hydrogen or battery power alone is insufficient. Additional measures on machine side are required for economic and social viability. Selected pathways and their potential are discussed in the following chapter 3. Furthermore, novel automation approaches to raise productivity in construction and to allow usage of emerging material-minimized construction methods, e.g. carbon-reinforced concrete (CRC), partly rely on fluid power drives and are discussed in chapter 4.

3. FLUID POWER RESEARCH FOR EFFICIENT MOBILE MACHINES

Most mobile working machines used in construction are driven by one central prime mover. This can either be an internal combustion engine or an electric motor for battery electric systems. The supplied power is transferred by one or several hydraulic pumps to a valve control system, which distributes the power and controls the multiple actuators by throttling. Especially the parallel movement of multiple actuators with significantly deviating load pressures poses a challenge concerning efficiency for conventional systems if the pressures cannot be decoupled by using separate pumps for the active drives. Meanwhile, several studies show that trained excavator operators use multiple drives simultaneously over a significant share of the operational time [7], which leads usually to a mismatch between available number of pumps, their flow capability, and the required motions even on heavy machines with several main pumps. Therefore, state-of-the-art valve systems are generally not built to drive each actuator with a single pump. Furthermore, state-of-the-art switching schemes do not allow for energy recuperation and generate extensive throttling losses to control assistive loads, e.g. during deceleration of the swing or boom down of an excavator.

In contrast to former products, the ongoing electrification of mobile working machines has the potential to realize emission free operation, if clean electric energy is assumed, which is targeted for 2050 [2]. The Chinese market for such machines is worldwide the first rapidly growing, while wheel loaders are the driving component [8]. To allow for cost and material effective machines with reasonable battery capacity and sufficient operation time between charging cycles at a time, efficiency is even more relevant for battery electric systems than for conventional ones.

As efficiency is increasingly important and the highest optimization potential lies on system level, most research activities concerning mobile hydraulic efficiency optimization of the last decades have focused on concepts to decouple the load pressures of different parallel used drives and to allow for energy recuperation. Consequently, different approaches have emerged:

At ifas for example, the STEAM architecture, which consists of a multi-pressure rail system with accumulator buffered quasi constant pressure rails, allowing for load adaptive selection of the driving and counteracting pressures on conventional cylinders, has been developed and tested. While its energy savings attributed to reduced hydraulic throttling losses stayed reasonable, huge improvements in overall efficiency have been achieved by its quasi-ideal loading of the ICE and pump. For three representative work cycles, efficiency increase of 27 % to 32 % has been achieved in comparison to a conventional load-sensing machine [9].

A second widely discussed approach is the usage of hydraulic multi-chamber cylinders, which are supplied by at least two constant pressure rails [10]. The fundamental concept is comparable to the multi pressure rail systems but differs in switching constant pressure on variable cylinder chambers instead of variable pressures on constant surfaces. Industrialization of this concept is pursued by companies *Norrhydro* and *Volvo*, claiming up to 50 % reduction in fuel consumption on an excavator [11]. Of course, combinations of these two approaches can be found in literature. A recent example, which considers three pressure rails and multiple three chamber cylinders, predicts efficiency increases of 22.3 % to 36 % in comparison to a state-of-the-art system [12].

Another approach to the same technical problem is the usage of hydraulic transformers to adapt the pressure from a constant pressure rail to the loading condition of the connected cylinder. Generally, these transformers consist of two coupled displacement units with at least one variable displacement to adjust pressures. Therewith, throttling losses can be omitted and power transferred in both directions, allowing for recuperation of cylinder power into the supply line and connected accumulators. Due to the high number of required displacement units, a highly integrated transformer design with improved efficiency and noise emissions is key. Significant achievements in this field have been realized by *Peter Achten* [13]. Simulated efficiency increases of the hydraulic system of an excavator using hydraulic transformers in comparison to its conventional counterpart are forecasted to be as high as 31 % [14]. Meanwhile, additional efficiency increase of the machine due to optimal loading of the prime mover was neglected in this study, showing even higher potential in general.

Another approach considers the direct displacement control of all actuators by a set of variable displacement machines, which are driven by one common prime mover. While it allows avoidance of control valves and associated losses, additional measures need to be taken to allow for energy recuperation, as the recuperated energy on the pump and prime mover shaft needs to be stored somehow. *Hippalgaonkar and Ivantysynova* have shown that for a 5 t mini excavator, the fuel consumption could be cut by half and thus efficiency doubled by pure displacement control with recuperation capability [15]. Efficiency gains for bigger machines, which have usually more efficient baseline hydraulics, will be probably lower while for realization, pump losses and investment costs are the main challenges, as the overall installed displacement volume of the machine must be significantly higher than for its conventional counterpart to achieve comparable performance.

A possible approach to reduce displacement volume and cost of the installed units has recently been proposed by *Schmidt and van Binsbergen-Galán*. By connecting two chambers from two different working cylinders, this connection and the two remaining ports can be controlled in pressure and/or flow by means of displacement units. The performed statical analysis forecasts a reduction in overall displacement volume of up to 36 % in comparison to a conventional displacement-controlled approach. [16]

If electrified machines are investigated, electro-hydrostatic actuators (EHA) with separate electric drives for all actuators are an option. A reduction in pump displacements is achieved by using fast rotating electric machines. Furthermore, the implementation of variable speed motors allows fixed displacement pumps to be used, e.g. internal gear pumps, to cut system costs [17]. The control of such EHA with required load-holding capability requires complex control algorithms, which have been widely researched [18]. Nevertheless, building space and cost constraints prohibit realization of pure electro-hydrostatically actuated serial machines to date.

A hybrid approach proposed by *Mu, Cheng, and Tang*, which consists of a conventional load-sensing system with only one cylinder, which is replaced by a four-chamber cylinder, to act in parallel as LS-cylinder and EHA, has shown to be an interesting and simple to implement alternative. In a case study for a battery electric 6 t excavator, this measure reduces energy consumption by 15.6 % with an EHA power rating of 7 kW. [19]

Another possible solution proposed by *Sah and Li*, consists of a multi pressure rail system, which is combined with small electro-hydrostatic units for each actuator to adjust pressure and flow to the required level. Therewith, the installed electrical power can be drastically reduced in comparison to EHA-driven machines, which means lower investments in electric motors and frequency converters, while the pump displacements of the controlling units stay comparatively large, as each unit needs to be sized for full actuator flow. [20]

In conclusion, despite many studies and realized full-size prototypes, hydraulic architectures with significantly improved efficiency are still waiting to come on the market. Different studies demonstrate that less than 50 % of the mechanical power of the prime mover shaft is transferred into working power [15, 21]. The different proposed system architectures, allowing for recuperation, reduced throttling losses, and optimal loading of the prime mover, and the achieved experimental efficiencies suggest that there is potential to cut power demand of existing machines by half. Unfortunately, costs, building space, robustness, and maintenance aspects are still hindering broader application of these approaches, especially for machines with internal combustion engines.

Meanwhile, the changed cost structure of battery electric systems makes efficiency measures economically more attractive, as battery capacity can be reduced for a more efficient system to achieve the same time of operation. Higher efficiencies also positively impact the product carbon footprint of the battery electric machines, the raw materials needed, and operational costs, while electrified machines are the most promising way to reach the greenhouse gas emission goals, under the condition that clean electric energy is available.

Hydrogen powered solutions are probably going to exist in parallel for remote environments without sufficient grid connection. Also in this case, hydraulic system efficiency is from utmost importance, as highly compressed or liquified hydrogen will stay resource demanding in production and therefore an expensive energy carrier. By using fuel cells, achievements in the field of battery electric systems can directly be applied to the hydrogen powered counterparts.

Ongoing research in this field, taking the specific possibilities of electrification into account, is forecasted to significantly improve sustainability and to allow for higher degrees in automation of mobile construction machines over the next two decades.

In contrast to the above-mentioned energy saving methods on system level, automation of construction machines is already ongoing in the market. Grade assist functions for excavators, which allow for assisted surface leveling, slope grading, or trenching and enable measuring the weight of the moved soil can be purchased as an option on selected recent machines. As a result, the process is accelerated and the operator's workload is reduced in parallel, while also less experienced operators are enabled to perform the task. Generally, for safety reasons, the operator still needs to command the machine, even if this command is reduced to continuous lever deflection to enable automatic motion. [22, 23]

Contrary to this, the company *Liebherr* showed recently a fully autonomous wheel loader for repetitive tasks, e.g. to repeatedly load hoppers or move goods from A to B [24]. Of course, this system is still very limited in comparison to the usual working tasks of some construction machines, e.g. wheeled excavators, but it represents a first step in the required direction and further, more challenging demonstrators are awaited.

Another possibility to reduce the physical load on the operator is teleoperation of mobile machines. This allows the operator to sit in a calm and warm cabin or office, while the excavator is performing its task outside. This can lead to significant risk mitigation in selected work environments, e.g. in demolition. Required components and software tools are already widely existent, thus enabling prototypical realizations. *Gerdes et al.* evaluated different visualization techniques for teleoperation and came to the result that stereoscopic visual feedback with a camera mimicking the operator's head movements is best suited. Main challenges are the user perception of the machine's motion, the awareness for the surroundings, and cyber security. [25]

This overview shows that current research and development in the field of fluid power is paving the way for an ecologically and socially more sustainable generation of mobile machinery. Meanwhile, novel opportunities for fluid power to explore new markets are created from the already ongoing transformation of the construction industry itself. In particular, the automation of manually dominated tasks, which is necessary to achieve greater productivity given the shrinking number of skilled workers, holds great potential.

4. NOVEL CONSTRUCTION PRINCIPLES INVOLVING FLUID POWER

The following section focuses on advanced approaches to automated formwork systems as an exemplary emerging technology in the construction sector that utilizes fluid power. A recent example with significantly improved productivity and safety is the PERI VARIOKIT Composite Track (VCT) system, which is shown in Figure 4.1. This system aims to build steel composite bridges faster and with higher quality. Therefore, the formwork is attached to a slide, which is supported by the bridges steel girder under the concrete slab and hydraulically moved to allow sequential buildup of the in-situ concrete carriageway slab. The slide is automatically moved along the entire bridge length by means of hydraulic cylinders using the pilgrim step method. The hydraulic drive mechanism, which threads through the anchors on the steel girder and manoeuvres the entire formwork to its intended location, is shown on the right in Figure 4.1. [26] Further examples are found in the field of automated climbing systems for high rise buildings and bridge posts, as e.g. the DOKA super climber system, which moves the formwork by one story at a time and assures its correct orientation by means of a set of hydraulic cylinders [27].

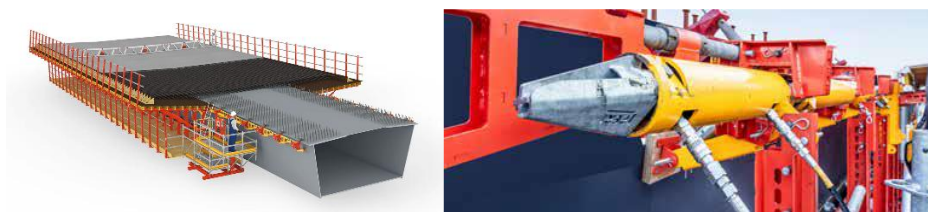


Figure 4.1. Automated formworks – PERI VARIOKIT Composite Track [26]

Besides more sustainable construction machines and higher productivity through automation, lower emissions and raw material usage associated with higher mechanical performance of the structural components are required to achieve sustainability goals in construction industry. One possible solution to meet these demands is to use innovative, sustainable materials, and tailored construction principles, which allow for industrialized prefabrication and fast assembly. Associated research in construction engineering has

therefore the potential to open new fields of application for fluid power systems, especially to automate prefabrication plants and on-site assembly.

This is where the research of the Collaborative Research Centre/Transregio 280 (CRC/TRR 280), involving researchers from TUD Dresden University of Technology, RWTH Aachen University, Leibniz Institute of Polymer Research Dresden, University of Hamburg, and TU Darmstadt, comes into play. The overarching goal of this research consortium is to develop innovative design strategies for material-minimized CRC structures to establish sustainable principles for a new approach to construction. [28–30]

The investigated carbon reinforcements consist of multiple endless fibers impregnated with a matrix resin to form bars or sheets. Due to the lower density and higher tensile strength than steel, smaller reinforcement cross-sections are possible, as shown in Figure 4.2. In addition to its use in renovating existing structures, such as residential buildings, industrial facilities, and bridges, CRC particularly demonstrates its full potential in new construction in the form of thin-walled, lightweight, and high-strength structural elements [28–31]. [32]

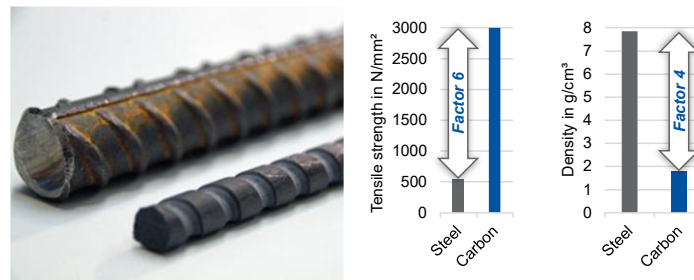


Figure 4.2. Carbon vs. steel reinforcement [32]

A further key advantage lies in the corrosion resistance of carbon fibers. Consequently, corrosion-related damage is prevented, resulting in longer lifetime for CRC and reduced maintenance costs. Additionally, the concrete cover can be reduced to the minimum required for structural stability, resulting in significant material savings. Depending on the application, concrete consumption can be reduced by up to 80 %. At the same time, CO₂-minimized binders can be used, which is significantly more complex for steel reinforcement due to oxidation issues [33]. These savings directly translate to manufacturing, transportation, installation, demolition, and recycling and positively impact economic efficiency and sustainability of the entire construction sector. [32]

Exploiting the full potential of CRC leads to significantly minimized material and volume, and thus to delicate construction designs. However, these material-minimized structural elements have very low stiffness in a flat or simply curved state. Therefore, they are not suitable for load-bearing installations, e.g. in ceiling structures, without additional measures. This is why new construction strategies and associated manufacturing methods are investigated.

A novel design strategy to achieve sufficient rigidity of these elements is to incorporate fold lines. This creates high local stiffness, resulting in a significantly higher load-bearing capacity. The strategy is inspired by classical origami and uses waterbomb tessellation. Therewith, material is used only where loads are transferred. In this way, loads up to 45 times the module's own weight can be carried. Many of these modules can be combined and

assembled into large-scale structures by exploiting the periodicity of the waterbomb tessellation, as shown in Figure 4.3; however, strict manufacturing tolerances must be maintained to evenly distribute loads in the structure. [34–36]

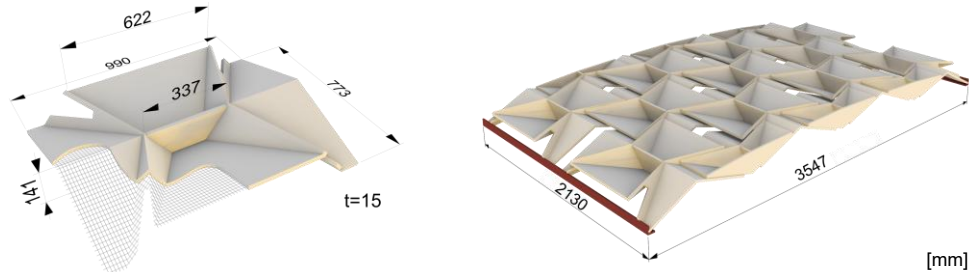


Figure 4.3. Waterbomb tessellated module and assembly [36]

These geometrically complex modules cannot be efficiently manufactured using conventional methods, such as lamination, casting, or injection molding [32]. The “fold-in-fresh” methodology represents a novel, efficient manufacturing process that is applicable to concrete in green state with soft or freshly impregnated carbon reinforcement [37]. In this process, the modules are folded from a flat, planar state into the desired final geometry using a folding tool that replicates the desired shape. The modules then cure inside the tool. Thus, this method enables a customized yet sustainable manufacturing process. [36]

The initial prototype of this folding tool was developed at ifas in cooperation with the Institute of Structural Concrete at RWTH Aachen University. The tool consists of a foldable surface that is actuated by pneumatic cylinders via a gear mechanism. The tool surface is composed of a continuous, flexible, fiber-reinforced plastic membrane, which is locally stiffened by steel facets in accordance with the waterbomb geometry. Thus, only the membrane is present in the narrow gaps between the facets, serving both as a folding joint and as a seal against the concrete matrix. Actuation causes a spatial displacement and tilting of the facets, resulting in a continuous transition from the flat, planar state into the folded structure of the final component.

Although folding a waterbomb cell theoretically involves only one degree of freedom, the soft characteristics of the membrane-based fold-lines allows flexing of the structure and thus deviations from the exact waterbomb geometry. On the one hand, this compensates for manufacturing and assembly inaccuracies throughout the entire tool, ensuring that the tool surface folds easily and no jamming occurs. On the other hand, however, to ensure defined movement of the facets and unique tool geometry in the folded state, flexing must be minimized. In the first demonstrator, this is achieved by mechanical synchronization and guidance of some degrees of freedom of the folding tool. Synchronization is achieved by implementation of a mechanically synchronized motion of two parallel linear guides, which hold the bearings of the outer facets to exploit the symmetrical properties of the waterbomb geometry. This mechanism is located beneath the tool surface and is directly actuated by five pneumatic cylinders to evenly distribute the driving force on the system and thus to avoid jamming. Meanwhile, to avoid excessive loads on the membrane joints, two additional cylinders are attached to the bearings of the inner facets, which hold the majority of concrete weight during folding due to their dominating surface. Thus, in sum, seven force-controlled

pneumatic actuators, which are pneumatically interconnected in three symmetrical groups, are used to evenly actuate the folding tool and allow for smooth movement.

Despite the resulting kinematic overdetermination, the system does not jam thanks to the elasticity of the membrane and pneumatic actuators. Smooth motion is achieved through a combination of throttles and time-based pressure control, which provides drive-group-specific, time-varying force. For a detailed description of the initial prototype of the folding tool, as shown in Figure 4.4, see *Bhat et al.* [38]. Besides the required force control, pneumatic actuation has been selected for its outstanding robustness and simple maintenance in the dusty and harsh environment of a construction laboratory and later industrial usage.

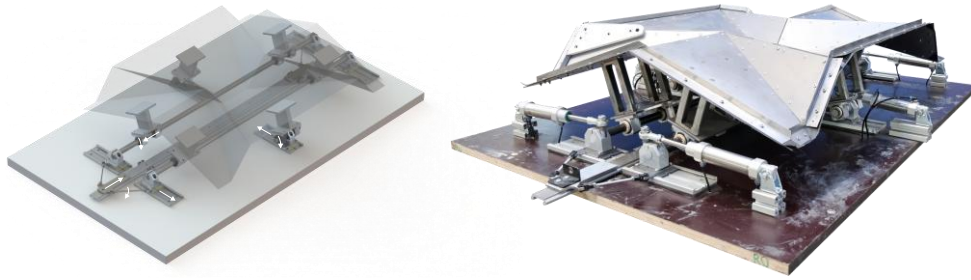


Figure 4.4. Kinematics and pneumatically actuated prototype of the folding tool in the folded state [38]

However, *Gomes et al.* [36] have shown that the prototype of the folding tool does not yet provide the manufacturing precision necessary to assemble large numbers of waterbomb modules. Current systematic geometric deviations in the manufactured modules largely result from inaccuracies in assembling the folding tool, specifically the drive and guiding components. The soft design of the joints on the tool's surface also plays a decisive role, as it does not represent an ideal revolute joint.

Future work will include developing a suitable joint design that exhibits higher stiffness and defined movements while providing sealing against concrete. This research relies on ongoing CT scans of a specially designed CT compatible joint test rig with a limited number of facets to analyze the interactions between concrete, reinforcement, joints, and the folding process. Furthermore, the gear mechanism in the tool's frame will be minimized so that, in the future, the facets will be primarily actuated directly by position-controlled pneumatic cylinders.

Especially when using stiff joints to allow for higher accuracy, pure position control of the tool surface with stiff actuators, e.g. electromechanical spindle drives, would require extensive path planning. If imperfect synchronization of the actuator movements during motion occurs or the tool surface requires small deviations from the calculated positions due to tolerances in manufacturing, significant internal stresses would result, which are associated with the risk of tool surface degradation or destruction. Here, the low stiffness of pneumatic actuators opens new possibilities, as despite position controlled, the tool surface is suspended by a set of soft actuators, allowing for simple overload protection thus limiting internal stress. Moreover, based on pneumatic elasticity, an incremental approach for position control can be pursued as described in the following.

For this incremental approach, three-dimensional geometry detection is required via a process-integrated optical measurement of the part geometry. A comparison of the current state with design data of the final component allows movement of the multiple actuators in small steps and thus an iterative stepping towards the final geometry. To fulfill the resulting requirements on actuator level, cylinder drives with small independent metering on/off valves and a pneumatic brake to allow for later hardening without crack inducing micro-motions are investigated.

This approach substantially reduces the number of sensors needed in the multi-actuator system in comparison to stroke sensor relying closed-loop position control of all actuators. Furthermore, switching valves reduce the risk for contamination induced downtime in comparison to servo-valves in the harsh environment. Hence, direct geometry control results in significant cost savings, higher robustness, and avoids extensive path planning and tolerance management on tool level. Additionally, geometric deviations in the tool surface resulting from manufacturing and joint design are directly accounted for.

The corresponding superordinate control system is still being developed that will transmit cylinder-specific step commands to the actuators by comparing the current geometry with the target geometry. Furthermore, current research is exploring additional possibilities for inline quality control, such as crack detection by means of the already installed optical system.

In future, the tool is intended to manufacture various geometries to ensure automated, economical production of user specific parts. To this end, the tool must achieve higher geometrical flexibility, as already identified by *Gomes et al.* [36]. Therefore, modifications to the tool surface and consequently to the actuators are necessary. Different concepts for the variation of the tool surface tessellation have been investigated. Replacing the entire tool surface with integrated joints while simultaneously reconfiguring the actuators has been identified as the most efficient approach. With this approach, a dedicated tool surface is available for each tessellation, while different geometries can be achieved per tool surface by altering the degree of folding.

Figure 4.5 shows a schematic illustration of the overall concept for the next generation of the folding tool. To adjust the drive system to varying target geometries, the pneumatic drives beneath the surface can be repositioned within the base frame according to the fold-specific requirements, and/or they can be substituted with actuators of varying lengths. Thus, the cylinders must be attached to the base frame and surface in a way that allows for easy relocation and exchange. For this, the base frame provides a large number of couplings for compressed air, which also serve as mechanical fixpoints between the cylinders and the tool frame, offering suitable mounting options. To facilitate use, the cylinders are equipped with decentralized electrical power supply, control, valves, and signal processing. These actuators, referred to as Servo-Pneumatic Actuators (SPA) in the following, can be implemented, for example, by integrating the pneumatic coupling, spherical joints, switching valves with a sufficiently low flow rate and a battery-powered microcontroller with wireless communication to a central control system, into a pneumatic cylinder with braking unit. The focus of the SPA is on positioning accuracy rather than achieving high travel speeds, as this is not necessary for the present application. The development of such SPA is the subject of current research at ifas.

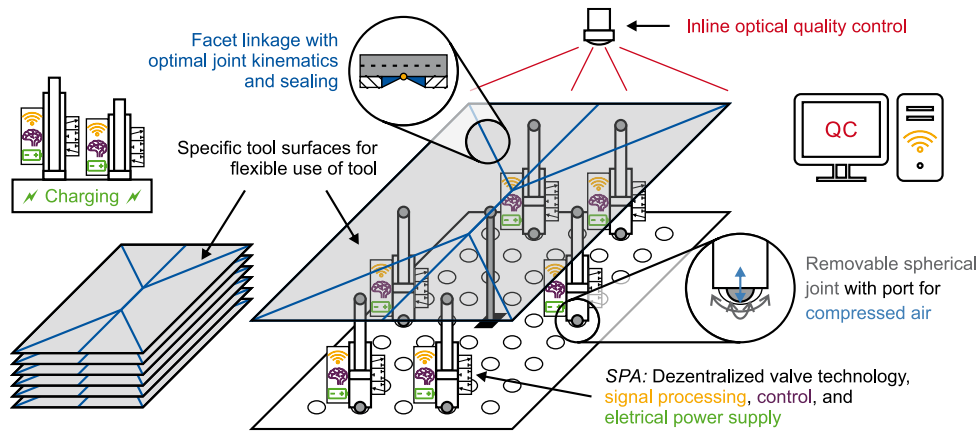


Figure 4.5. Overall concept for the next generation of the folding tool, featuring specific tool surfaces, Servo-Pneumatic Actuators, and inline optical quality control

5. CONCLUSION

The construction industry faces severe challenges to date and in near future. The global shortage in construction workers and requirements on low emissions will force battery electric mobile machinery with higher degrees of automation and new materials enabling novel construction principles with associated automated production methods into the market.

While the field of mobile construction machines is one of the leading markets for fluid power components and therefore in focus of many research initiatives, automated manufacturing of prefabricated concrete parts with significant potential for emission reductions is rarely investigated as a potential future market for fluid power. Especially, when it comes to high clamping forces, which are required for the formwork of heavy concrete parts, hydraulics is well suited to reach higher degrees of automation. Contrary to this, pneumatics can enable automation of specialized formworks and folding tools for lightweight and material efficient design and manufacturing methods. Additional research in this interdisciplinary field will be pursued at ifas.

6. ACKNOWLEDGEMENT

This work was funded by the Deutsche Forschungsgemeinschaft (DFG, German Research Foundation) – SFB/TRR 280. Project ID: 417002380. The authors would like to thank the DFG for supporting this research project.

7. REFERENCES

- [1] H. Ritchie, L. Rodés-Guirao, E. Mathieu, M. Gerber, E. Ortiz-Ospina, J. Hasell & M. Roser, ‘Population Growth’, 2023, ‘<https://ourworldindata.org/population-growth>’.
- [2] J. Jiang, S. Shi, & A. E. Raftery, ‘Mitigation efforts to reduce carbon dioxide emissions and meet the Paris Agreement have been offset by economic growth’,

Communications Earth & Environment, 823, 2025,
doi:10.1038/s43247-025-02743-x.

- [3] United Nations Environment Programme, ‘Global Status Report for Buildings and Construction 2024/2025: Not just another brick in the wall – The solutions exist. Scaling them will build on progress and cut emissions fast.’, Paris, 2025, doi:10.59117/20.500.11822/47214.
- [4] International Energy Agency, ‘World Energy Outlook 2025’, Paris, 2025.
- [5] Global Cement and Concrete Association, ‘Getting to net zero’, London, 2026, ‘<https://gccassociation.org/concretetefuture/getting-to-net-zero/>’.
- [6] L. Schmidt, K. Schmitz, T. Minav & A. Senatore, ‘Estimating the Global Emissions by Fluid Power Machinery’, Proceedings of the 15th Int. Fluid Power Conference (15th IFK), pp. 1411–1426, Aachen, Germany, March 9 – 11, 2026.
- [7] B. Sender, J. Sprink & K. Schmitz, ‘Operator effect on the performance of a wheeled excavator’, Proceedings of the 15th Int. Fluid Power Conference (15th IFK), pp. 859–873, Aachen, Germany, March 9 – 11, 2026.
- [8] J. Li, H. He, Z. Shao & Y. Ma, ‘Zero-emission construction machinery in China, 2025’, Int. Council on Clean Transportation (ID 443), Market Spotlight, September 2025.
- [9] R. Leifeld, M. Vukovic & H. Murrenhoff, ‘Hydraulic Hybrid Architecture for Excavators’, ATZ Offhighway worldwide, 9, pp. 44–49, August 2016, doi:10.1007/s41321-016-0523-9.
- [10] M. Linjama, H.-P. Vihtanen, A. Sipola, & M. Vilenius, ‘Secondary Controlled Multi-Chamber Hydraulic Cylinder’, Proceedings of the 11th Scandinavian Int. Conference on Fluid Power, SICFP’09, Linköping, Sweden, June 2 – 4, 2009.
- [11] K. Heybroek & M. Sahlman, ‘A hydraulic hybrid excavator based on multi-chamber cylinders and secondary control – design and experimental validation’, Int. Journal of Fluid Power, 19(2), pp. 91–105, 2018, doi:10.1080/14399776.2018.1447065.
- [12] Z. Xu, M. Bertolin, A. Vacca & J. Nilsson, ‘A Hydraulic Architecture Based on Multi-common Pressure Rail Principle Using Multi-chamber Cylinders for Excavators’, Advancements in Fluid Power Technology: Sustainability, Electrification, and Digitalization, GFPS 2024, pp. 339–356, September 2025 doi:10.1007/978-3-031-84505-5_22.
- [13] P. Achten, ‘Fundamentals of Hydraulic Transformers’, Proceedings of the 14th Int. Fluid Power Conference (14th IFK), pp. 794–811, Dresden, Germany, March 19 – 21, 2024, doi:10.13052/rp-9788770042222.
- [14] M. Inderelst, S. Losse, S. Sgro & H. Murrenhoff, ‘Energy Efficient System Layout for Work Hydraulics of Excavators’, Proceedings of the 12th Scandinavian Int. Conference on Fluid Power, SICFP’11, Tampere, Finland, May 18 – 20, 2011.

- [15] R. Hippalgaonkar & M. Ivantysynova, 'A Series-Parallel Hydraulic Hybrid Mini-Excavator with Displacement Controlled Actuators', Proceedings of the 13th Scandinavian Int. Conference on Fluid Power, SICFP2013, pp. 31–42, Linköping, Sweden, June 3 – 5, 2013, doi:10.3384/ecp1392a4.
- [16] L. Schmidt & M. van Binsbergen-Galán, 'On Displacement Controlled Drive Networks', Proceedings of the 15th Int. Fluid Power Conference (15th IFK), pp. 1099–1113, Aachen, Germany, March 9 – 11, 2026.
- [17] T. Pietrzyk, D. Roth, K. Schmitz & G. Jacobs, 'Design study of a high speed power unit for electro hydraulic actuators (EHA) in mobile applications', Proceedings of the 11th Int. Fluid Power Conference (11th IFK), Vol. 2, pp. 232-245, Aachen, Germany, March 19 – 21, 2018.
- [18] F. Figge & K. Schmitz, 'EHAs in mobile machinery: Experimental validation of a high-speed EHA with recuperation and load-holding capability', Proceedings of the 15th Int. Fluid Power Conference (15th IFK), pp. 1699–1715, Aachen, Germany, March 9 – 11, 2026.
- [19] H. Mu, M. Cheng & X. Tang, 'Energy efficiency evaluation of hybrid distributed-centralized load sensing system for electrified excavator', Proceedings of the 15th Int. Fluid Power Conference (15th IFK), pp. 311–326, Aachen, Germany, March 9 – 11, 2026.
- [20] R. N. Sah & P. Y. Li, 'Motion Control of a Propel Circuit Testbed for the Hybrid Hydraulic Electric Architecture (HHEA)', Proceedings of the ASME 2025 Int. Design Engineering Technical Conferences and Computers and Information in Engineering Conference. Anaheim, CA, USA, August 17 – 20, 2025, doi:10.1115/DETC2025-168419.
- [21] M. Vukovic, 'Hydraulic Hybrid Systems for Excavators', Dissertation, RWTH Aachen, Shaker Verlag, 2017, ISBN: 978-3-8440-5312-8.
- [22] Volvo CE, 'Dig Assist – Powered by Volvo Co-Pilot', Product brochure, 2023.
- [23] Cat, 'Cat® Grade Technology For Excavators', Product brochure.
- [24] Liebherr, 'Innovations in Liebherr wheel loaders: autonomous operation and hydrogen drive', Liebherr-Werk Bischofshofen GmbH, Press release, Feb.18, 2025.
- [25] S. Gerdes, J. Sprink, C. Haas, A. Opgenoorth & K. Schmitz, 'VR-control for teleoperation of mobile machines: A review on the practical implementation', at–Automatisierungstechnik, 71(11), pp. 928–938, 2023, doi:10.1515/auto-2023-0027.
- [26] PERI, 'VCT Composite Track – Composite Bridges: We go Underneath!', PERI S.E., Product brochure, 2025.
- [27] Doka, 'Super Climber SCP', Doka GmbH, Product brochure, 2013.
- [28] B. Beckmann, J. Bielak, S. Bosbach, et al., 'Collaborative research on carbon reinforced concrete structures in the CRC/TRR 280 project', Civil Engineering Design, 3(3), pp. 99–109, 2021, doi:10.1002/cend.202100017.

- [29] B. Beckmann, V. Adam, S. Marx, R. Chudoba, J. Hegger & M. Curbach, 'Novel Design Strategies for Material-Minimized Carbon Reinforced Concrete Structures – Overview of the Research in CRC/TRR 280', Proceedings of the fib Symposium 2023 – Volume 2, pp. 1242–1251, Istanbul, Turkey, June 5 – 7, 2023, doi:10.1007/978-3-031-32511-3_127.
- [30] M. Curbach, M. Hegger, J. Bielak, C. Schmidt, S. Bosbach, S. Scheerer, et al., 'New perspectives on carbon reinforced concrete structures – Why new composites need new design strategies', Civil Engineering Design, 5(5–6), pp. 67–94, 2024, doi:10.1002/cend.202200008.
- [31] D. Ehlig, F. Schladitz, M. Frenzel & M. Curbach, 'Textilbeton – Ausgeführte Projekte im Überblick', Beton- und Stahlbetonbau, 107(11), pp. 777–785, 2012, doi:10.1002/best.201200034.
- [32] M. Curbach, J. Hegger, F. Schladitz, M. Tietze & M. Lieboldt, 'Handbuch Carbonbeton – Einsatz nichtmetallischer Bewehrung', Berlin, Ernst & Sohn, 2023, doi:10.1002/9783433608661.
- [33] W. Seifert & M. Lieboldt, 'Ressourcenverbrauch im globalen Stahlbetonbau und Potenziale der Carbonbetonbauweise', Beton- und Stahlbetonbau, 115(6), pp. 469–478, 2020, doi:10.1002/best.201900094.
- [34] C. G. Gomes, S. Stüttgen, M. Weiß, R. Akpanya, A. C. Niemeyer, D. Robertz & R. Chudoba, 'Topological interlocking assemblies based on origami inspired carbon-reinforced concrete waterbomb modules', Proceedings of the fib Symposium 2025, pp. 2867–2877, Antibes, France, June 16 – 18, 2025, doi:10.5281/zenodo.18267005.
- [35] H. Spartali, J. D. van der Woerd, J. Hegger & R. Chudoba, 'Stress redistribution capacity of textile-reinforced concrete shells folded utilizing parameterized waterbomb patterns', Proceedings of the IASS 2022 Symposium affiliated with APCS 2022 conference, pp. 96–106, Beijing, China, September 19 – 22, 2022, doi:10.5281/zenodo.10812858
- [36] C. G. Gomes, C. Bertram, O. Reinertz, K. Schmitz & R. Chudoba, 'Flexibility and precision: Manufacturing concept for folded tessellated lightweight carbon-reinforced concrete slabs', Proceedings of the fib Symposium 2025, pp. 2682–2693, Antibes, France, June 16 – 18, 2025, doi:10.5281/zenodo.18267538.
- [37] J. D. van der Woerd, C. Bonfig, J. Hegger & R. Chudoba, 'Construction of a vault using folded segments made out of textile reinforced concrete by fold-in-fresh', Proceedings of the IASS Annual Symposium, Hamburg, Germany, September 25 – 28, 2017.
- [38] S. Bhat, K. Heins, C. Bertram, O. Reinertz, K. Schmitz & T. Gries, 'Ein Ausblick auf das Zusammenspiel von Faltprozess und Drapierbarkeit der Textilbewehrung', Beton- und Stahlbetonbau, 119(12), pp. 925–935, 2024, doi:10.1002/best.202400062.

Novel Poppet Valve for Digital Hydraulic Transformer

Elton Bishop

Purdue University Fort Wayne, ebishop@pfw.edu

Abstract.

Need for an efficient and robust hydraulic transformer for secondary controlled drives is well established. The 4-bit, linear-acting Digital Hydraulic Transformer (DHT) approach has given promising results, but in development of more advanced functionality, valves failed due to high demands. No valve was found that met requirements, so effort was undertaken to design a valve, specifically for DHT, from first principles, that shifts reliably in extreme conditions. This novel poppet valve is conceived, manufactured, tested and evaluated as potential DHT Bit-1 valve. Apparatus is set up to establish p-Q curve, and to measure C_v and leakage. As well, apparatus is set up to explore critical flow and response at critical flow but found to be inadequate for the flows and pressure spikes involved. Valve p-Q curve and leakage are applied to existing DHT model and impact on performance is examined, including efficiencies and End Of Stroke strategies.

Keywords. Digital hydraulic, hydraulic transformer, 2/2 valve, balanced, double piloted.

1. INTRODUCTION

1.1. Background

It is well known that hydraulic transformers operating on a constant pressure net can eliminate many losses associated with conventional hydraulics and increase the overall efficiency of machines. First, they eliminate valve losses. Second, they allow energy regeneration from aiding and braking loads. Third, the secondary controlled architecture allows the prime mover to operate at it's point of highest efficiency. The main reason hydraulic transformers are not generally used is the market is still waiting on a winning design¹⁾.

The Digital Hydraulic Transformer (DHT) concept was first disclosed in a U.S. patent application in 2005²⁾, and first presented academically at the Eleventh Scandinavian International Conference on Fluid Power, Linköping, Sweden, in 2009³⁾. The first technology of its kind, it represented the nexus of secondary controlled drives and parallel channel digital hydraulics, both of which were still new technologies, with relatively low industry acceptance, at that point^{4) 5)}.

The central concept of the DHT involves dividing the working area of a piston into axially-offset, binary weighted annular areas, Fig. 1. Selectively pressurizing the individual annular chambers results in a cumulative output force that can be incrementally raised or

lowered, which is essentially the ability to change piston size on the fly. With just four bits, this digital piston is capable of a wide range of cumulative force output illustrated by Fig. 2.

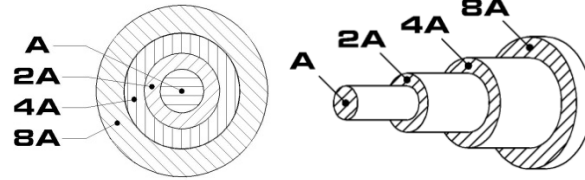


Figure 1. Axially-offset binary-weighted, areas



Figure 2. Range of four-bit system output

Inside the DHT is a special displacer piston, the transtatic bridge (TSB), which is two of these digital pistons oriented back to back. A matrix of 2/2 valves controls which of three pressures acts on each individual area. Thus, it is possible to vary displacement on both sides such that hydraulic power transformation according to discrete ratios may occur. In the case of the 4-bit DHT, a total of 143 unique transform ratios are possible when all combinations are used. Regardless the output state, the product of input pressure and flow equals the product of output pressure and flow, as shown in Equation 1.

$$Q_{in} \times p_{in} = Q_{out} \times p_{out} \quad (1)$$

Fundamentally, the DHT combines high pressure fluid, low pressure fluid and transform fluid in specific volume ratios to vary output pressure and ultimately displace the fluid volume required by the actuator. Figure 3 shows a simplified DHT schematic.

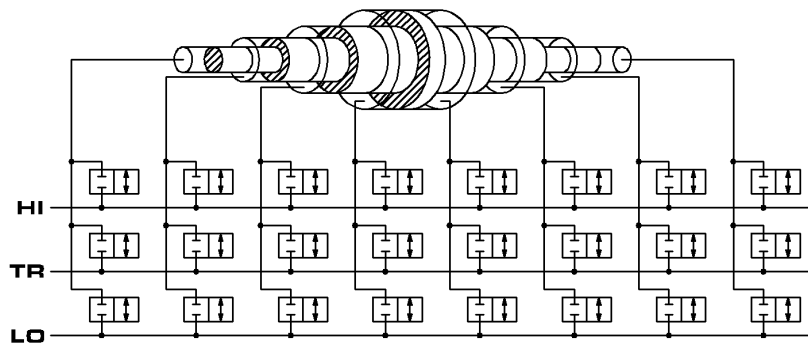


Figure 3. 4-bit DHT schematic

The TSB travels axially in its housing during transformation. When it nears its mechanical travel limit the pressures applied to each side of the TSB are exchanged end for end so that

transformation can continue, but with the TSB now moving in the opposite direction. By these reciprocation means unlimited flows are made possible.

During this End of Stroke (EOS) event, output flow and pressure momentarily drop to zero and then the entire system is accelerated back towards steady state. This happens quickly enough that, in systems with non-negligible mass, inertia damps velocity fluctuation, but it cannot completely eliminate it. While this pressure and flow discontinuity may be acceptable for some applications not requiring fine control, such as bulk transfer, most applications are not and so work has gone into developing strategies to combat and minimize pressure and flow fluctuation at EOS.

One approach to mitigate the EOS output fluctuation is to inject high pressure fluid directly into the transform rail to supplement transform flow while bridge changes direction. This could be accomplished using a 2/2 servo valve plumbed between high rail and transform rail, but with 24 valves already, it is the view of the author that the DHT cannot endure any more complexity and expect commercial success, so a servo valve is ruled out. As design constraint, the function must be achieved using only the bit valves already making up the transformer.

1.2. Problem

In the DHT matrix schematic architecture, flow directly from high rail to transform rail is possible through any of the eight bit rails with both HI and TR valves open. Which bit rails are chosen to provide cancellation flow depends on output state. Opportunity exists for motoring EOS cancellation in chambers that are contracting to their minimum volume at TR pressure transitioning to expanding at HI pressure. In this case the HI valve can be turned on in advance so that it is open to flow through bit rail and through TR valve into TR rail. In addition to providing cancellation flow, the additional axial force on the bridge in the time from HI valve opening to TSB velocity zero serves to accelerate the TSB in the new direction faster.

A pulse of high pressure hydraulic fluid through HI and TR valves in series can satisfy the volume requirement that is the difference between steady state DHT output and what the bridge is actually delivering. But because the volume injection time is shorter duration than the period it is compensating for the flow rate must be higher than steady state. For instance, if Bit-1 steady state is 5 gpm, instantaneous injection flow rate might be 20 gpm or higher and the valves must still close reliably.

Turning one valve on before another valve on the same bit rail turns off creates an opening overlap. The opening overlap has to be carefully controlled to inject precise amounts of high pressure fluid into the output rail to cancel EOS flow discontinuity. During this overlap valves experience pressure differences upwards of several thousand psi and flow increases sharply.

In development of this phased overlap control it was discovered that the direct acting solenoid operated spool valves of the DHTM475 transformer were being subjected to conditions beyond what they were designed to accommodate and were failing to close.

In this case, high enough Δp acting for even a few milliseconds accelerates enough flow through valves for inertial flow forces on spools to dominate return spring force and the

NC valves stay open when de-energized. In conventional systems this overflowing condition rarely or never occurs, but in the DHT application this occurs at every EOS.

Avoiding overflowing valves is especially critical for the DHT application because of the mode of failure. Unpredictability of transform ratio is not something that can be tolerated in a system with a transform ratio as high as 15x. If the wrong valve or valves overflow and pressurize, with high pressure, chambers that are to be pressurized with low pressure, the output pressure generated from a typical 3,000 psi accumulator pressure could be transformed to as high as 45,000 psi, which would cause damage to equipment and potentially harm people. This necessitates valves with reliable, consistent response, both on and off.

Demanding even more of valves, when a 2/2 valve switches at high pressure feeding a finite chamber volume through a tube, water-hammer pressure peaks can reach 2x to 3x rail pressure. Across many valves switching in coordination, the cumulative transient environment is hostile. Valves must not only survive 5,000 psi static; they must survive the dynamic peaks, and resultant instantaneous flows, created by neighbouring valves switching.

Also, conventional valves restrict flow too much. The DHT is especially sensitive to valve flow resistance because through each bit oil flows through two valves on its path between entering and exiting the transformer. Because we want the work done by the hydraulic system doing work on the load rather than internally through valves, efficient valves with low pressure drop are needed.

In addition to the overflowing and efficiency issues, the conventional spool valves used in DHTM475 exhibit an overall amount of leakage that is large enough to be of concern. Although not measured directly, overall energy expenditure due to leakage was observed to be significant. A valve with reliable, consistent response, both on and off, with low restriction and that exhibits low leakage, is clearly necessary for continued development.

Over several years, numerous distributors, applications engineers, company technical representatives, engineers and leaders from the many names in hydraulic valves from the Americas, Europe and Asia were tasked by the author with providing bidirectional 5, 10, 20 and 40 gpm 2/2 valves that can close reliably at or near critical flow.

Many valves were recommended and reviewed and several were considered, including internally piloted logic valves with high flows, but with response that is highly flow and pressure dependent. Also considered were 4/2 spool valves with combined flow and opposing flow forces, but leakage would be even worse than already with 2/2 spool valves, response would be slower and there would be extra plumbing involved. Several piloted poppet valves were considered, but pilot sections were found to be intentionally throttled for slow opening and closing, which is opposite of what is required by DHT. Servo valves were ruled out due to cost and complexity. There were no satisfactory candidates found.

1.3. Solution

After searching but finding no satisfactory results, a decision was made in August 2024 to attempt the design and manufacture of a new valve, from first principles, specifically for the DHT. The approach chosen was that of brute force: hydraulically force the poppet closed against flow forces with as large of pilot area as necessary. Ideally, the valve will

close reliably in flows all the way up to and including critical flow at burst pressure, but that is a lot to ask.

There are also downsides to increasing pilot area such as slower response and increased size that must be weighed. This is in addition to the already slower response time and overall increase in size owing to the jump from direct acting to pilot actuated valves. On the positive, because closing function is achieved hydraulically there is no need for a return spring so it may be eliminated. A poppet concept was chosen over spool due to the possibility of double blocking, minimized leakage and simplified manufacturability.

The overall concept includes a double balanced poppet valve that is continuously piloted either on or off, with return pilot piston area increasable to meet arbitrary pilot force requirement independent of poppet size or shape. No return spring is necessary because the closing function is achieved hydraulically.

Of the four valve sizes required, the smallest valve was selected to be attempted first due to it being the most difficult to design and manufacture, as well as requiring the minimum of test rig flow.

The requirements on the new valve concept are summarized as follows:

- Ability to close regardless of flow or Δp
- Poppet with 45° conical seat seal – No leakage
- Double blocking – Hydrostatically balanced at each port
- Double piloted – No spring
- Nominal flow rate: 5 gpm @ ≤ 20 psid
- 0.375"Ø flow path (for ≤ 20 l/s at rated flow)
- 800 psi pilot pressure
- Working pressure: 5,000 psi with 4:1 safety factor

In addition to the above requirements, demands on the new valve concept as follows:

- Switching time of ≤ 50 ms
- 5 Hz at 20,000 psi fatigue-proof
- Compact
- Scalable
- Higher efficiency
- Increased reliability
- Decreased electrical demand
- Low cost
- Easy to manufacture

2. JANUS VALVE

Because a valve capable of meeting the above requirements and demands must have a fitting name, this valve was given the name Janus, for the Roman god of gates, transitions, beginnings and endings. The 5 gpm Janus valve, designated JV05, shown in section in Figure 4, is conceived, manufactured, tested and evaluated as potential DHT Bit-1 valve.

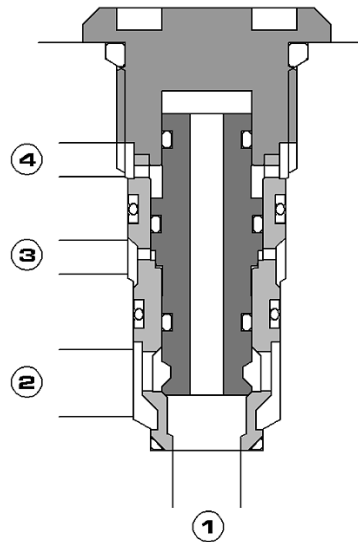


Figure 4. JV05 valve mid-line section

2.1. Design

Design of the Janus valve is an exercise in pragmatism that began with a simple cylindrical poppet having a small 45° bevel around one end. This bevel is lapped with a conical seat in the valve body and is self centering for minimized port to port leakage. The end opposite of bevel is the balance piston, which cooperates with the balance cylinder integrated into the cap. A hole for pressure equalization runs through the poppet axially from the face of one end to the face of the other end so that poppet is hydrostatically balanced. This means pressure levels and fluctuations at nose port 1 have little influence on poppet behaviour. The diameter of the balance hole may be increased for minimized fluid restriction and poppet mass.

Pressure at side port 2 acts radially on poppet and is therefore self-cancelling and balanced such that pressure and flow fluctuations at side port 2 have little effect on poppet behaviour.

In order to shift the poppet axially, on and off, a pilot piston is integrated around the poppet and chambers cooperating with pilot piston for actuation in each direction are integrated into the body. Pilot port 3 is pressurized to open the valve. Pilot port 4 is pressurized to close the valve. The area of the pilot piston, for a given pilot pressure, must be sufficient as to produce enough closing force to overcome the highest flow force the poppet can theoretically ever see. Practically, this is at critical flow at maximum input pressure.

Fluid paths in pilot section are unrestricted for fast response. Free acceleration of poppet from pilot pressure can, even over a short stroke, produce poppet velocity high enough to cause damage to poppet and/or seat at closing. To avoid damage to poppet and seat at closing, a cushion is integrated into the pilot-on chamber to decrease poppet velocity just before metal to metal impact.

This cushion traps a large portion of swept area of the pilot-on piston in a cushion chamber from which the path out is through a progressively thinner annular orifice created between poppet cushion piston and the body cushion cylinder. There is no check valve.

Instead of a check valve, the cushion piston is wide with a shallow, short stroke such that it acts a short distance with high pressure. At the beginning of valve opening, because initial flow into pilot-on chamber, rising from zero, is low, the annular orifice restriction is of little influence. Additionally, the orifice becomes less restrictive as the poppet moves away from the seat, which allows progressively greater flow over the few thousandths of an inch the cushion acts on.

Because of the short stroke, the poppet can move just a small distance to be out of range of cushion. This arrangement allows the cushion to sufficiently brake the poppet at closing yet not significantly impede immediate response upon opening. In addition, the check-valve-less cushion design allow a cleaner, more compact, less complex overall unit compared to a design with check valve.

Another cushion is designed into the balance chamber for decelerating poppet at the end of opening that is even simpler and shorter stroke than the closing cushion. Rudimentary but effective, two broad, flat faces coming into contact in oil squeeze oil out from between them at progressively higher pressure, which acts on and quickly brings to rest the poppet.

The Janus valve and its cavity were designed as one. The cavity is designed to be machined with standard tooling and require no form tool. One particular departure from standard valve cavities is its flat bottom at a specific depth.

The seal around the nose of the valve, separating port 1 from port 2, is of corner crush design. An o-ring is placed in the flat bottom of the cavity before valve is inserted. Assembly compresses the o-ring in the corner between valve nose and the flat bottom of the valve cavity. Axial dimensions of the sleeve and cap are held closely so that the nose just touches bottom at full torque. This corner seal is chosen over the traditional o-ring with back up rings piston seal due to the elimination of the requirement to fasten the valve's body parts. With no threads, pins, wires or other method of fastening the two body parts, the overall size and mass of the valve, as well as manufacturing effort, is minimized. The parts are simply held in compression by the o-ring. In addition to minimized complexity, the corner compression seal allows a shorter overall installed length.

All three valve components are designed to be chucked in the lathe just once to machine all their critical features, which allows closer concentricity and greater precision due to everything turning about the exact same axis. Only non-critical features are machined in secondary operations.

2.2. Pilot valve

The SV08-40 4-way, 2-position solenoid operated spool valve from Hydraforce was selected to pilot the JV05 prototype. It handles a maximum flow of 3 gpm at pressures up to 3,000 psi. With this SV08-40/JV05 arrangement it is possible to configure the valve to be normally closed or normally open by switching the pilot line connections.

2.3. Prototypes

A prototype JV05 valve, Figure 5, was manufactured on a manual lathe with standard tooling. Each of the three pieces was chucked just once to machine all of their critical features. Of note is that after the body is roughed out, a drill on the tool post comes in radially to drill cross holes, which allows the inside to then be finished, eliminating any burrs left from drilling, while never un-chucking the part.

A body to accommodate both the JV05 and SV08-40 as one unit, shown in Figure 6, was manufactured on a manual Bridgeport mill with standard tooling. This is the combination used for testing and characterization.



Figure 5. JV05



Figure 6. Valve body

2.4. Flow

Effort was put into ensuring valve flows freely. As a starting point, both ports are sized to match nominal size fluid path in block. In this 5 gpm instance the diameter is .375", which keeps fluid velocity just under 20 feet per second at rated flow. This flow area is maintained or exceeded throughout the entire flow path.

Flow through piloted open valve was simulated in SolidWorks, with assistance from Dr Donald Mueller and Dr Zhuming Bi in the Civil and Mechanical Engineering Dept. of Purdue University Fort Wayne, to verify flow forces and estimate critical flow. A flow trajectory simulation of critical flow from port 2 to port 1, at 5,000 psi pressure difference, Figure 7, shows no critical point at which pressure suddenly changes. Pressure drop is fairly consistent over the entire fluid path through valve.

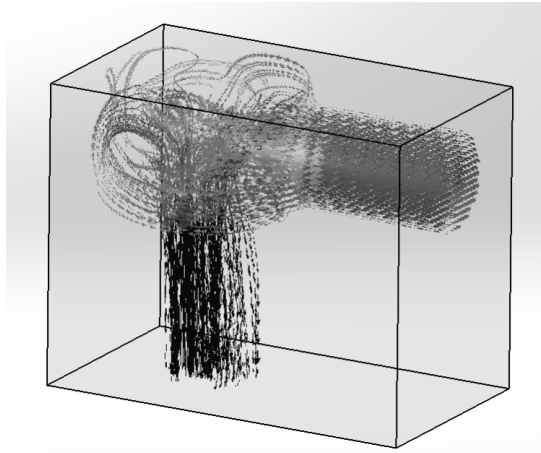


Figure 7. Flow Simulation

2.5. *p-Q Curve*

To relate flow through valve and port to port pressure differential, JV05 was plumbed to a hydraulic test stand equipped with power supply, reducing valve, variable area flow meter, gauge port test cells, pressure gages and differential pressure gages. Flow meter and differential pressure gages shown in Figure 8. A needle valve plumbed between output port and tank served as variable load. By varying needle valve, flow through valve was adjusted and readings of flow and pressure differential were recorded. JV05 p-Q curve, flow 1 to 2, with ISO 32 oil around 65°F, is given in Figure 9.

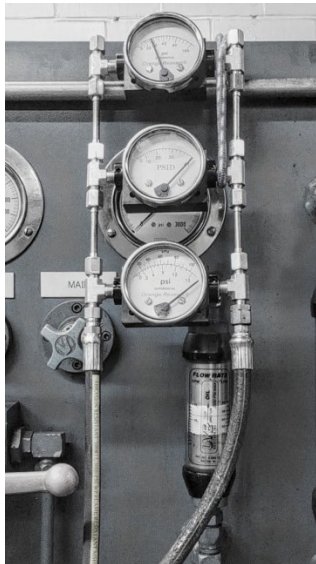


Figure 8. Test rig

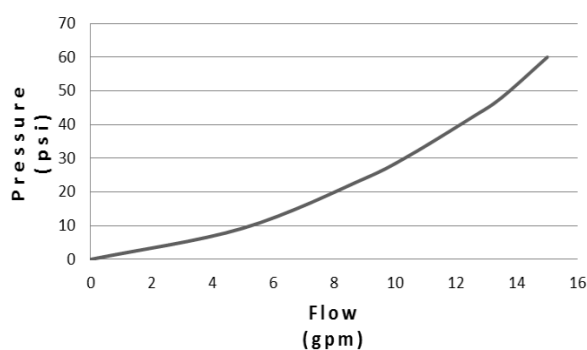


Figure 9. Pressure differential vs. flow, flow 1 to 2

With measured pressure differential of 9 psi at 5 gpm, the design goal of ≤ 20 psi at 5 gpm was clearly surpassed. 15 gpm is 3x nominal flow.

2.6. C_v

The valve coefficient, C_v , was measured by applying exactly 1 psi water pressure at the inlet port while the valve was piloted open with outlet port open to atmospheric pressure.

Water supply was adjusted to achieve exactly 1 psi, read from a new, quality 0 - 3 psi gauge plumbed at valve inlet port. This setup is shown in Figure 10. Once flow and pressure were observed to be correct and steady state, output flow was directed into a level, graduated container and a stopwatch was simultaneously started. At precisely 60 seconds, flow was directed away from the bucket. Amount of water in the container was read and recorded.

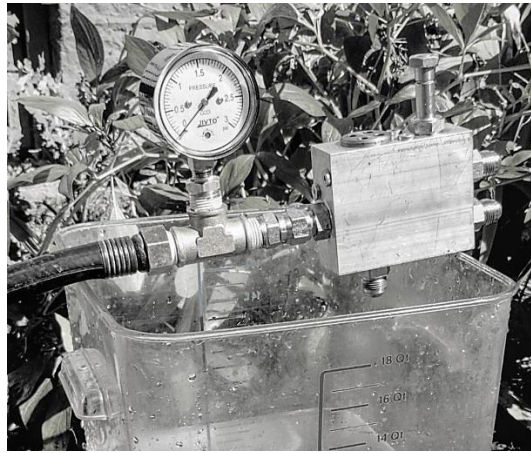


Figure 10. Cumulative flow and pressure measurement apparatus

This experiment was carried out twice in each direction of flow. With volume resolution of the container only down to the quart, volumes of water collected were interpolated using digital calipers measuring between surface of water and the closest graduations.

The average amounts of water that flowed through the valve in one minute are recorded as follows:

$$C_v^{1-2}=2.35 \text{ gpm}$$

$$C_v^{2-1}=2.20 \text{ gpm}$$

2.7. *Critical Flow*

Critical flow through a hydraulic valve is the maximum flow rate achievable for a given upstream pressure, which occurs when the downstream pressure drops to roughly 50 - 55% or less of the upstream pressure. In this state, reducing the downstream pressure further will not increase the flow rate. This critical flow in hydraulic valve is due to cavitation or flashing.

In order to fully understand critical flow one must understand the pressure distribution throughout the valve. Typically, flow through hydraulic valves must get past a main restriction, directly after which is a point of maximum fluid velocity and minimum pressure called the vena contracta, and in these cases standard equations exist to relate

pressure drop, pressure recovery, specific gravity, C_v , vapour pressure, etc. However, through the Janus valve there is no such point of restriction, which makes prediction of critical flow difficult.

An attempt at achieving and measuring critical flow through the Janus valve was made using the stopwatch, graduated barrel and pressure gages at hand. Operating on combined flow from full accumulator and 18 hp prime mover input, with input pressure held constant by reducing valve, it was possible to make 20 second runs without running out of adequately pressurized fluid. With input pressure of 1,000 psi, JV05 was found to be flowing around 52 gpm at around 600 psid. Past this point further reduction in outlet pressure did not visibly or audibly increase flow, so there is some belief this was in or at critical flow. However, it was not possible to verify this numerically in real-time with the equipment being used. Due to the difficulty of returning the fluid back to tank and the mess it made the experiment was performed only once. Better flow meters, as well as a higher flow test bench, are needed to verify and characterize the critical flow behaviour. Because of this uncertainty, it cannot be claimed that critical flow was achieved or observed with publication-level confidence.

It should be noted that even at this extreme flow point the valve opened and closed reliably the eight times it was cycled. This would have been explored further but the jumping, jarring and flexing of the test rig from pressure spikes at switching presented danger too great to ignore. At minimum, accumulators must be fitted to each port and the low pressure return line replaced with larger one of higher pressure rating before any additional exploration of critical switching is attempted.

2.8. Response

In order to understand dynamic aspects the valve was connected to test rig with input pressure held constant by a reducing valve and flow controlled by a needle valve. A pressure sensor was fitted between valve output and needle valve. Plumbing volume between valve, sensor and needle valve was kept to a minimum to minimize compression effects and wave propagation issues. A pressure sensor was also applied to each of the pilot lines between pilot valve and main stage. All three pressure sensors are Omega PX613 thin film transducers with a published 5 ms output delay.

A Dataq DI-1100 4-channel data recorder and Windaq software was used for data acquisition. The following images, Figures 11 - 14, show traces of command signal, pilot-on chamber pressure, pilot-off chamber pressure and main stage output pressure over time. Input pressure was held at 1,000 psi, flow was adjusted to 5 gpm and pilot pressure was set at 800 psi. Each channel recorded 1,000 samples per second. Time between divisions is 2 ms. When interpreting the traces it is important to remember that while the command signal trace in channel 1 gets its input directly from pilot valve solenoid wiring, and is thus an immediate representation of state, the three pressure traces are delayed by 5 ms due to sensor limitations.

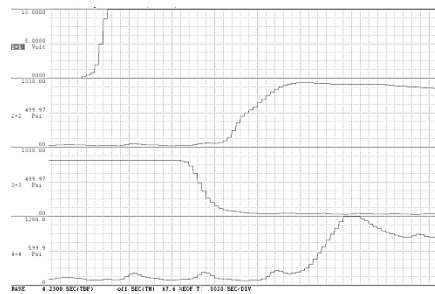


Figure 11. Off-to-On, Flow 1-2

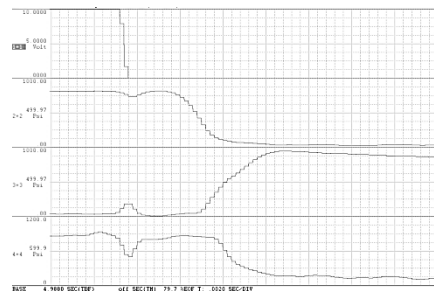


Figure 12. On-to-Off, Flow 1-2

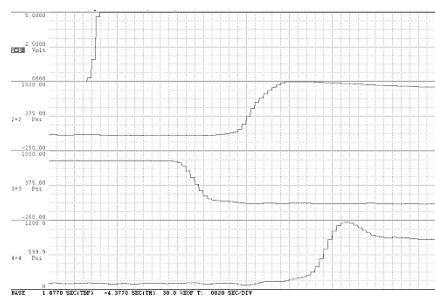


Figure 13. Off-to-On, Flow 2-1

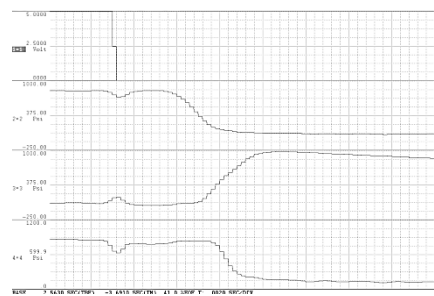


Figure 14. On-to-Off, Flow 2-1

With flow 1 to 2, transitioning Off-to-On, Fig. 11, the first indication of change after the command signal goes high is the release of pressure holding the poppet closed in the pilot-off chamber, which starts at around 14 ms. This pressure falls to its minimum at around 24 ms. Within this timeframe the pilot-on chamber pressure begins to increase starting at around 22 ms and reaches its maximum at around 36 ms. The changes in pilot chamber pressures act on the pilot piston areas causing the poppet to open. While somewhat difficult to disambiguate from the noise in channel 4 data, the first indication of main stage opening is at around 40 ms. Center crossing occurs at around 47 ms.

When the pilot valve is de-energized, Fig. 12, the flyback voltage momentarily affected pilot and output pressure readings so those fluctuations should be ignored. Pilot-on pressure begins to fall at 5 ms and reaches its minimum at around 20 ms. Within that timeframe pilot-off chamber pressure starts to increase at around 13 ms and reaches its maximum at around 28 ms. Changing pilot pressures force the poppet closed. Output pressure begins to decrease at around 15 ms and center crossing occurs at roughly 20 ms.

With flow 2 to 1, transitioning Off-to-On, Fig. 13, performance is similar to opening with flow 1 to 2, but with a center crossing at around 48 ms.

Upon closing, Fig. 14, performance is again similar to that of flow 1 to 2, but with a center crossing at around 22 ms.

With opening response of 47 or 48 ms, and closing response of just 20 or 22 ms, the response of this JV05/SV08-40 valve combination meets the goal of 50 ms.

But while a fast response is good, more important is consistency, reliability and a fast transition. The actual response time is not all that critical because, as EOS is initiated in control well before metal to metal impact in the bridge, the timing of each valve can be determined in advance of execution, and signals to valves with the longest delay are sent first, valves with shortest delay, last, such that the crossings can be timed to millisecond precision regardless of a magnitude of valve response delay.

2.9. *Leakage*

When the prototype JV05 valve was first assembled and an attempt was made to assess its port-to-port leakage, not even one drop of leakage was observed, with 3,000 psi at either port, over any time period. Of course, this was with a new precision lapped poppet seat with no damage.

After hours of operation, testing and pushing physical limitations, valve port-to-port leakage was measured again by pressurizing one port of the piloted off valve and leaving the other port open to atmospheric pressure and directed into a beaker. With input pressure at port 1 held constant at 3,000 psi, 38 drops were observed to leak from port 2 over a 10 minute period. With pressure applied at port 2, 34 drops in ten minutes leaked from port 1. Expressed with a single digit of precision over a minute, this equates to:

$$Q_{\text{leak}}^{1-2} = 4 \text{ drops/min}$$

$$Q_{\text{leak}}^{2-1} = 3 \text{ drops/min}$$

While this is more leakage than desired it is still less leakage than through the comparable direct acting spool valves the DHT has used up to now. Cushion design will continue to improve to better protect poppet and seat from damage at closing, and it should be possible to decrease leakage with improved geometry, material specification, machining, lapping and fluid cleanliness.

2.10. *DHT Model*

To understand how the Janus valve can impact DHT performance, an Excel-based, semi-empirical DHT model, originally reported in the Fifty-third National Conference on Fluid Power (NCFP'14) paper⁶⁾, was updated with actual JV05 p-Q and leakage data. The original model was tuned to third party testing data of the DHTM475 and the only things changed in the model are valve p-Q relationships and leakage rates. Everything about the bridge and tubes remained unchanged.

Modeled schematic of digital hydraulic transformer with double piloted, double blocking poppet valves and 4/2 solenoid pilot valves is shown in Figure 15.

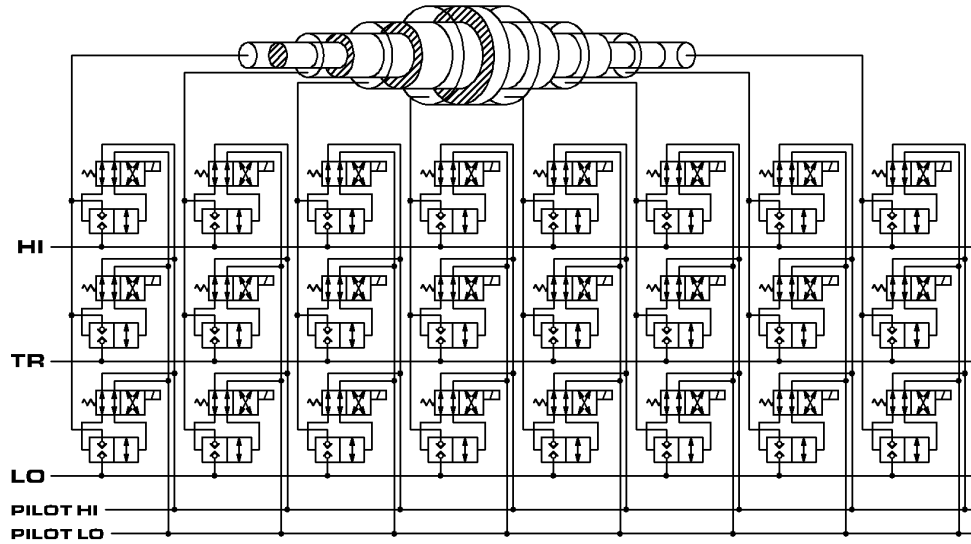


Figure 15. DHT schematic with Janus valves

2.10.1. Mechanical Efficiency

Mechanical efficiency at a given flow and pressure is mainly dependent upon the pressure drops across valves and sliding friction of the transtatic bridge. Beginning assumptions were made:

- p-Q curves, 1 to 2 and 2 to 1, are identical.
- p-Q curves of 10 gpm, 20 gpm and 40 gpm valves are identical to JV05 when scaled to nominal flow.
- Accumulator pressure is fixed at 3000 psi.
- Static friction is not considered in this model.
- Seal friction is estimated using a two-dimensional, second order, lumped parameter model, with input parameters Δp and TSB velocity.
- Coefficients are tuned manually to match experimental data.

The mechanical efficiency map of the DHT with Janus valves is given in Figure 16. The main horizontal axis scale is flow rate from zero to rated flow (75 gpm), the depth axis scale is output pressure from zero to maximum (accumulator pressure) and the vertical axis scale is percentage efficiency from zero to 100%.

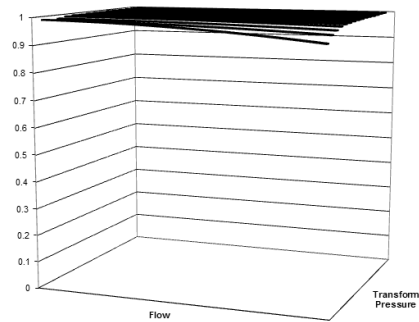


Figure 16. Mechanical Efficiency of DHT

As can be seen in Fig. 16, mechanical efficiency is well above 90% throughout the entire operating range. This is marked improvement over performance with previous valves. Only at lower transform ratios and higher flows does the mechanical efficiency decrease toward 90%. Worst case corresponds to maximum flow at zero pressure. It should be noted, however, that at and around this extreme operating point there is little power transmitted, so lower efficiency is of little consequence.

2.10.2. Volumetric Efficiency

Volumetric efficiency at a given flow and pressure is mainly dependent upon valve leakage and leakage past seals in the transtatic bridge. Beginning assumptions were made:

- Seal leakage is assessed using a two-dimensional, second order lumped parameter model.
- The input parameters are Δp and velocity.
- Coefficients are tuned manually to match manufacturer data.
- Pilot valve leakage is not considered.

The volumetric efficiency map of the DHT with Janus valves is given in Fig. 17. The main horizontal axis scale is flow rate from zero to rated flow (75 gpm), the depth axis scale is output pressure from zero to maximum (accumulator pressure) and the vertical axis scale is percentage efficiency from zero to 100%.

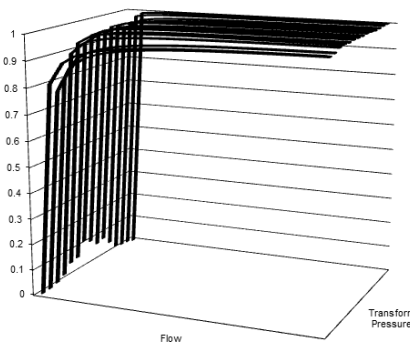


Figure 17. Volumetric Efficiency of DHT

As can be seen in Fig. 17 volumetric efficiency of the DHT is above 90% for the majority of the operating range and over 95% for at least half of it. This is improvement over previous design. Only at low flow conditions does the volumetric efficiency decrease, with worst case at zero flow. It should be noted that at and around zero flow there is very little power transmitted so lower efficiency is of little overall consequence.

2.10.3. Overall Efficiency

Multiplying mechanical efficiency by volumetric efficiency yields overall efficiency. This overall efficiency is also understood as a ratio of power out divided by power in. The modelled overall efficiency map of the DHT with Janus valves is given in Fig.18. The main horizontal axis scale is flow rate from zero to rated flow (75 gpm), the depth axis scale is output pressure from zero to maximum (accumulator pressure) and the vertical axis scale is percentage efficiency from zero to 100%.

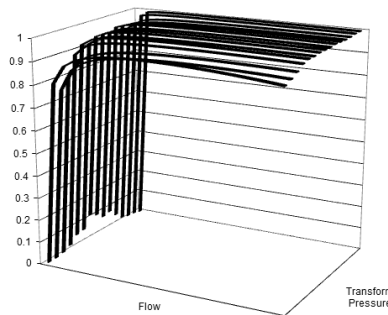


Figure 18. Overall Efficiency of DHT

As can be seen in Fig. 18 overall efficiency for the majority of the output range is above 90%, and over half the output range is above 95%. This is significant improvement over previous design with spool valves. At lower transform ratios and both higher and lower flow conditions the overall efficiency decreases slightly towards 90%, and, as is the case with volumetric efficiency, overall efficiency falls to zero at zero flow. It should again be noted that little power is transmitted through the system at and around these extreme points, so lower efficiency is of less consequence to the system overall. This is especially true at low flows.

2.11. Impact on EOS and DHT

The Janus valve's ability to turn off at high flow means there is potential for it to reliably support direct rail to rail flow at EOS in the DHT application. So long as valves continue to respond to commands in a consistent way, development of this EOS approach can continue.

In addition to its functionality, the Janus valve's low restriction means lower mechanical losses and higher overall efficiency in the DHT.

Beyond this, because every detail of the valve is aimed at solving a problem for just one application, and because all of the design decision-making and manufacturing are

collocated and in parallel with DHT development, potential exists that the Janus valve truly meets a need that no other valve can.

3. CONCLUSION

The JV05 double piloted, balanced, 2-way, 2-position poppet valve, designed specifically for the 4-bit, linear-acting Digital Hydraulic Transformer, is conceived, manufactured, tested and evaluated as potential DHT Bit-1 valve. The valve is designed around a simple cylindrical poppet that is actuated in each direction by separate pilot pressures. Because the closing function is achieved hydraulically there is no need for a return spring so it is omitted. An integrated cushion slows the poppet just before impact with seat to eliminate damage to poppet and seat.

One departure from conventional cartridge valve form is a corner nose seal that holds parts in compression while assembled in their cavity. Another departure is that the flow area through the valve is greater than through lines, which is opposite most valves.

All of the requirements are met with a 3-piece design, and each component is designed for ease of manufacture.

Apparatus was set up to establish p-Q curve, which included achieving a differential pressure of just 9 psi at 5 gpm flow, measuring a favorable C_v and measuring a small amount of leakage. As well, apparatus was set up to explore critical flow and response at critical flow but found to be inadequate for the flows and pressure spikes involved.

Opening response delay was found to be around 48 ms, while closing response was found to be around 22 ms. This overall response of the JV05/SV08-40 valve combination satisfies the goal of 50 ms.

Valve p-Q curve and leakage data were applied to existing DHT model and impact on performance was examined, including improved efficiencies and End Of Stroke strategies over previous DHT iterations with conventional spool valves. The model indicates possibility of overall efficiency over 90% for the majority of the operating range. However, the most critical aspect is the valve's ability to shift reliably at high flows because it is required for successful rail to rail flow at EOS.

There is also potential for the Janus valve to find application outside the DHT, as a general purpose, configurable, bi-directional, two-way, two-position directional control valve, operating in either a normally open or normally closed state, to start, stop or isolate fluid flow in industrial and mobile hydraulic systems. While there are many 2/2 valves on the market already with excellent performance and low cost, how many can close reliably at critical flow? Especially where mode of failure is catastrophic and/or a mission is critical, the Janus valve aims to perform as commanded regardless any condition thrown at it.

While the results of testing so far are promising, there is a large amount of work still needed to bring the JV05 forward in development in areas such as:

- geometry optimization
- material specification
- fatigue testing

- proof/burst testing
- critical performance characterization
- development of 10 gpm JV10

The Janus valve, a bidirectional, double piloted, balanced poppet valve, with its conical seat and integrated cushion, appears to be a decent option to pursue as part of DHT development going forward. The valve's ability to close at high flow sets it apart from others. Because the Janus valve is so far proving it can close even at high flows, it appears development of DHT EOS functionality may continue.

In addition to the benefit of being able to close, with pressure drop just a fraction that of equivalent nominal flow valves, it appears the Janus valve will allow the DHT higher mechanical efficiency throughout the operating range than was possible previously. And with leakage just a fraction that of previous spool valves, the Janus valve should also allow the DHT much higher volumetric efficiency throughout the operating range. With both higher mechanical efficiency and higher volumetric efficiency the overall efficiency can only be higher as well. These are results of a semi-empirical DHT model using measured JV05 data.

4. REFERENCES

- [1] Achten, P. "Fundamentals of Hydraulic Transformers" 14th International Fluid Power Conference (14.IFK), Dresden, Germany, 2024
- [2] Bishop, E. "Digital Hydraulic System" U.S. Patent Application Serial No. 60/740345, U.S. Patent and Trademark Office, 29 November 2005.
- [3] Bishop, E. "Digital Hydraulic Transformer – Approaching Theoretical Perfection in Hydraulic Drive Efficiency" 11th Scandinavian International Conference on Fluid Power (SICFP'09), Linköping, Sweden, 2009
- [4] Kordak, R. "Hydrostatic Drives with Control of the Secondary Unit" Hydraulic Trainer, Vol. 6, Bosch Rexroth AG, 1996
- [5] Linjama, M. "Energy Saving Digital Hydraulics" Second Workshop on Digital Fluid Power, Linz, Austria, 2009
- [6] Bishop, E. "DHTM475: The Evolution of Flow" 53rd National Conference on Fluid Power (NCFP'14), Las Vegas, Nevada, USA, 2014

Unsteady CFD Analysis of Alternating Flow Patterns in a Restrictive Orifice of a Pressure Regulator Valve

Naoki Hirose¹, Seiei Masuda¹, Yasushi Matsunaga¹, Fumio Shimizu²,
Kazuhiro Tanaka², Takeshi Yamaguchi²

¹ IHI Corporation, Control Systems Engineering Department, 3-2-1 Daikanyama,
Akishima City, Tokyo, Japan*

² Kyushu Institute of Technology, Graduate School of Computer Science and Systems
Engineering, 680-4 Kawazu, Iizuka-city, Fukuoka, JAPAN**

E-mail: hirose3694@ihi-g.com; masuda4361@ihi-g.com; matsunaga4536@ihi-g.com;
shimizu@phys.kyutech.ac.jp; tanakafa@fka.att.ne.jp; aw.fluidlab@gmail.com

Abstract.

The damping force generated by the damping orifice installed in a pressure-regulating valve is regarded as a key factor governing the overall stability and dynamic response of the valve. However, this damping force varies in a complex manner depending on the local flow field around the orifice, the amplitude of the volumetric flow rate, and the excitation frequency, and its fundamental mechanisms have not yet been fully clarified. To elucidate these mechanisms, three-dimensional unsteady CFD analyses were performed for the flow around the damping orifice within a pressure-regulating valve system, and the resulting flow patterns were examined in detail. The analyses revealed that, under unidirectional-flow conditions, a highly unsteady vortical structure is formed in which starting and stopping vortices are generated and subsequently collapse, whereas under alternating-flow conditions, distinctly different flow fields appear on the upstream and downstream sides of the orifice. This paper reports on these characteristic flow-field structures.

Keywords. Pressure Control Valve, Damping Orifice, Alternative Flow, Flow Pattern, CFD

1. INTRODUCTION

The configuration of the pressure regulator system is shown in Figure 1.1. In the pressure regulator valve, which operates with the pre-compression of the feedback spring as the input and regulates the outlet pressure to remain constant against variations in flow demand, the damping force generated by the damping orifice is extremely important for ensuring system stability. The system damping directly changes the phase margin of the valve spring-mass dynamics; increasing the damping force enlarges the phase margin and thereby enhances stability. On the other hand, excessive damping causes velocity saturation of the valve. Once the valve motion is velocity-saturated, its dynamic response becomes insufficient over the operating range, and therefore the trade-off between stability and responsiveness becomes critical. Research on orifice flows has a long history, particularly regarding discharge coefficients under steady-flow conditions [1]. Subsequently, transient flow and system dynamics in piping systems without orifices have been investigated [2,3], and macroscopic

studies have also been conducted for piping systems that include orifices [4]. In Reference [4], the dynamic characteristics of orifices are reported in detail, and it is concluded that,

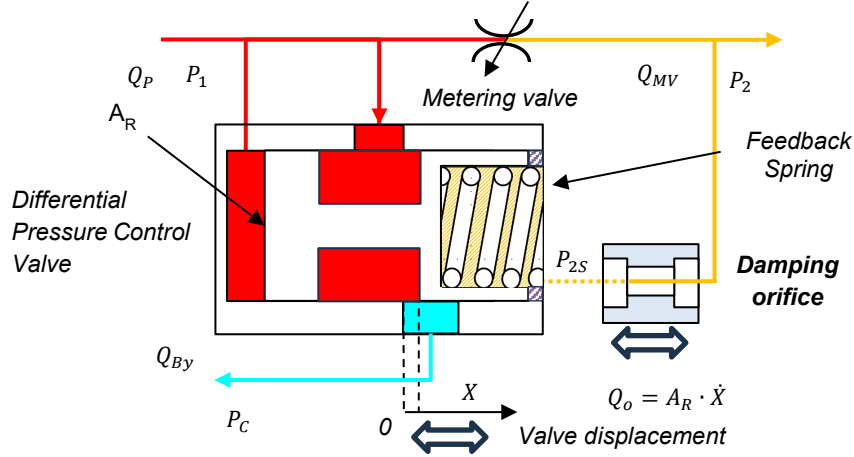


Figure 1.1 Schematic Diagram of a Pressure Regulator Valve

for excitation frequencies below 1000 [Hz], unsteadiness has no significant influence on the flow characteristics. In addition, there are interesting studies that attempt to derive the fluid inertia associated with constrictions in piping [5]. Because inertial and resistive effects of the fluid occur simultaneously and are difficult to separate, further work has combined CFD results with lumped-parameter model to extract the inertial contribution of the orifice section [6]. From a more microscopic standpoint, wall-friction losses in pulsating flows have been examined [7,8], but no study has addressed the combined effect of pulsating flow and an orifice. In view of the above, very few studies have investigated the frequency response and flow patterns of damping orifices as they appear in pressure regulator systems. Masuda et al. [9] compared unsteady CFD results with a one-dimensional bond-graph model [10] and showed that the damping characteristics of an orifice subjected to alternating flow can be represented by a second-order lag system with an additional first-order lead element, and that the resonance frequency decreases relative to the value predicted by the 1D model. They further predicted [11] that this reduction in resonance frequency is caused by two mechanisms: (i) added mass that is carried together with the fluid column in the orifice bore, and (ii) a pulsating jet that temporarily blocks the inlet and outlet of the orifice under alternating-flow conditions, deflecting the flow by 90° from the axial to the radial direction and thereby forming a flow field analogous to the curtain area in a nozzle-flapper mechanism, which enhances the inertial effect. However, their paper does not discuss in detail the relationship between excitation frequency and the associated flow patterns. Oshima et al. [12] and Miyashita [13] reported experimental results on the frequency response of damping orifices but did not address the internal or external flow structures. Loudon et al. [14] analytically solved the unsteady Navier–Stokes equations for one-dimensional laminar flow between parallel plates driven by a sinusoidal pressure gradient and examined in detail the dependence of the velocity profile, volumetric flow rate, and wall shear stress on the Womersley number [15], including changes in profile shape, amplitude, and phase lag. They demonstrated that, similar to circular pipe flow, quasi-steady behaviour occurs for $Wo < 1$, whereas phase lag and distortion of the velocity profile occur for $Wo > 1$. However, their analysis does not consider the flow patterns upstream and

downstream of an orifice in unsteady flow. Figure 1.2 shows the flow visualization results obtained by Pullin et al. [16].

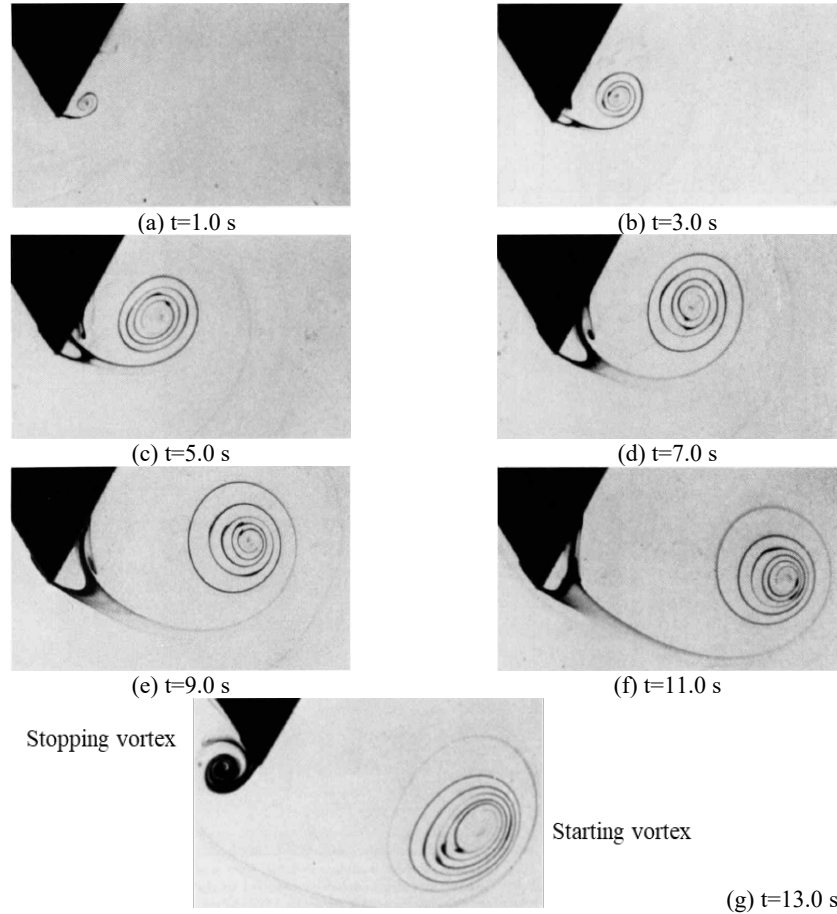


Figure 1.2 Starting & Stopping vortex on a wedge photographed by Pullin & Perry [16]

In this study, with the aim of examining and validating the hypothesis proposed by Masuda et al. [9,11] concerning inertial effects in alternating orifice flow, three-dimensional unsteady CFD simulations are performed for the flow around the damping orifice of a pressure regulator valve. The resulting flow structures in the vicinity of the orifice under forward-flow and alternating-flow conditions are investigated in detail and compared with the flow-visualization results of Pullin et al. [16]. Pullin et al. [16] used dye visualization to observe in detail the starting and stopping vortices generated at the leading edge of a 30° wedge and reported the formation process and temporal evolution of the vortex structures associated with the onset and cessation of the flow.

2. NUMERICAL MODELS & CONDITIONS

In this study, three-dimensional unsteady CFD analyses were conducted for the flow field around the damping orifice used in a pressure regulator system, under forward-flow and alternating-flow conditions. This section describes the geometrical configuration of the computational models and the corresponding simulation conditions.

2.1 Numerical calculation conditions

All numerical simulations were performed using Ansys CFX [17] in unsteady mode. A schematic of the computational domain is shown in Figure 2.1. A circular orifice with an inner diameter of 0.62[mm] and a length of 0.7[mm] was placed at the center of a circular pipe, and straight pipe sections of 25[mm] were provided upstream and downstream of the orifice. This configuration ensures that a fully developed flow reaches the orifice without being affected by the inlet and outlet boundary conditions. The computational grid was locally refined in the vicinity of the orifice and within the downstream shear-layer development region, resulting in a total of approximately 5 million cells. Further details are given in Section 2.4. The simulation conditions are summarized in Table 2.1. A specified volumetric flow rate was imposed at the inlet boundary, a constant-pressure condition was applied at the outlet, and no-slip conditions were enforced on all pipe and orifice walls. The Reynolds number, defined using the orifice diameter as the characteristic length and the maximum flow velocity among the analysed cases, was $Re = 290$; therefore, all simulations were carried out under laminar-flow assumptions.

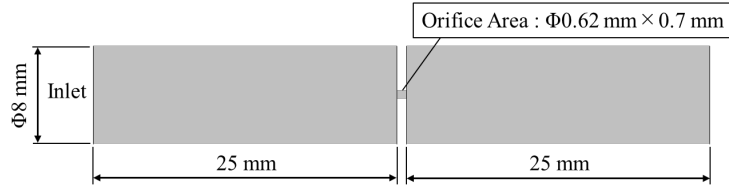


Figure 2.1 3D model of the orifice

Table 2.1 Analysis conditions (general)

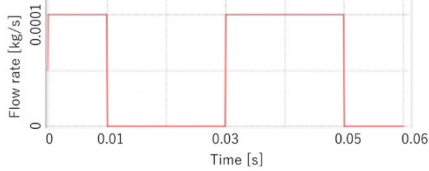
Software used	Ansys® CFX [17]
Analysis type	Transient
Turbulence model	Laminar
Fluid density [kg/m ³]	750
Fluid kinematic viscosity [m ² /s]	9.5×10^{-7}
Grid number [-]	5,000,000

2.2 Unidirectional Flow

The computational conditions for the forward-flow case are summarized in Table 2.2. At the inlet, a stepwise variation of the mass flow rate, as shown in Figure 2.2, was imposed. Specifically, a constant mass flow rate of 0.0001 kg/s was prescribed for $0 \leq t < 0.01$ [s]; the mass flow rate was set to zero 0 [kg/s] for $0.01 \leq t < 0.03$ [s]; a constant mass flow rate of 0.0001[kg/s] was again imposed for $0.03 \leq t < 0.05$ [s]; and the mass flow rate was set to zero for $0.05 \leq t < 0.06$ [s]. This configuration enables observation of the periodic vortex

shedding and the formation process of stopping vortices that occur when the flow rate is abruptly reduced from a steady forward-flow condition.

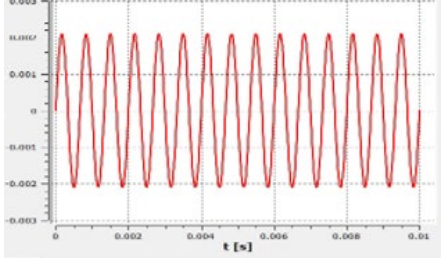
Table 2.2 Analysis condition (unidirectional flow)

Inlet flowrate as boundary condition [kg/s] Flow rate:0.0001 [kg/s] : (t=0-0.01, 0.03-0.05 [s]), Flow rate:0 [kg/s] : (t=0.01-0.03, 0.05-0.06 [s])	 Figure 2.2 Inlet flowrate in unidirectional flow
Outlet boundary condition [MPa]	4

2.3 Alternative Flow

The computational conditions for the alternating-flow case are summarized in Table 2.3. At the inlet, an alternating volumetric-flow boundary condition was imposed as shown in Figure 2.3, by applying a sinusoidal velocity with an amplitude of $a = 0.0021$ [m/s] to generate alternating flow. In this study, two excitation frequencies, 200 [Hz] and 1500 [Hz], were considered, and the frequency dependence of the resulting flow field was evaluated for each case. A constant outlet pressure of 4.2 [MPa] was prescribed as the downstream boundary condition. The time step was chosen such that enough time steps per period was secured for each frequency condition. Calculations were performed over several periods, and data obtained after the flow field had reached a periodically steady state were used for the analysis.

Table 2.3 Analysis condition (alternating flow)

Inlet boundary condition [m³/s] Amplitude; $a=0.0021$ [m/s] Frequency; $f=200, 1500$ [Hz] Time variation of displacement; $\dot{X} = a \sin(2\pi ft)$ (1) Cross section area of boundary; A Flow rate $Q = A\dot{X}$ (2)	 Figure 2.3 Inlet flowrate in alternating flow
Outlet boundary condition [MPa]	4.2

2.4 Grid independence verification.

The results of the mesh-dependence study are shown in Figure 2.4. Mesh sensitivity was evaluated using three models: (a) a tetrahedral mesh with 3 million cells, (c) a hexahedral mesh with 5 million cells, and (e) a hexahedral mesh with 9 million cells. For this comparison, the inlet boundary condition (time-varying flow rate) was taken from the input waveform in Figure 2.3, and the flow fields at $t = 0.036$ [s] were examined. The vorticity iso-surface at $\omega = 0.03$ was chosen such that well-developed vortex rings were present in the solution for cases (e) and (f).

As shown in Figure 2.4(b), the 3-million-cell mesh yields a velocity field that appears smeared, indicating insufficient spatial resolution; moreover, vortical structures are visible only inside the orifice. In contrast, Figure 2.4(d) demonstrates that the 5-million-cell mesh

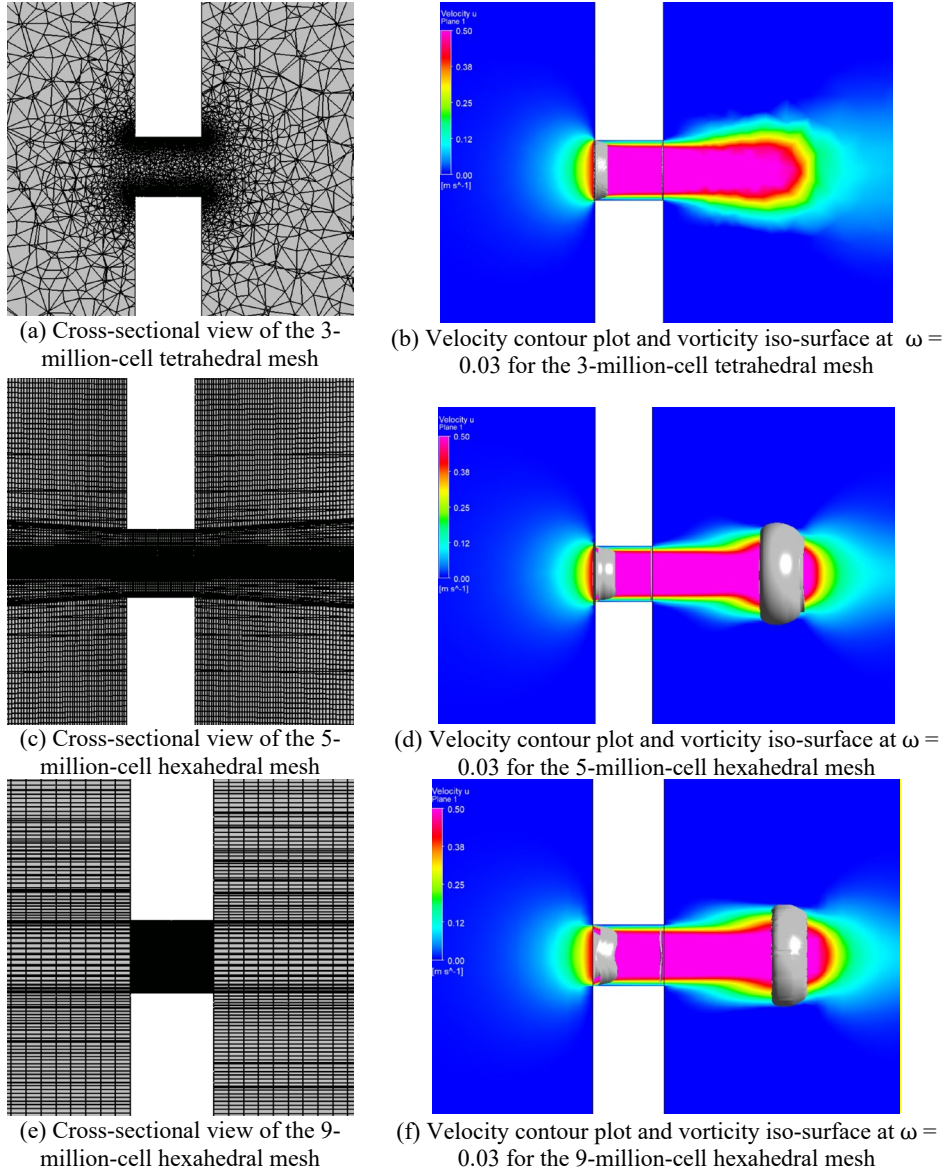


Figure 2.4 Comparison of results as grid independence verification

provides improved spatial resolution, with sharp velocity gradients and vortices captured not only within the orifice but also in regions where the velocity attains its maximum. As shown in Figure 2.4(f), further increasing the mesh size to 9 million cells results in essentially the same flow structures and velocity distributions as those obtained with the 5-million-cell mesh in Figure 2.4(d). Based on these results, a mesh with 5 million cells was adopted for all subsequent simulations. In this mesh, 38 cells are distributed in the radial direction within the orifice, and the first cell height at the wall is approximately $2[\mu\text{m}]$. The detailed mesh configuration is shown in Figure 2.5.

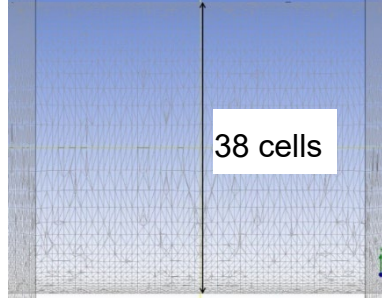


Figure 2.5 Analysis grid of the orifice area

3. ANALYSIS RESULTS

3.1 Unidirectional Flow

The inlet condition is a rectangular waveform as shown in Figure 2.2, and the discussion here focuses on the flow corresponding to the nonzero mass flow rate of $0.0001[\text{kg/s}]$ in the interval $t = 0.03 - 0.05 [\text{s}]$, following the zero-flow interval $t = 0.01 - 0.03 [\text{s}]$.

Time-series distributions of vorticity contours, velocity vectors in the x-direction, and the Q-criterion are presented for each phase, and the vortex structures and flow patterns are compared on this basis. Here, the x-direction velocity is defined as positive for flow from left to right in the Figures. The vorticity field is visualized over the range $0-5000$, and regions with Q values greater than 1×10^7 are used to identify vortex cores.

The Q-criterion is an objective measure for defining and extracting vortical structures in a flow field. By visualizing iso-surfaces of positive Q, where local rotation dominates over strain (deformation), the growth and decay of vortex rings can be clearly captured. Mathematically, Q is defined using the second invariant of the velocity-gradient tensor as

$$Q = \frac{1}{2}(\Omega_{ij} \cdot \Omega_{ij} - S_{ij} \cdot S_{ij}) \quad (3)$$

where Ω_{ij} is the rotation tensor (rotational component of the flow), and S_{ij} is the strain-rate tensor (shear and deformational component of the flow).

3.1.1 Periodic Vortex Generation

In this section, the periodic vortex-generation process under forward-flow conditions is organized and discussed along the time sequence (a)–(h), based on vorticity contours, streamwise x-direction velocity vectors, and Q-criterion fields obtained from the CFD analysis. Figure 3.1.1 shows the temporal evolution of these three quantities in the vicinity of the orifice for the forward-flow case. In frame (a), the fluid begins to accelerate in the positive (rightward) direction. Upstream of the orifice, viscous friction at the inner wall decelerates the fluid adjacent to the wall, and a boundary layer with a wall-normal velocity gradient form and develops. Within this boundary layer, vorticity is continuously generated and accumulated at the wall due to the velocity difference between the core flow and the near-wall region and is transported downstream while remaining attached to the wall from

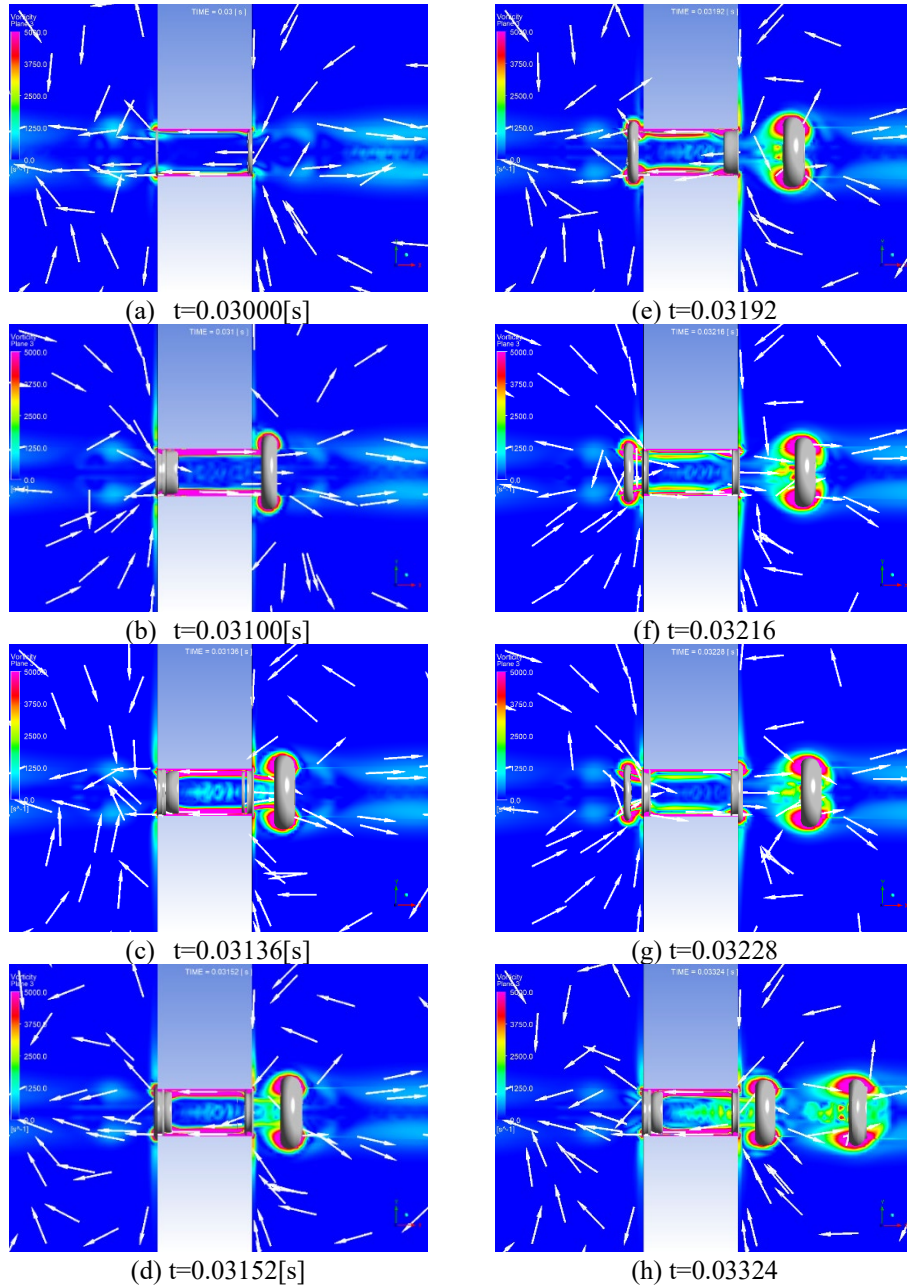


Figure 3.1.1 Vorticity contour, velocity vectors, Q-criterion for unidirectional flow (Q=const.)

the orifice inlet to the outlet. In addition, a small vortex ring is formed at the orifice inlet (left side) due to slight flow separation that occurs as the flow turns around the inlet corner into the orifice. In frame (b), when the boundary layer reaches the vicinity of the orifice exit edge, it separates from the wall under the adverse pressure gradient induced by the sudden

expansion of the flow passage, thereby forming a free shear layer. This shear layer then rolls up under itself-induced velocity and generates a starting vortex ring at the orifice exit edge. The starting vortex thus formed produces an induced velocity field associated with its circulation, giving rise to a vortical structure that carries a reverse flow relative to the mainstream in the vicinity of the orifice exit edge. In frames (c) and (d), as the starting vortex is gradually transported downstream by the mainstream, it strongly interacts with the reverse-flow vortical structure near the exit edge, generating a large velocity shear along their interface. This shear causes part of the vortex to be cut off and split along the streamwise direction, which can be clearly observed in the flow field. In frame (e), prior to fragmentation, the reverse flow induced by the starting vortex would nominally tend to enter the orifice interior, but such inflow is suppressed by the downstream-directed mainstream. At the moment when the starting vortex is fragmented, the vortical mass remaining on the upstream side exhibits a transient motion opposite to the mainstream due to inertia, leading to a localized backflow near the orifice inlet. As a result, a new vortex ring (hereafter referred to as the secondary vortex ring) is formed in the vicinity of the orifice inlet edge. In frames (f), (g), and (h), the secondary vortex generated near the inlet edge is rapidly diffused and attenuated as it is transported downstream through interaction with the mainstream and eventually dissipates. Concomitant with the dissipation of the secondary vortex ring, vorticity is again generated on the inner wall of the orifice due to the renewed velocity difference between the mainstream and the near-wall region, and the streamwise boundary-layer structure along the inner diameter wall is re-established. Subsequently, the same sequence of events—(a) boundary-layer development, (b) roll-up of the free shear layer and generation of a starting vortex, (c)(d) fragmentation of the starting vortex by its induced flow, (e) generation of a secondary vortex ring near the inlet, and (f)(g)(h) dissipation of the secondary vortex ring and reconstruction of the boundary layer—is repeated in time, thereby sustaining periodic vortex-ring generation at the orifice exit.

From the above, the present CFD analysis qualitatively demonstrates that periodic vortex generation in forward orifice flow is governed by the series of unsteady phenomena described in (a)–(h). This vortex-generation mechanism shows good qualitative agreement with the vortex-ring formation process reported in the visualization experiments of Pullin et al. [16]. In this case, vortex rings are transported downstream at time intervals of $\Delta t = 0.0017$ [s]. This interval depends, among other factors, on the magnitude of viscous resistance associated with the orifice length.

3.1.2 Stopping Vortex Generation

In this section, the vortex-formation process that occurs when the inlet flow rate is stepped to zero at $t = 0.01$ [s] from a forward-flow state is examined along the time sequence (a)–(h), based on vorticity contours obtained from the CFD analysis. Figure 3.1.2 shows the temporal evolution of the vorticity distribution in the vicinity of the orifice. In frame (a), the inlet mass flow rate is held constant at 0.0001 [kg/s], and the periodic vortex-generation cycle described in Figure 3.1.1(a)–(h) is still ongoing. At this stage, the free shear layer separated from the orifice exit edge rolls up and forms a starting vortex at the exit edge.

In frame (b), the influence of the abrupt reduction in inlet flow rate at $\Delta t = 0.01$ [s] has not yet become pronounced. The starting vortex that has formed begins, through its own induced velocity field, to generate a region of reverse flow relative to the mainstream in the vicinity of the orifice exit edge. In frame (c), the effect of the stepwise reduction of the inlet flow rate at $t = 0.01$ [s] starts to appear. The fluid mass inside the orifice and near the exit, excluding the near-wall region, continues to move downstream due to inertia, whereas

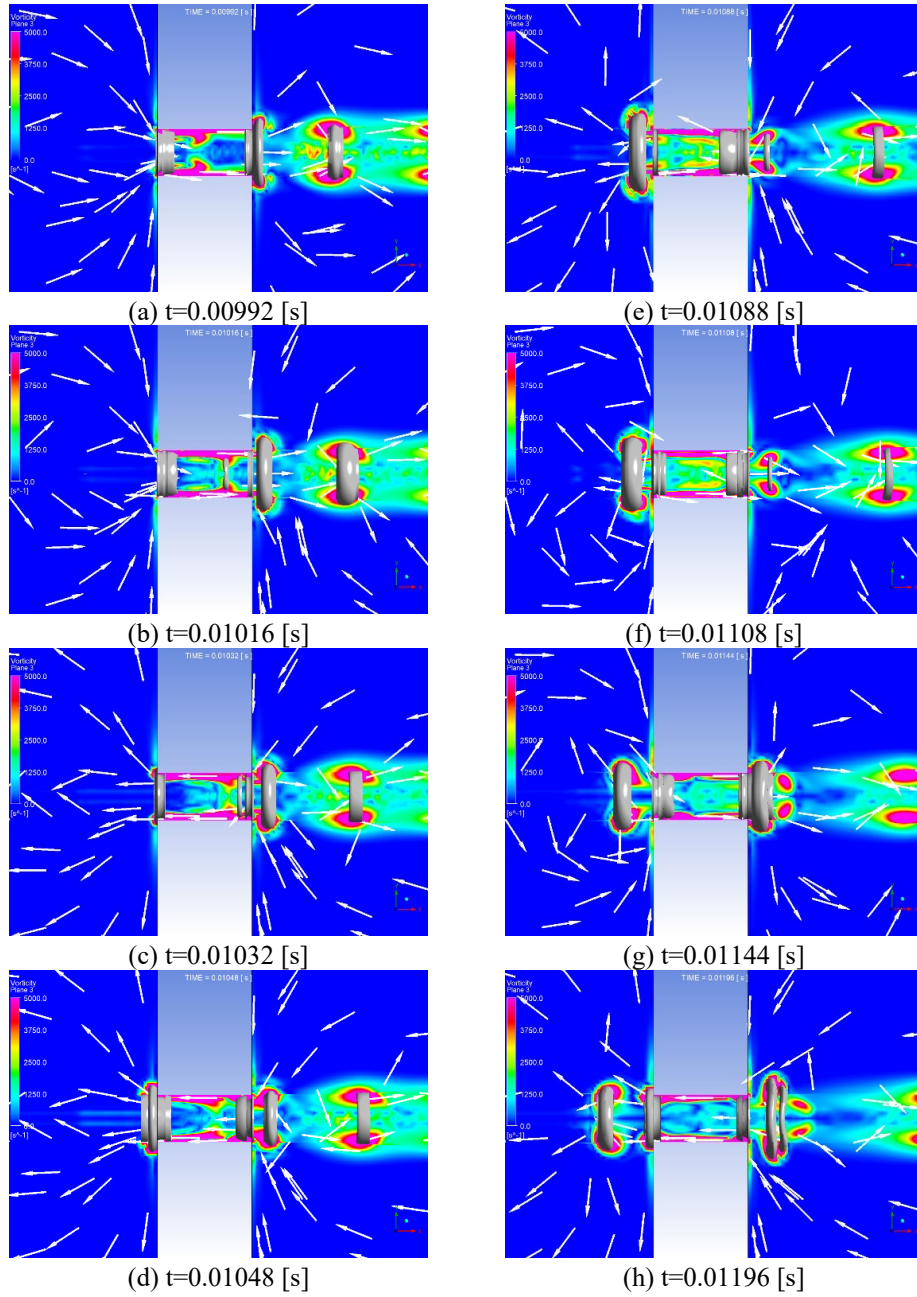


Figure 3.1.2 Vorticity contour, velocity vectors, Q-criterion for unidirectional flow (step flow rate drop to zero at $t=0.01$ [s])

the flow at the orifice exit edge is pulled back toward the orifice interior as the mainstream decelerates. This backflow, together with the induced velocity field of the pre-existing starting vortex, generates a new vortex ring of opposite sign immediately downstream of the orifice exit edge. This structure is referred to as a stopping vortex. It is found to be in

qualitative agreement with the stopping vortex reported in the visualization experiments of Pullin et al. [16]. In frame (d), the starting vortex located at the orifice exit edge is subjected to strong shear in the vicinity of the exit edge due to the combined action of the stopping vortex and the mainstream and is consequently split. The fragmented vortex mass, influenced by the stopping vortex and by fluid inertia at the moment of fragmentation, exhibits a transient motion opposite to the mainstream, thereby giving rise to a new vortex ring of opposite sense to the mainstream at the orifice inlet edge. In frame (e), the transient reverse flow directed into the orifice interior leads to growth of the vortex ring at the inlet edge and simultaneously draws the vortex mass remaining on the outlet side back toward the interior of the orifice. In frame (f), similarly to frame (c), the reversal of the main-flow direction causes the flow near the orifice to be redirected downstream. As a result, a stopping vortex of opposite sign to the inlet-edge vortex ring is generated just downstream of the inlet edge. At the same time, the vortex mass that remained on the outlet side is pushed further downstream by the mainstream. In frame (g), the inlet-side vortex ring is again subjected to strong shear and is fragmented due to fluid inertia associated with the change in flow direction toward the downstream side and due to its interaction with the stopping vortex. Simultaneously, a new vortex ring is formed at the orifice exit edge, and the vortex masses that remained in the outlet opening and within the orifice bore are transported downstream by the mainstream. In frame (h), the change in the main-flow direction causes the vortex ring on the outlet side to be fragmented, while a new vortex ring is generated on the inlet side. Although vorticity continues to be generated periodically from the shear layer, the absence of external flow-rate supply leads to an overall reduction in the vorticity level, and the vortical structures are markedly attenuated. Thus, immediately after the abrupt reduction in flow rate, repeated generation and subsequent fragmentation of vortex rings at the orifice inlet and exit edges occur through the splitting of the starting vortex and the subsequent formation of stopping vortices. The present CFD analysis confirms that this process ultimately converges to a vorticity-dissipation regime dominated by viscous diffusion.

3.1. Alternative Flow

In this section, the flow field around the orifice under alternating-flow conditions is examined by comparing vortex structures and flow patterns using streamwise x-direction velocity contours, velocity vectors, streamlines, and Q-criterion distributions.

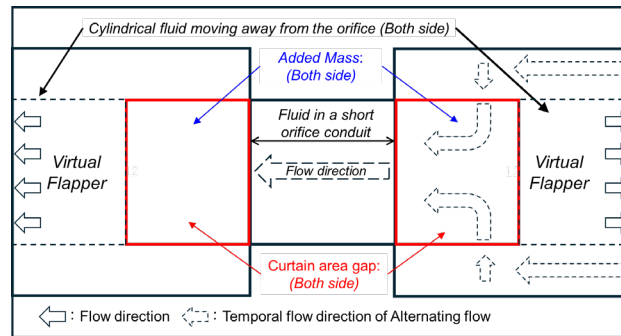


Figure 3.2.1 Simplified flow pattern near an orifice

From the flow patterns discussed in the previous section, it was found that the upstream and downstream flow fields have a significant influence on the dynamic characteristics of the flow within the orifice. Accordingly, two characteristic regions are identified upstream and

downstream of the orifice, as shown in Figure 3.2.1 (hereafter referred to as the “virtual flapper” region and the “virtual curtain” region). These regions are considered to affect, among other properties, the added mass associated with the internal orifice flow. In what follows, the detailed flow patterns within these regions under alternating-flow conditions are analysed.

3.2.1 Flow patterns (200[Hz])

Figure 3.2.2 shows the correspondence between time in the input flow-rate waveform at 200 [Hz] and the phases (a)–(h) (which match the Figure labels).

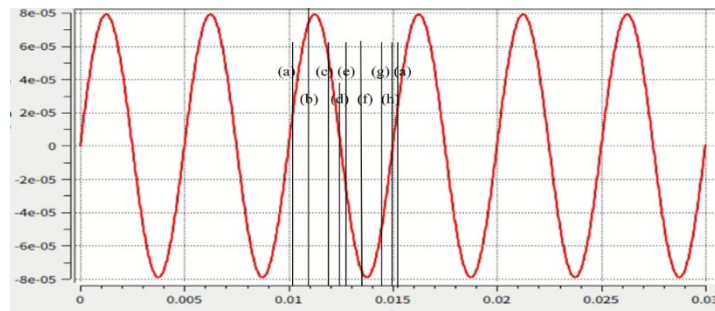


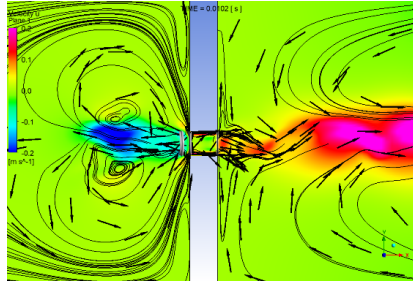
Figure 3.2.2 Correspondence between time history of input waveform and phase labels

Figure 3.2.3 presents the time-series distributions of streamwise x-direction velocity contours, velocity vectors, and Q-criterion for each phase (a)–(h) within one cycle at 200 [Hz]. Here, the x-direction velocity is defined as positive for flow from left to right in the Figures, and the velocity contours are visualized within the range ± 0.2 [m/s]. The vortex-core regions are visualized using Q values greater than 2×10^6 for the 200 [Hz] case. A total of six cycles were computed, and the data shown are taken from after the second cycle, when the flow field was judged to have reached a periodically steady state.

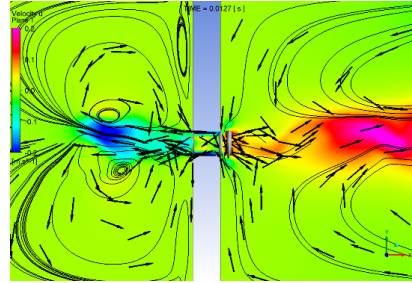
At phase (a), the main flow, which in the first half-cycle had been directed from the right chamber to the left chamber, switches direction and a flow from the left chamber to the right chamber begins to develop. In the central region of the orifice bore, a low-velocity region forms in association with this flow-reversal process, and a fully developed jet has not yet been established. Near the inlet orifice edge (left side), the Virtual Flapper region, a small-scale reverse-flow zone that remains due to inertia from the previous half-cycle, persists. The main flow circumvents this reverse-flow region by passing around its outer periphery and enters the orifice with a slight contraction at the inlet (left side), where a vortex ring is formed. This constitutes the Virtual Curtain region. On the other hand, near the outlet orifice edge (right side), a shear layer corresponding to the new main-flow direction begins to form, and a starting vortex is generated in association with this shear layer.

At phase (b), the main flow from the left chamber to the right chamber further intensifies, and the x-direction velocity in the central region of the orifice bore attains an almost uniform positive value in the streamwise direction. The vortex ring that was generated at the orifice inlet edge (left side) in phase (a) grows significantly under the influence of the increasing main-flow velocity. At the orifice exit edge (right side), the roll-up of the shear layer associated with the new main-flow direction progresses, and the vortex ring corresponding

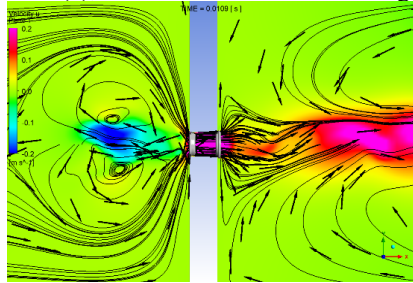
to the starting vortex for the rightward flow develops downstream. At phase (c), although the main flow from the left chamber to the right chamber has just passed the peak flow-rate



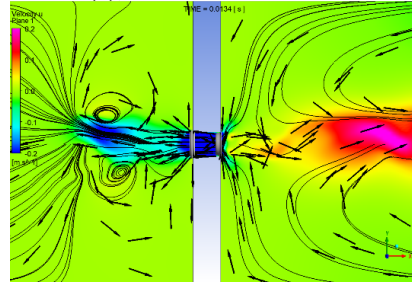
Phase (a); main flow directed to the right



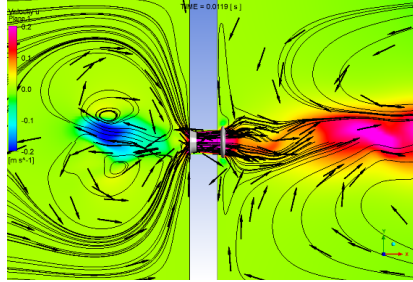
Phase (e); main flow directed to the left



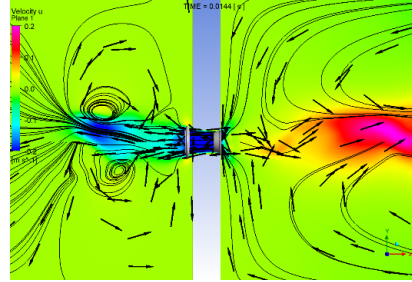
Phase (b); main flow directed to the right



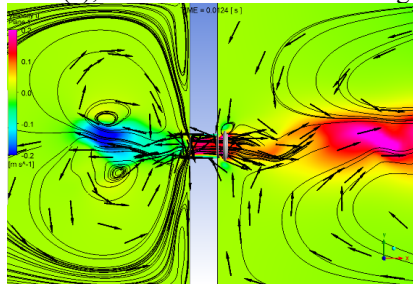
Phase (f); main flow directed to the left



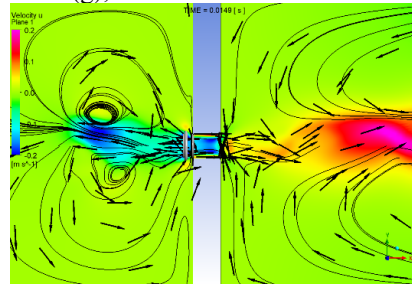
Phase (c); main flow directed to the right



Phase (g); main flow directed to the left



Phase (d); main flow directed to the right



Phase (h); main flow directed to the left

Figure 3.2.3 Velocity contours, vectors, streamlines & Q-criterion iso-surface for 200[Hz] alternating flow

amplitude occurring near phase (b), the x-direction velocity in the central region of the orifice bore still exhibits an almost uniform positive value in the streamwise direction, and

a jet-like flow is established. At the orifice exit edge (right side), the starting vortex associated with the rightward flow grows further downstream, and the region of concentrated vorticity expands into the right chamber. At phase (d), the main flow from the left chamber to the right chamber is in the stage just before reversing direction. The x-direction velocity in the central region of the orifice bore maintains a positive component but decreases markedly, and the flow rate approaches its minimum. The vortex ring that has developed near the orifice inlet edge (left side) decays under the influence of the decelerating main flow. In contrast, the fluid mass in the central region of the orifice and near the exit continues to be carried downstream (to the right) by inertia, while at the orifice exit edge the deceleration of the main flow induces a reverse flow along the wall toward the upstream side (left). The interaction between this reverse flow and the induced velocity field of the pre-existing starting vortex leads to the formation of a stopping vortex ring near the orifice exit edge (right side), which rotates in the opposite sense to the starting vortex associated with the rightward flow. At phase (e), the main-flow direction begins to reverse from left-to-right to right-to-left, i.e., from the left chamber toward the right chamber to the right chamber toward the left chamber. Inside the orifice, a low-velocity reverse-flow region persists in the central part, while a leftward flow has already developed near the wall. As time progresses, the overall flow field transitions to a quasi-uniform right-to-left stream. The vortex ring formed near the orifice exit edge (right side) up to the preceding phase (d) decays and is simultaneously split, resulting in two vortical structures, one at the orifice exit edge on the left side and one on the right side. At the orifice exit edge on the left side, which now acts as the outlet for the new (leftward) main-flow direction, roll-up of the wall-bounded shear layer induced by the reversed main flow advances; consequently, the left-side vortex ring aligned with the right-to-left flow is generated and grows. The right-side vortex ring, together with the residual reverse-flow region (Virtual Flapper), decays under the influence of the leftward main flow. From phase (e) to phase (h), the main-flow direction is fully reversed, and a flow from the right chamber to the left chamber is established. The distribution of the velocity contours exhibits a nearly mirror-symmetric pattern relative to phases (a)–(d), reflecting the flow-direction reversal. A Virtual Flapper region, characterized by a reverse-flow zone opposite to the main (leftward) stream, appears near the orifice inlet (right side), and a Virtual Curtain region is formed in which the main flow bypasses this reverse-flow zone, entering the orifice from the outer periphery toward the center.

3.2.2 Flow patterns (1500[Hz])

Figure 3.2.4 shows the correspondence between time in the input flow-rate waveform at 1500 [Hz] and the phases (a)–(h) (which match the Figure labels).

Figure 3.2.5 presents the time-series distributions of streamwise x-direction velocity contours, velocity vectors, and Q-criterion for each phase (a)–(h) within one cycle at 1500 [Hz]. Here, the x-direction velocity is defined as positive for flow from left to right in the Figures, and the velocity contours are visualized within the range ± 0.2 [m/s]. For the 1500 [Hz] case, vortex-core regions are visualized using Q values greater than 1×10^7 . In all cases, the data are taken from simulations after 30 cycles, at which point the flow field was judged to have reached a periodically steady state.

At phase (a), the main-flow direction switches from that of the preceding half-cycle, and the flow begins to enter from the left chamber into the right chamber. At this time, a new starting vortex is observed to form near the orifice exit edge (right side). In the central region of the orifice, excluding the near-wall region, a low-velocity zone remains, and the flow field is in transition from a nearly uniform reverse-flow state to a nearly uniform main-flow state. In

the vicinity of the orifice inlet edge (left side), a reverse-flow region (Virtual Flapper region) persists, in which the flow from the previous half-cycle is retained due to fluid inertia. The main flow bypasses this Virtual Flapper region by passing around its periphery and enters

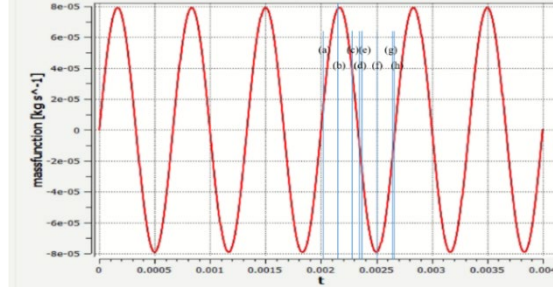


Figure 3.2.4 Correspondence between time history of input waveform and phase numbers

the orifice with a contracted cross-section at the inlet (left side), forming a Virtual Curtain region. Near the orifice inlet edge (left side), the inflow from the outer periphery of this Virtual Curtain region and the residual reverse-flow region (Virtual Flapper region) together give rise to the formation of a new vortex ring. At phase (b), the main flow from the left chamber to the right chamber is further intensified, and a jet-like flow with an almost uniform streamwise velocity is established inside the orifice. The vortex ring that formed near the orifice inlet edge (left side) in phase (a) grows significantly due to the increased inflow velocity along the outer periphery of the Virtual Curtain region. Meanwhile, at the orifice exit edge (right side), the starting vortex generated by the roll-up of the shear layer further develops in the downstream direction and appears as a strong concentration of vorticity around the jet. At phase (c), although the main flow from the left chamber to the right chamber has just passed the maximum flow-rate amplitude occurring near phase (b), the x-direction velocity in the orifice-bore region still maintains an almost uniform distribution in the streamwise direction, and a strong jet is sustained. At the orifice inlet edge (left side), the shear layer formed up to the previous phase continues to grow downstream, whereas at the orifice exit edge (right side), the starting vortex ring begins to decay. At phase (d), the main flow that has been directed from the left chamber to the right chamber up to the preceding phase weakens, and the flow is in the process of reversing from right to left. The fluid mass in the central region of the orifice and near the orifice exit (right side), which constitutes a Virtual Flapper region, still tends to move toward the downstream side of the right chamber due to inertia from its prior motion. In contrast, near the orifice exit edge (right side), deceleration of the main flow induces a reverse (leftward) flow along the inner-wall surface of the orifice. The strong shear arising from the interaction between this reverse flow and the induced velocity field of the starting vortex leads to the formation of a stopping vortex, which rotates in the opposite sense to the starting vortex, near the orifice exit edge (right side). The vortex ring that existed near the orifice inlet edge (left side) in phase (c) is still present, although it is attenuating. At phase (e), the main-flow direction has completely reversed, and the flow now proceeds from the right chamber toward the left chamber. A low-velocity region remains in the central part of the orifice, but the near-wall region has already started to accelerate, and the flow field is transitioning toward a nearly uniform main-flow field in the new (leftward) direction. Meanwhile, up to the preceding

phase (d), a Virtual Curtain region has formed near the right orifice edge from a locally stagnated state, and a reverse-flow region (Virtual Flapper) persists downstream of it. From

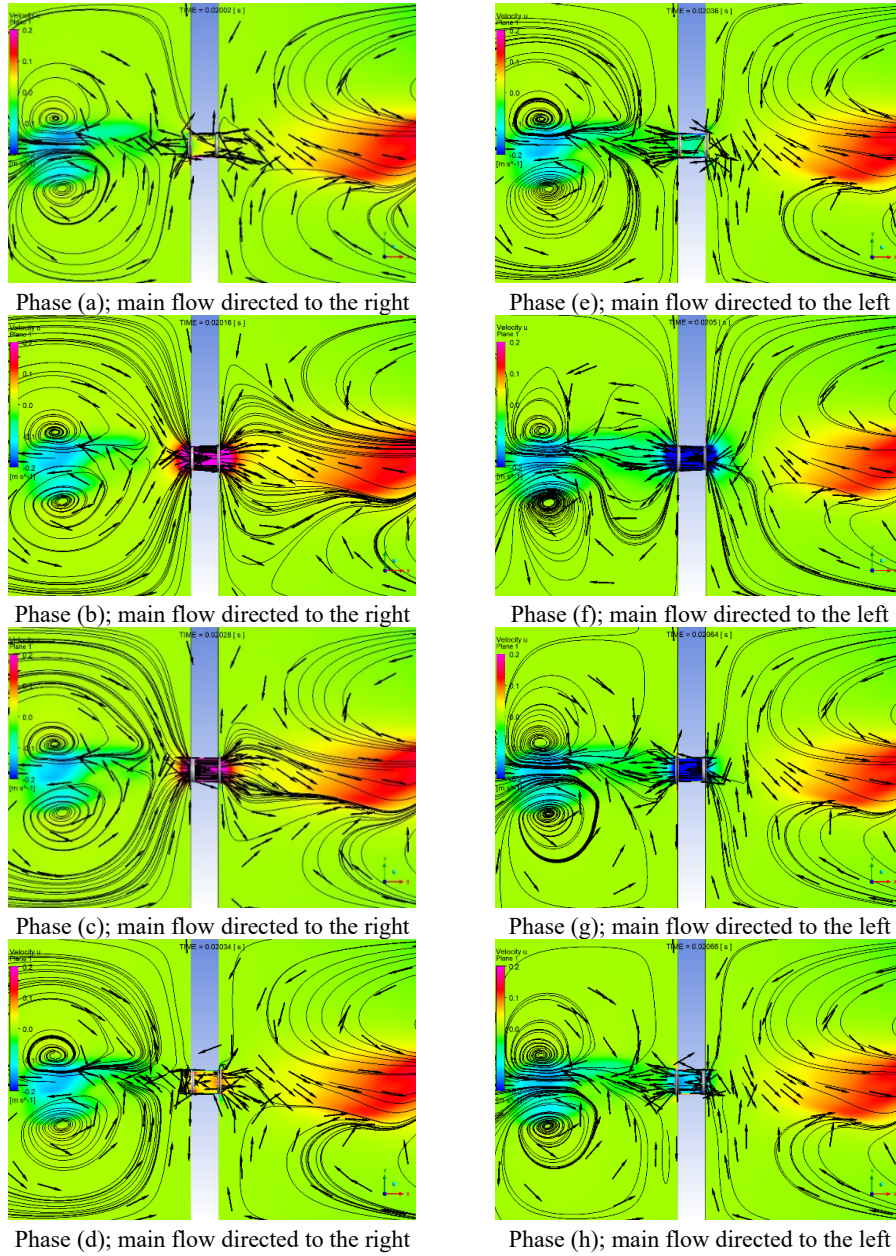


Figure 3.2.5 Velocity contour & Q-criterion iso-surface for 1500[Hz] alternating flow

phases (e) to (h), the main-flow direction remains reversed, and a flow from the right chamber to the left chamber is established. The distributions of the velocity contours exhibit

patterns that are nearly mirror images of those in phases (a)–(d), reflecting the reversal of the flow direction. A Virtual Flapper region, characterized by a reverse-flow zone opposite to the main (leftward) stream, appears near the right orifice, while a Virtual Curtain region forms in which the main flow avoids this reverse-flow zone and enters from the outer periphery toward the orifice center.

3.2.3 Comparison of Flow Patterns (200 V.S. 1500 [Hz])

Under the 200 [Hz] condition, each half-cycle is sufficiently long that the velocity distribution inside the orifice becomes nearly uniform at an early stage. As a result, the starting vortex, stopping vortex, and the reverse-flow Virtual Flapper and Virtual Curtain regions are relatively weak, and the entire system undergoes a quasi-steady reciprocating motion as a single entity.

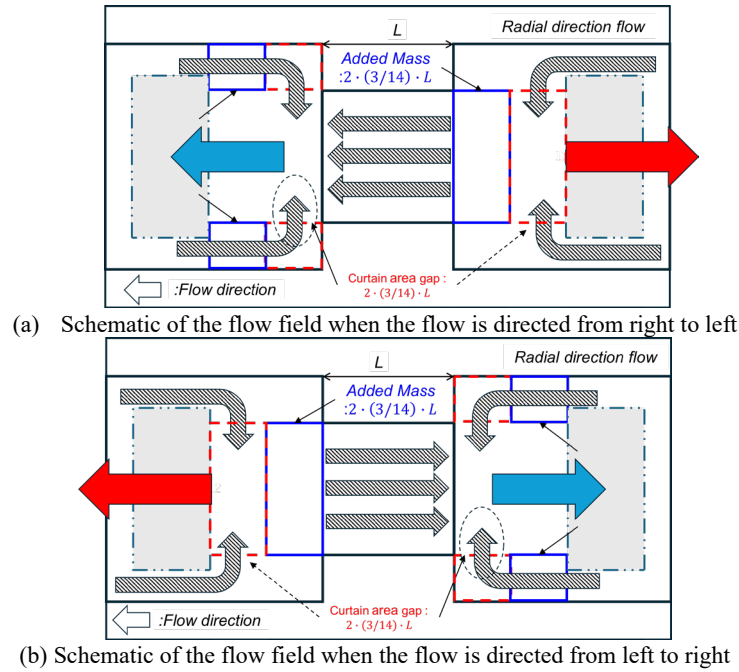


Figure 3.2.6 Existence of Virtual Flapper and Virtual Curtain regions with a well-defined flow direction

By contrast, under the 1500 [Hz] condition, the switching of the main flow does not fully propagate across the entire orifice cross-section, and the Virtual Flapper and Virtual Curtain regions at the orifice inlet, as well as the starting and stopping vortices on the outlet side, remain strongly developed. Consequently, throughout phases (a)–(d), a strongly non-uniform and highly unsteady flow field is formed, characterized by pronounced velocity gradients and vortical structures. Phases (e)–(h) exhibit flow-direction-reversed patterns that are nearly mirror images of phases (a)–(d), and the same tendencies are observed. From these results, the existence of Virtual Flapper and Virtual Curtain regions with a fixed flow direction, as illustrated in Figure 3.2.6, has been confirmed. It has been clarified that changes

in the flow velocity within these regions significantly affect the fluid inertia in the vicinity of the orifice.

4. EFFECTIVE INERTIAL LENGTH IN ALTERNATING ORIFICE FLOW

In this section, the effective inertial length in oscillatory orifice flow is discussed based on the numerical results for the 200 [Hz] and 1500 [Hz] cases. As is evident from the time-series results in Figure 3.2.3 and 3.2.5, the Virtual Flapper and Virtual Curtain regions are periodically formed and disappear on both sides of the orifice in synchrony with the acceleration and deceleration of the main flow. These structures constitute a continuous velocity field with the fluid column inside the orifice, indicating that not only the fluid within the orifice, but also the fluid in the upstream and downstream chambers, is accelerated and decelerated as a single system. This implies an inertial effect equivalent to an apparent extension of the orifice length. In other words, the presence of the Virtual Flapper and Virtual Curtain regions and their associated added-mass effect can be interpreted as imparting, in addition to the geometric orifice length, an extended region near the orifice end faces (here referred to as the effective inertial length). Inspection of the velocity-contour fields in Figure 3.2.3 and 3.2.5 shows that, for both 200 [Hz] and 1500 [Hz], reverse-flow zones and quasi-cylindrical velocity-distribution regions are formed over several millimetres in the vicinity of the orifice openings. Because these regions are periodically accelerated and decelerated together with the main flow, the effective inertial length exceeds the geometric length by a substantial margin. Focusing on the frequency dependence, under the 200 [Hz] condition each half-cycle is sufficiently long that the Virtual Flapper and Virtual Curtain regions remain relatively weak, and the velocity distribution within the chambers is smoothed over time. As a result, the region corresponding to the extended length of the orifice fluid column is confined to the immediate vicinity of the orifice, and the increase in effective inertial length is considered to be relatively moderate compared with the geometric length. In contrast, under the 1500 [Hz] condition, the flow-direction reversal proceeds to the next half-cycle before it has fully propagated across the entire orifice cross-section. Consequently, the Virtual Flapper and Virtual Curtain regions persist strongly across phases, and a broad additional liquid-column region is maintained in the upstream and downstream chambers. The fluid volume that is effectively coupled to the orifice fluid column is therefore larger than in the 200 [Hz] case, and the effective orifice length greatly exceeds the geometric length, with a tendency to increase further as the oscillation frequency increases. These findings demonstrate that, in oscillatory orifice flow, an inertia assessment based solely on the geometric orifice length is inadequate. Introducing an effective inertial length that incorporates the Virtual Flapper and Virtual Curtain regions and their associated added-mass effects is essential for correctly describing the damping characteristics of oscillatory flow through an orifice. It should be noted, however, that the present analysis focuses on a local region in the vicinity of the orifice and does not evaluate the momentum balance for the entire system, including the full chamber volumes. Accordingly, the spatial extent of the Virtual Flapper and Virtual Curtain regions is not necessarily captured completely, and some uncertainty remains in the quantitative evaluation of the effective inertial length.

5. CONCLUSIONS

In this study, CFD analysis was used to clarify the processes of periodic vortex generation and stopping-vortex formation in unidirectional orifice flow, and to qualitatively investigate

new features specific to alternating orifice flow, namely the Virtual Flapper flow, the Virtual Curtain region, and their associated added-mass effects. The main findings are as follows.

5.1 Unidirectional Flow

It was confirmed that the vortical structures around the orifice are composed of the following sequence of unsteady processes:

- vorticity generation within the boundary layer along the inner wall of the orifice
- formation of a starting vortex due to roll-up of the free shear layer at the orifice exit edge
- generation of a stopping vortex through the interaction between the induced flow of the starting vortex and the mainstream, and the resulting fragmentation of the starting vortex
- generation of secondary vortex rings in the vicinity of the orifice inlet
- downstream transport and dissipation of the secondary vortex rings, accompanied by reconstruction of the boundary-layer structure.

It was also confirmed that this near-orifice region (corresponding to the Virtual Curtain region) plays an important role in the flow dynamics.

5.2 Alternative Flow

Under alternating Flow conditions, a flow composed of the Virtual Flapper region and the Virtual Curtain region exists in a fixed spatial arrangement around the orifice and exhibits periodic strengthening and weakening. The flow inside the orifice reverses its direction in synchrony with the acceleration and deceleration of the mainstream. Within the Virtual Curtain region, a spatially fixed flow is continuously formed from the outer-diameter side toward the inner-diameter side on both sides of the orifice, while its intensity varies in time. Due to the Virtual Flapper and Virtual Curtain regions and their associated added mass, additional liquid-column regions are generated upstream and downstream of the orifice, and the effective inertial length greatly exceeds the geometric orifice length. Therefore, when evaluating the damping characteristics of alternating flow through an orifice, it is necessary to consider not only the geometric orifice length but also the effective inertial length.

References

- [1] Johansen, F. C. : Flow through Pipe Orifices at Low Reynolds Numbers, A.M. I. Mech. E., p.231-245 (1929).
- [2] A. F. D'Souza, R. Oldenburger, Dynamic Response of Fluid Lines, Journal of Basic Engineering, p.589-598(1969).
- [3] Benjamin, E. W., Victor, L.S.: Fluid Transient in Systems, PRENTICE HALL, p. 2-9 (1993).
- [4] Lau, K. K., Edge, K.A. and Johnston, D.: Impedance Characteristics of Hydraulic Orifices, Proc. Instn. Mech. Engrs, Part I: J. Sys. Control Eng. 209. p.241-253 (1995).
- [5] D. Nigel Johnston: Prediction of Fluid Inertance in Nonuniform Passageways, Journal of Fluids Engineering, Vol. 128, No.2, p.266-275 (2006).
- [6] Higo, H., Tanaka K., Yamaguchi, T.: New One Dimensional Modeling for Complex Oil Passage Coupled with CFD Results, Proceedings of the ASME/BATH 2014 Symposium on Fluid Power & Motion Control FPMC2014, 2014.
- [7] Uchida, S.: The Pulsating Viscous Flow Superposed on the Steady Laminar Motion of Incompressible Fluid in a Circular Pipe, ZAMP Vol.7, p.403-422 (1956).
- [8] Munekazu, O., Manabu, I., Tateo, Usui.: Flow Pattern and Frictional Losses in Pulsating Pipe Flow, Part5, Wall Shear Stress and Flow Pattern in a Laminar Flow, Bulletin of the JSME, Vol. 24, No.187, p.75-81 (1981).

- [9] S. Masuda, et. al., ‘Modelling and simulation of damping orifices for pressure control valves, Proc. ISFP 2024 JFPS, 1A2-03, <https://doi.org/10.5739/jfps.52.8>
- [10] Rosenberg, R.C. and Karnopp,D.C.: Introduction to Physical System Dynamics, McGraw-Hill, 1983.
- [11] Seiei Masuda, et. al., ‘Modelling and Simulation of Damping Orifices for Pressure Control Valves 2nd Report), Proceedings of the 15th International Fluid Power Conference (15th IFK), 9–11 March 2026.
- [12] S. Oshima, et. al., ‘Frequency Characteristics of Cylindrical Short Chokes for Hydraulic Valves (In case of using general hydraulic oil), J. Hydraulics. Pneumatics (in Japanese), Vol.15, No.2, 1984, pp.157-165,
- [13] S. Miyagawa, ‘Frequency Characteristics of Cylindrical Short Chokes, J. JSME (in Japanese), Vol.42, No.355, 1971, pp.847-856, <https://doi.org/10.1299/kikai1938.42.847>
- [14] Catherine Loudon, et. al., ‘The Use of the Dimensionless Womersley Number to Characterize the Unsteady Nature of Internal Flow, Journal of Theoretical Biology, vol. 191, No.1, 1998, doi:10.1006/jtbi.1997.0564
- [15] J. R. Womersley, ‘An elastic tube theory of pulse transmission and oscillatory flow in mammalian arteries, WADC Technical Report 56-614, Wright Air Development Center, Wright-Patterson AFB, OH, USA, Jan. 1957.
- [16] M. Van Dyke, ‘An Album of Fluid Motion, The Parabolic Press, Stanford, CA, 1982.
- [17] ANSYS: CFX-Pre User's Guide Release 2025R1 (2025)

NOMENCLATURE

Symbol	Explanation of variable symbols
A_R	Metering valve opening area
P_1	Inlet pressure of metering valve
P_2	Outlet pressure of metering valve
P_{2S}	Pressure of Spring room
P_C	Pressure downstream of Pressure Control valve
Q	Q-criterion
Q_0	Flow rate through damping orifice
Q_{By}	Bypass flow rate through differential pressure control valve
Q_{MV}	Flow rate through metering valve
Q_P	Pump flow rate
S_{ij}	Shear and deformational component of the flow
t	Time
x	Cartesian coordinate in the orifice streamwise direction
X	Valve displacement
W_0	Womersley number
ω	Vorticity
Ω_{ij}	rotational component of the flow

New Sensing Method for Cartridge Valves

Massimiliano Ruggeri^{1*}, Sara Baldoni², Chiara Gialluca²,
Christopher Rosi², Matteo Ferri³, Luca Belsito³

1. CNR - STEMS, via Canal Bianco, 28 – 44124 Ferrara, Italy

2. VIS Hydraulics S.r.l. Innovation Center, Via G. Marconi, 115, 41028 Serramazzoni (MO) Italy

3. CNR – ISMN, Via Piero Gobetti 101, 40129 Bologna, Italy

* corresponding author, massimiliano.ruggeri@cnr.it

Abstract.

Trends in hydraulic system design tend to avoid proportional valves because of energy losses intrinsic to their operation. Yet valves are fundamental in many applications and often are used in functional safety critical applications. Current systems require increasing precision in controls and actuations, as well as higher safety performance levels in both components and systems. Both of these requirements involve sensors to make the component and systems fully observable and controllable. It is particularly difficult to implement sensors integrated directly in cartridge valves. However, sensors installed directly inside the valves would allow direct observation of their behaviour. This would provide highly valuable diagnostic information from functional safety perspective and would be crucial for precise control of valve operation. The paper presents a new sensor integration method for cartridge valves, which allows a direct observation of their functionality and enables a precise tuning of the valve in low flow metering condition, as well as the direct control of valve safe position. The applied sensor is a miniaturized MEMS sensor whose usage is the consequence of a mechanical-electronic valve co-design approach. The valve design, the sensor and its readout electronics, as well as the tests are presented, with a discussion of the possible usage of the assembly and a more general idea for new design of both proportional and status valves.

Keywords. Cartridge Valve, Proportional Valve, Pressure Sensing, Functional Safety, Valve Diagnosis, Sensors, MEMS, DETF.

1. INTRODUCTION

The increasing demand for compact, high-performance fluid power systems has driven significant research toward the integration of miniaturized sensors directly within hydraulic components, particularly valves [1]. Miniaturized pressure sensors represent an important focus of this research. In conventional hydraulic systems, pressure sensing is typically performed using externally mounted transducers connected via ports or tubing, which

introduces delays, parasitic volumes, and measurement uncertainty. Recent advances in Micro-Electro-Mechanical Systems (MEMS) technology have enabled the development of pressure sensors with dimensions on the order of millimetres or below, opening the possibility of true in-situ sensing within valve bodies and flow passages. MEMS devices leverage micro-machined silicon structures—such as membranes, beams, or cavities—to transduce pressure into electrical signals via piezoresistive, capacitive, or resonant mechanisms and, to date, piezoresistive sensing has remained the dominant approach due to its robustness and compatibility with industrial environments [2]. Resonant MEMS sensors represent an alternative approach offering significantly higher strain sensitivity and excellent long-term stability.

State-of-the-art miniaturized pressure sensors typically achieve footprints below 1–2 mm and thicknesses below 0.1 mm, enabling integration into confined geometries such as micro-channels, catheter tips, or embedded cavities [3].

Advanced designs employ membrane-based architectures, where pressure-induced deformation is measured through integrated strain gauges or piezoresistive films. More recent developments include integrated microchannel sensors, in which the sensing element is fabricated directly adjacent to or within the fluid path, minimizing dead volume and improving dynamic response [4]. These approaches are particularly relevant for hydraulic valves, where accurate measurement of transient pressure gradients across small control volumes is critical for performance characterization and control using both absolute and differential pressure sensors [5].

A key trend in the literature is the move toward fully embedded sensing architectures, where the sensing element is structurally integrated into the component rather than mounted as a discrete device. This includes chip-level integration using techniques such as flip-chip bonding [6], through-silicon vias, or wafer-level packaging, allowing the sensor to be embedded within metallic or polymeric housings while maintaining electrical connectivity and environmental isolation. In harsh environments such as hydraulic systems, the sensing element is typically protected by a metallic or polymeric isolation diaphragm, enabling exposure to high-pressure fluids while shielding sensitive electronics. These packaging strategies are essential to ensure compatibility with high pressure levels, temperature variations, and aggressive media typical of fluid power applications.

Emerging research further explores ultra-miniaturized and high-sensitivity sensing technologies, including devices based on two-dimensional materials (e.g., PtSe₂ or graphene membranes), which exhibit significantly enhanced sensitivity due to reduced mass and increased gauge factors. Such sensors demonstrate the potential for sub-millimeter, high-resolution pressure measurement, although their application in hydraulic environments remains largely at the research stage [7]. Additionally, multifunctional micro-sensors capable of simultaneously measuring pressure, temperature, and flow are being developed, offering the prospect of integrated diagnostic capabilities within a single embedded device [8]. Despite these advances, several challenges remain for the adoption of embedded pressure sensors in hydraulic valves. These include reliable sealing under high pressure (often exceeding 250–300 bar), vibration resistance, long-term stability in oil-contaminated environments, thermal drift compensation, and the integration of signal conditioning electronics within limited space. Consequently, while MEMS-based miniaturized sensors have demonstrated strong potential for embedded valve sensing, their widespread industrial implementation is still constrained by packaging, robustness, and cost considerations. Nevertheless, ongoing developments in microfabrication, materials, and system integration

indicate that fully sensorized hydraulic valves, with pressure sensing embedded directly within the valve body, represent a key direction for next-generation fluid power systems.

Pressure is one of the most important parameters to measure in hydraulic systems, but both for flow prediction and for functional safety concerns, valve spool position sensing is still one of the most critical embedded measurements; indeed most sensors are integrated but not fully embedded (often mounted externally on valve body) and not miniaturized enough [9]. For that reason several studies focus on valve spool position inference through EMC analysis as shown in [10] or innovative approach to valve actuation design through piezo-electric devices integrating piezo-electric sensors for spool movement sensing [11].

Finally, functional safety requirements ask for higher diagnosability both indirect, through virtual sensors and software, and direct, through real sensors. For that reason, there are many studies to infer valve faults, even using machine learning, like the one in [12], in which sensing the current in the coil, both spool stuck and spring failure valve faults can be identified. From the direct diagnosis side, many studies attempt, with a multi-sensor approach and a sensor fusion, to make diagnosis and predictive diagnosis of valves [13]. But the crux of these studies is the need for high sampling frequencies to perform the Fast Fourier Transform (FFT) of the signals and evaluate the remaining useful life of the components.

Valve diagnosis and, in particular, faults of its core is a key point for functional safety, due to the need to understand if valves are in their safe position, and that diagnosis is also made through different sensors like accelerometers for an indirect sensing of the core position [14]. Unfortunately, these methods do not also include the possibility of improving the control of the valve spool position.

2. THE MEMS-DETF SENSOR

The type of MEMS used in this research is a double-ended tuning fork (DETF) type MEMS developed by CNR from 2012 and published in [15], [16] and [17], and now in production through a spin-off company. It is based on a micro-mechanical tuning fork structure connected to micro capacitors vibrating in vacuum inside the MEMS chip.

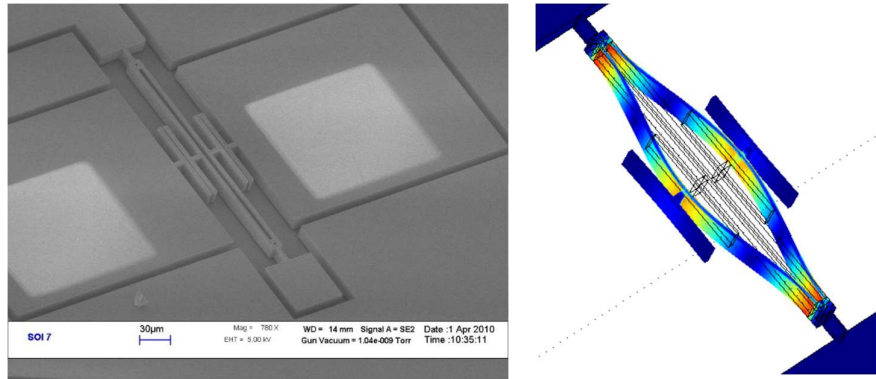


Figure 1. MEMS-DETF structure (left) and working principle (right)

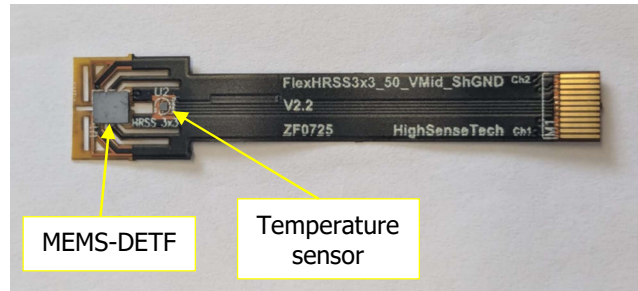


Figure 2. MEMS-DETF and MEMS Temperature Sensor mounted on flat cable

The tuning fork is maintained in continuum vibration at its resonance frequency through an external electronic circuit in the readout electronics connected to the MEMS chip pinout. The tuning fork is connected to the MEMS chip at its ends and when the chip is subjected to strain, then the tuning fork changes its resonance frequency according to the strain. If the strain is a compressive strain, then the frequency becomes lower, while if the strain is an extensive strain, then the resonance frequency increases. This frequency shift is linear with the strain. Due to the high stiffness of silicon, it shows a very high sensitivity of $\sim 200 \text{ p}\epsilon$, 10.000 times the sensitivity of a traditional flexible strain gauge. This characteristic together with the possibility to vary the sample frequency (until 10-15 kHz) by changing the buffer size of frequency values, corresponding to each MEMS oscillation, used to calculate the frequency in the microcontroller's reciprocal frequency counter peripheral, make the sensor particularly suitable for hydraulic applications. Of greatest interest are those applications in which it is difficult to find a position where to install a traditional sensor, or where a high sample frequency is required. The MEMS-DETF has been applied to hydraulic components through years and published in [18], [19] and [20]. In this work the sensor has been applied to a valve for the first time.

3. VALVE WORKING PRINCIPLE

The valve used in this project is manufactured by VIS Hydraulics, and the technical code is PFR0.S08, represented with its hydraulic symbol in Figure 3.



Figure 3. Proportional flow regulator and its hydraulic symbol

This is a non-pressure compensated proportional flow regulator, equipped with a solenoid, designed to control the amount of oil flowing between port 2 (inlet port) and port 1 (outlet port), in response to a specific current applied to the solenoid.

As shown in Figure 4, by regulating the current applied to the solenoid, it is possible to regulate the flow between the minimum and the maximum admitted values.

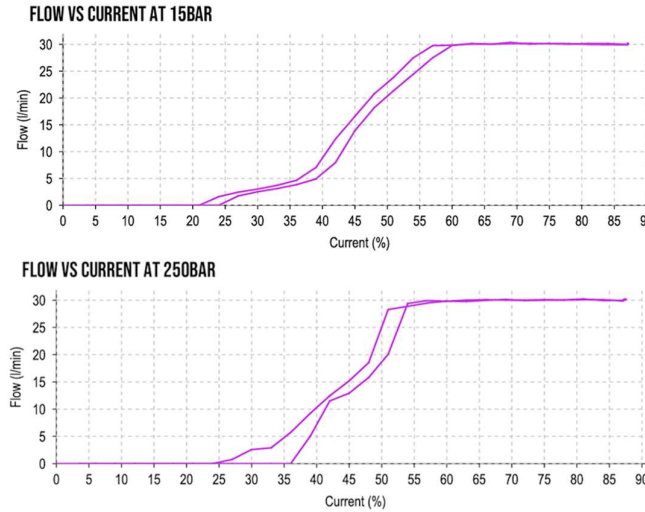


Figure 4. flow regulation curves at different pressure drops

The current flowing through the coil generates an electromagnetic field that magnetizes the internal elements, and an attraction force is generated between the so-called moving plunger and the fixed armature, leading to an internal movement of the pilot and main poppet. By changing the current value, the attraction force changes accordingly, and a different opening point is reached, thus obtaining a different regulated flow. Because the valve is not pressure compensated, the amount of the regulated flow in response to the current may change depending on the pressure variation across the inlet and outlet ports.

As shown in Figure 5, the original valve is modified by creating an internal pressure chamber (called sensing chamber), designed to withstand the same pressure acting on the internal stage. The pressure in the sensing chamber is detected by the MEMS sensor applied in the modified nut.

In the real case, something happens inside the valve, which can be divided in three stages:

- Coil off: the valve is closed and there is no oil flow between inlet and outlet port, and the pressure inside the valve (and in the sensing chamber) is equal to the inlet pressure, as showed in Figure 6.
- Coil on with current between 20% and 60% of the steady-state current: when the electromagnetic force generated by the electromagnetic field acting on the moving plunger is sufficient to start to open the axial orifice, the main poppet lifts itself, reaching an equilibrium position with a certain distance from the pilot poppet. By

increasing the current, this distance is maintained (so both the main and the pilot poppet lifts together with the increasing magnetic force inside the pole tube).

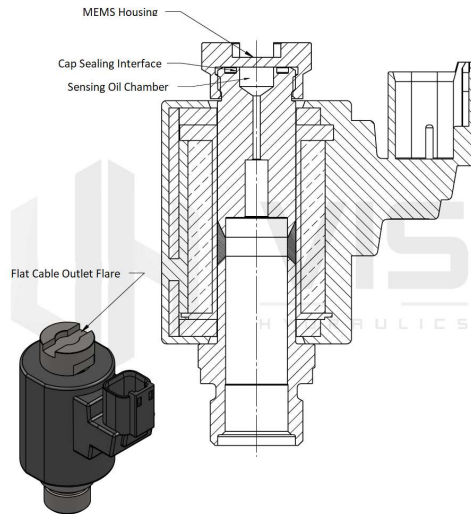


Figure 5. New valve design with the sensing chamber

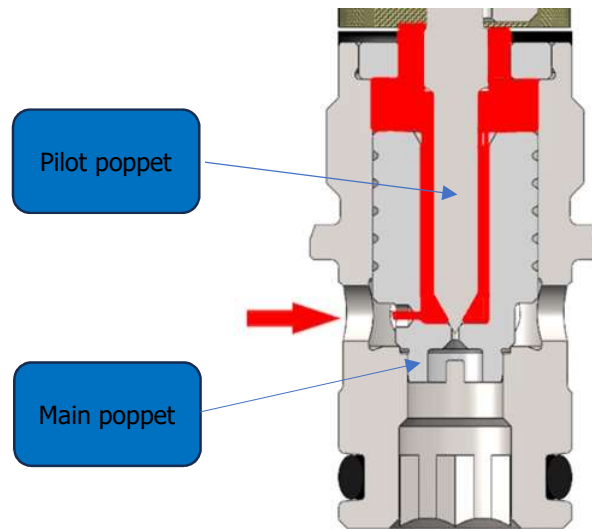


Figure 6. internal pressure level @ 0 A current

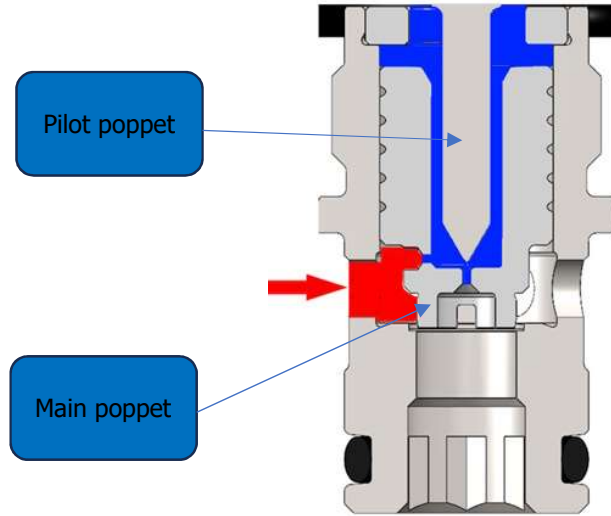


Figure 7. Internal pressure level at intermediate current.

The pressure inside the sensing chamber is no longer the inlet one (Figure 7), but now there is an intermediate pressure that can be described by recalling the turbulent flow equation through a restrictor, derived from the Bernoulli equation:

$$Q = C_d \cdot A \cdot \sqrt{2 \cdot \frac{\Delta p}{\rho}} \quad (1)$$

where Q is the flow, C_d is the efflux coefficient, A is the orifice area, Δp is the pressure drop across the orifice, and ρ is the fluid density.

At equilibrium, neglecting the leakages it can be assumed that the flow across the radial orifice is the same as the flow across the axial one, and under steady-state conditions we can neglect some terms, obtaining the following equation:

$$C_d \cdot A_{in} \sqrt{p_{in} - p_{ch}} = C_d \cdot A_{out}(x) \cdot \sqrt{p_{ch} - p_{out}} \quad (2)$$

Where, in our case, the inlet area is related to a fixed radial orifice, while the outlet area is given by a percentage of the area of the axial orifice, depending on the distance between the pilot and the main poppet.

From simulations, we know that the intermediate position of the pilot poppet opens around the 40% of the outlet area; with this assumption, the pressure in the sensing chamber can be approximated, obtaining:

$$p_{ch} = \frac{A_{03}^2 \cdot p_{in} + A_{04}^2 \cdot p_{out}}{A_{04}^2 + A_{03}^2} \quad (3)$$

For the valve under study:

- With an inlet pressure of 200 bar, the internal calculated pressure is around 133 bar.

- With an inlet pressure of 100 bar, the internal calculated pressure is around 67 bar.
- With an inlet pressure of 50 bar, the internal calculated pressure is around 33 bar.

More generally a linear dependency from p_{in} and p_{out} can be plotted and the result for p_{out} 1, 10, 50 and 80 bar is shown in Figure 8.

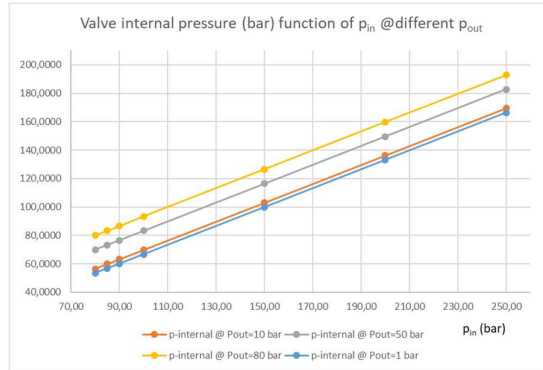


Figure 8. Internal pressure function of p_{in} (inlet) and p_{out} (load)

- Coil on with current above the 65% of the steady-state current. Now the electromagnetic force is sufficiently high to pull up the plunger and the pilot poppet, while the main poppet reaches its maximum stroke; this leads to the complete valve opening, and the pressure inside drops to the minimum value that is usually the p_{out} , the load pressure. In this condition the maximum pressure drop of the valve is reached, around 14 bar in the test bench where the p_{out} is the reservoir pressure. In other condition in real applications the pressure is related to the p_{out} , the pressure of the load.

4. TEST SETUP

The sensor was glued on the cap (Figure 9 left) using the Micro Measurements M Bond AE-10 epoxy and it was thermally conditioned in order to stabilize the glue for higher temperatures. Then the cap was mounted on the valve pole tube, facing the sensing chamber. Then the valve was mounted on a special three-way manifold. A first setup was prepared at the static test bench to verify the sensor sensitivity in respect to the pressure strain in the sensing chamber (Figure 9 right). The main objective of the test was to qualitatively verify that the sensor was able to acquire pressure transients with sufficient resolution with respect to the variation of the resonance frequency with respect to the pressure variation.

To evaluate the sensor performance a program application under Windows was created. The program communicates with the sensor readout electronics control unit through the USB 3.0 connection and it permits acquiring at high speed all the data from the sensor, allowing the complete acquisition of the raw data.

The observed response during various tests with 30 bar steps and 50 bar steps, respectively, manually generated in the bench, was that the sensor and its double ended tuning fork sensing elements inside were capable of acquiring the pressure variation. In particular, due to the asymmetrical shape of the cap, the Sensor 1 (Channel 1 in yellow in Figure 9), directed

in the same axis as the machining created to position the sensor in a protected position, by removing material from the cylindrical cap, is more sensitive than Sensor 2 (Channel 2 in green in Figure 9). Sensor 2 is aligned along the axis orthogonal to the aforementioned mechanical machining.



Figure 9. The sensor glued on the cap (left) and the valve at the static pressure test bench (right)

- The greater presence of material therefore makes the cap more rigid in the direction orthogonal to the machining, making the tuning fork less sensitive in that same direction. From a quantitative point of view, allowing for small measurement errors in data collected manually during the tests, it can be stated that Sensor 2 is approximately one-third less sensitive compared to Sensor 1.



Figure 10. The static test bench (manual pressure regulation bench) and the application program for resonance frequency acquisition. Valve is mounted without coil.

Specifically, the data collected indicate an average sensitivity of:

- Channel 1: 62.2 Hz/bar;

- Channel 2: 41.4 Hz/bar.

Such data can be easily obtained from graphs such as the one shown in the Figure 11, which are obtained from the data recorded by the PC application that records the data communicated in real time by the sensor via USB communication. In the test shown in the graph in Figure 11, steps of 50 bar from around 1.5 bar to 300 bar are manually generated using the static pressure bench. In the test the resonance frequency of both sensors, together with the component temperature, is acquired in real time and stored for post processing, here made through Excel after conversion of the CSV data files.

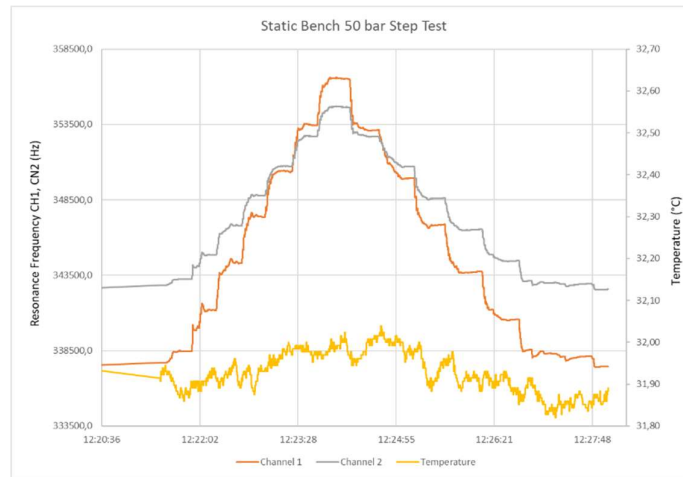


Figure 11. 50 bar steps test at static pressure test bench

In order to evaluate the sensor's resolution with respect to the variation in Hz per bar, as a comparison it can be used the resolution obtainable from a standard reciprocal frequency counter of a modern microcontroller, which allows acquisition of tenths of a Hz even with data of hundreds of kHz. Therefore, the resolution that Sensor 1 can achieve is greater than 0.002 bar and the Sensor 2 a resolution greater than 0.003 bar.

5. SENSOR QUALIFICATION AT TEST BENCH

To have a complete test qualification both in static and dynamic conditions, a test campaign was planned and executed at the Innovation Centre premises of the VIS Hydraulics Company. The valve was mounted on the test bench and connected to both the inlet and outlet ports in the same configuration used in actual machine applications (Figure 12). It is a proportional flow control valve, whose regulated flow rate depends both on the force transmitted by the coil to the spool and on the inlet pressure, therefore is of the uncompensated type. Therefore, acquiring the inlet pressure in real time could be a very interesting feature for the electronic control of the valve. The first test was designed to verify whether the sensor is able to acquire the inlet pressure when this pressure is returned to the sensing chamber created inside the valve, near the sensorized cap (see Figure 5 for reference).

The first test was performed maintaining the valve unpowered in the closed position, which can be critical due to the need for the pressure to be returned in the sensing chamber at the end of pole tube. Pressure steps were generated at the test bench and the sensor signals were acquired during the test. As shown in the upper plot of Figure 13, the sensors signal is changing according to the pressure generated by the test bench (lower plot of Figure 13). Moreover, the resonance frequency steps are proportional to the pressure steps, due to the screwed cap strain under the pressure in the sensing chamber.

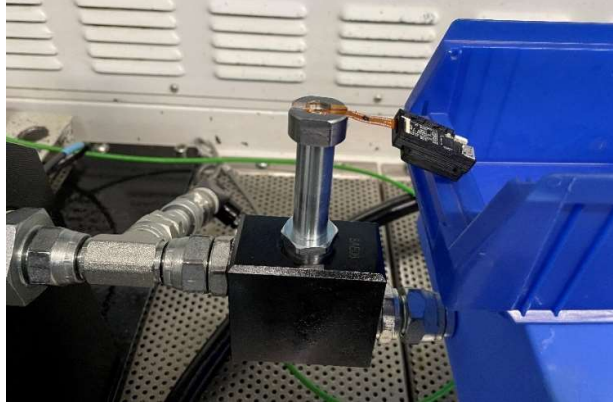


Figure 12. The valve with the sensor and the readout electronics at test bench

As it can be seen in the figure, the resonance frequency levels are not the same at the beginning and end of the test, despite the initial and final pressures being equal. This is due to the temperature difference between the valve and the cap, which is detected by the MEMS temperature sensor in the flat cable adjacent to the MEMS-DETF strain sensor. This effect, which has a significant impact on the measurement, results from the thermal deformation of the metal component. This deformation is essentially linear with temperature and can be compensated for using two possible methods: a temperature-dependent strain ratio parameter between silicon and metal, or by detecting the change in resonance frequency due to thermal deformation.

In the case under study it was found that the thermal strain is around $F_T = 3235,8 \text{ Hz/}^\circ\text{C}$. This temperature compensation was then applied to the acquired sensors resonance frequencies starting from the initial test temperature using the simple equation:

$$f_{corr}(t) = f(t) - F_T * (T(t) - T(t_0)) \quad (4)$$

where f_{corr} is the compensated resonance frequency, f is the uncompensated resonance frequency, F_T is the thermal strain frequency correction factor, $T(t)$ is the current temperature and $T(t_0)$ is the initial temperature of the test. The result of the compensation is presented in Figure 14. In the Figure, the resonance frequency levels at the beginning and the end of the test, corresponding to the same pressure levels, are the same, validating the accurate temperature compensation made through (4).

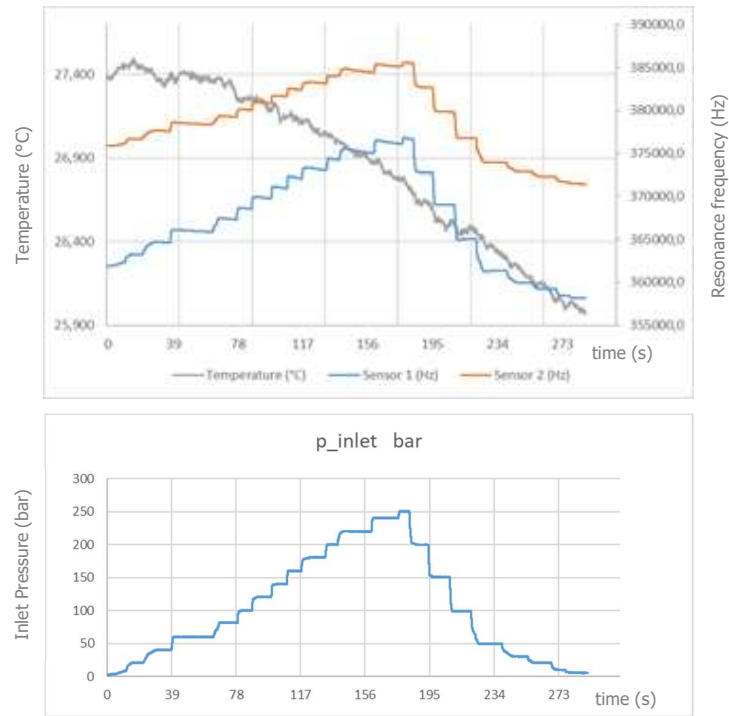


Figure 13. Uncompensated Sensor Response to the Pressure steps at Test Bench

In order to understand the relationship between the sensor signals and the real pressure values acquired through the sensors installed in the test bench, a comparison is made with a test where 50 bar steps of the test bench pressure were generated both increasing and decreasing pressure from 9 bar to 250 bar.

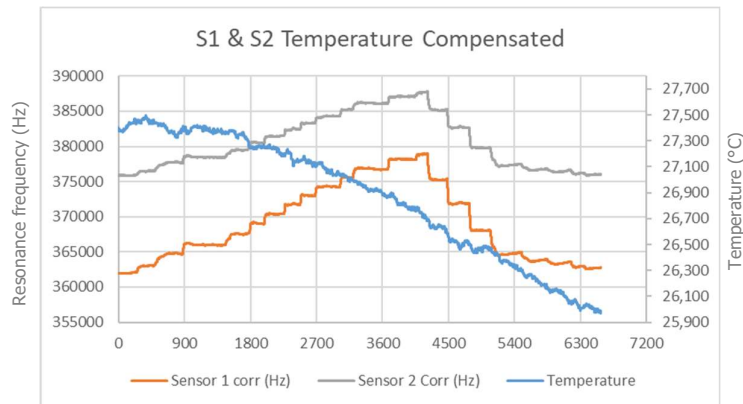


Figure 14. Temperature Compensated Sensor Response to the Pressure

A Matlab® script was used to adapt both the scale and the timing of the acquisitions. The sensor signal is compensated in temperature and the timing is shifted in order to make the starting time of both acquisitions synchronized with the pressure steps. The result is shown in Figure 15.

The pressure estimated from the raw MEMS resonance-frequency signal was compared with the reference pressure measured by the inline pressure sensor. The MEMS frequency was converted into pressure using a linear calibration derived from the mean values measured during the static pressure plateaus. The calibration yielded a scale factor of 0.0120 bar/Hz, corresponding to a MEMS pressure sensitivity of approximately 83.3 Hz/bar. To minimize the influence of possible time-synchronization errors between the two acquisition systems, only the static pressure conditions were considered, excluding the transition regions. The reconstructed MEMS pressure shows good agreement with the reference measurement, with an RMSE of 1.157 bar, corresponding to 0.461% of the full-scale pressure range.

In fact, as it can be noted in the figure, the MEMS-DETF sensor is capable of acquiring pressure values with excellent agreement with the data from the sensors used in the test bench.

The final result of the test is that the sensor installed in the screwed cap allows measuring the inlet pressure, which is returned to the sensing chamber obtained in the terminal position of the valve's pole tube.

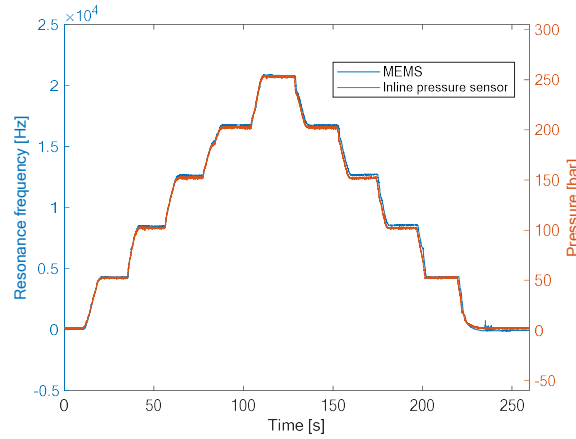


Figure 15. Temperature Compensated Sensor Response (normalized) to the Pressure Steps test

In order to complete the sensor qualification it is fundamental to understand what happens to the internal pressure when the valve is commanded and then the plunger changes its position, consequently the pilot poppet and main poppet change their positions accordingly.

6. FUNCTIONAL TESTS: INTERNAL PRESSURE OBSERVER

Once it was demonstrated that the sensor facing the internal sensing chamber is capable of acquiring the inner pressure with excellent agreement with the test bench sensors,

subsequent tests were performed at the same test bench, by testing the valve under all its operating conditions. Electrical commands were generated for the valve to power the coil and activate the valve's internal spool, causing it to open proportionally to the command. The tests were repeated at different circuit supply pressures (p_{in}), in order to verify the general applicability and the repeatability of the functionality tested.

A test setup was created where the valve was controlled using a production electronic board usually sold together with the valve. The board, connected with a PC application through a CAN network using a CAN to USB gateway, is capable both to program, to calibrate and to control the valve output, also generating repeatable timed commands. Indeed the system is also used for endurance tests in the company facilities. A PWM ramp and the corresponding current ramp were generated from 0% to 100% and way back, in order to replicate an entire cycle of valve command from valve fully closed to valve fully open and then the opposite transient until valve fully closed condition. The test was repeated at different inlet pressure (p_{in}) values, in order to validate the sensor signal. Here the results for $p_{in} = 100$ bar and $p_{in} = 200$ bar are presented.

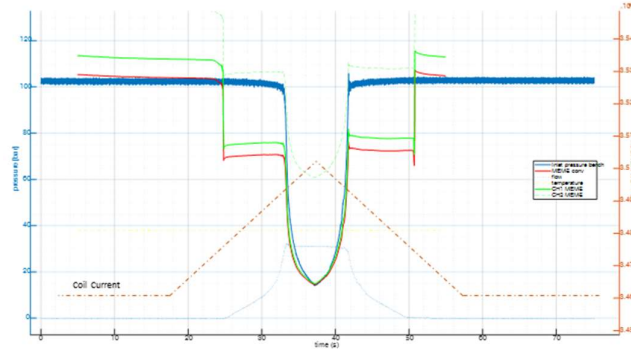


Figure 16. Full Valve Cycle at $p_{in} = 100$ bar

As it can be observed in Figure 16 for the test at $p_{in}=100$ bar, the control cycle lasts 40 seconds, that means that it is a quasi-static test, because the dynamics are very limited.

In the test diagram it can be observed both the line pressure and the MEMS signals, as well as the oil flow through the valve and the command to the valve coil (coil real current).

- The first behaviour that can be noticed is that the internal pressure is not equal to Inlet pressure p_{in} from the instant the poppet starts moving, as explained in the valve working principle discussion.
- The second behaviour that can be noticed is that the internal pressure remains constant from the first moment of valve opening in metering conditions until saturation opening. Practically speaking the internal pressure holds a nearly constant value across the entire flow regulation range and its value is near the theoretical one of 67 bar calculated for 100 bar of inlet pressure (red line in the figure, corresponding to the temperature compensated MEMS signal).
- The third behaviour that can be noticed is that when the valve is fully open the internal pressure becomes equal to the sum of the load pressure and the pressure difference across the valve ($p_{load} + \Delta p$).

The test case at $p_{in} = 200$ bar is presented in Figure 17. As it can be observed in the figure, the behaviour of the valve under coil actuation is the same as in the previous case. The difference in this case is that the equivalent internal pressure measured by the MEMS sensor in the regulation phase is around 133-135 bar, a value completely compatible with the value previously calculated in equation (3).

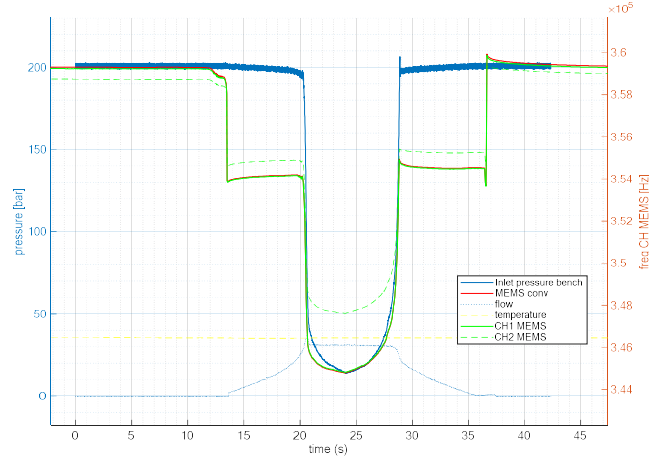


Figure 17. Full Valve Cycle at $p_{in} = 200$ bar

Another interesting result to be noticed is that the valve opens at different current values due to pressure and this event is not directly observable without the MEMS-DETF sensor. Conversely, the automatic control could be used to acquire the change in internal pressure and could regulate the metering phase of the valve control depending on the current and the boundary conditions; this would ensure repeatability of the command regardless of the valve's operating conditions.

7. CONCLUSION

The MEMS-DETF sensor installed in the screwed cap of the cartridge valve, facing the sensing chamber, where the internal valve pressure is reported, demonstrates some important aspects of valve sensor-oriented design. The sensor further demonstrates its capability of being used as an external pressure sensor in hydraulic applications applicable to thick walls due to its high sensitivity. The mechanical-electrical co-design of the valve demonstrates that a combined approach to design enables the development of new components with higher diagnosability rate and easier to control precisely, due to new sensor integration methods. The sensor applied to cartridge valves is a powerful element for both control and diagnostics. Moreover, it allows the precise metering control, even if the valve is uncompensated. Indeed it is able to detect the exact force - and relative coil current - at which the valve spool starts to move. From the diagnosis point of view, it allows a direct observation of the effects of the valve spool (and poppet) movement. Based on the internal valve pressure acquisition and its relationship with the inlet pressure measurement, it is possible to evaluate the status of the valve. This allows determining whether the valve is closed, partially open for flow

regulation, or fully open. Furthermore, it can also be used as a redundant pressure sensor due to the proven relationship between the Inlet pressure and the valve internal pressure. The information on the status of the valve is crucial for functional safety critical applications, where the safe state is often related to the de-energized position of a safety valve (e.g. load dump) and/or to a position of a proportional valve, to prevent both single critical failures and latent faults. The next steps in studying the potential applications of these promising MEMS-DETF sensors in valves, involve an innovative approach to valve design, aiming to create areas and volumes where all the pressures involved in valve control can be measured. The ultimate goal will be the design of a fully integrated valve with the sensors required for its complete control.

8. REFERENCES

- [1] Liu, J., Yuan, C., Matias, L., Bowen, C., Dhokia, V., Pan, M. and Roscow, J. (2024), Sensor Technologies for Hydraulic Valve and System Performance Monitoring: Challenges and Perspectives. *Adv. Sensor Res.*, 3: 2300130. <https://doi.org/10.1002/adrs.202300130>.
- [2] Song P, Ma Z, Ma J, Yang L, Wei J, Zhao Y, Zhang M, Yang F, Wang X. Recent Progress of Miniature MEMS Pressure Sensors. *Micromachines*. 2020; 11(1):56. <https://doi.org/10.3390/mi11010056>.
- [3] W. Hasenkamp, D. Forchelet, K. Pataky, J. Villard, H. Van Lintel, A. Bertsch, Q. Wang, P. Renaud, Polyimide/SU-8 catheter-tip MEMS gauge pressure sensor, 2012, *Biomedical Microdevices*, № 5, p. 819-828, Springer Science and Business Media LLC, doi: 10.1007/s10544-012-9661-8.
- [4] M. Dijkstra, M.J. de Boer, J.W. Berenschot, T.S.J. Lammerink, R.J. Wiegerink, M. Elwenspoeck, Miniaturized thermal flow sensor with planar-integrated sensor structures on semicircular surface channels, *Sensors and Actuators A: Physical*, Volume 143, Issue 1, 2008, Pages 1-6, ISSN 0924-4247, <https://doi.org/10.1016/j.sna.2007.12.005>.
- [5] Fan, Xutao et al. "A High-Performance Micro Differential Pressure Sensor." *Micromachines* 2024, 15(11), 1396; <https://doi.org/10.3390/mi15111396>.
- [6] Huang M, Wu X, Zhao L, Han X, Xia Y, Gao Y, Cui Z, Zhang C, Yang X, Qiao Z, Li Z, Han F, Yang P, Jiang Z. Small-size temperature/high-pressure integrated sensor via flip-chip method. *Microsyst Nanoeng.* 2024 Jul 23;10:104. doi: 10.1038/s41378-024-00723-3. PMID: 39050588; PMCID: PMC11266362.(integrated pressure and temperature sensor)
- [7] Zhou Y, Werner EM, Lee E, Chu M, Nguyen T, Costa KD, Hui EE, Khine M. High-resolution integrated piezoresistive sensors for microfluidic monitoring. *Lab Chip*. 2021 Jan 7;21(1):83-92. doi: 10.1039/d0lc01046d. Epub 2020 Dec 10. PMID: 33300516; PMCID: PMC9521707.
- [8] Chi-Yuan Lee, Chen Chia-Hung, Chin-Lung Hsieh, Chong-An Jiang, Siao-Yu Chen, Flexible, Multifunctional Micro-Sensor Applied to Internal Measurement and Diagnosis of Vanadium Flow Battery, *Micromachines*; Basel Vol. 13, Fasc. 8, (2022): 1193. doi:10.3390/mi13081193.
- [9] M. Papoutsidakis, A. Srivastava and S. Chowdhary, Displacement Sensors for Linear Electrical, Hydraulic and Pneumatic Actuators, 2019 Amity International Conference on Artificial Intelligence (AICAI), Dubai, United Arab Emirates, 2019, pp. 269-273, doi: 10.1109/AICAI.2019.8701227.

- [10] Wilber Acuña-Bravo, Enrico Canuto, Marco Agostani, Marco Bonadei, Proportional electro-hydraulic valves: An Embedded Model Control solution, *Control Engineering Practice*, Volume 62, 2017, Pages 22-35, ISSN 0967-0661, doi 10.1016/j.conengprac.2017.01.013.
- [11] Šimic M, Heraković N. Investigation of Innovative High-Response Piezoelectric Actuator Used as Smart Actuator–Sensor System. *Applied Sciences*. 2024; 14(18):8523. <https://doi.org/10.3390/app14188523>.
- [12] BN, P., Yelchuri, H., Shastri, V., & Prabhakar, T. V. (2022). PreMa: predictive maintenance of solenoid valve in real-time at embedded edge-level. *arXiv preprint arXiv:2211.12326*.
- [13] Li X, Zhang L, Tan T, Wang X, Zhao X, Xu Y. Multi-Sensor Data Fusion and Vibro-Acoustic Feature Engineering for Health Monitoring and Remaining Useful Life Prediction of Hydraulic Valves. *Sensors (Basel)*. 2025 Oct 11;25(20):6294. doi: 10.3390/s25206294. PMID: 41157348; PMCID: PMC12568100.
- [14] Z. Wang, W. Li, C. Feng, B. Qiu and X. Lu, "Hydraulic Valve Core Fault Diagnosis Applied with a Data-driven Approach," 2025 IEEE International Instrumentation and Measurement Technology Conference (I2MTC), Chemnitz, Germany, 2025, pp. 1-6, doi: 10.1109/I2MTC62753.2025.11079224.
- [15] M. Ferri, F. Mancarella, L. Belsito, A. Roncaglia, J. Yan, A. A. Seshia, K. Soga, J. Zalesky, "Strain sensing on steel surfaces using vacuum packaged MEMS resonators", *Procedia Eng.*, vol. 5, pp. 1426–1429, Jan. 2010.
- [16] L. Belsito, M. Ferri, F. Mancarella, A. Roncaglia, J. Yan, A. A. Seshia, K. Soga, "High-resolution strain sensing on steel by Silicon-On-Insulator flexural resonators fabricated with chip-level vacuum packaging", in *Proc. 17th Int. Conf. Solid-State Sens. Actuators Microsyst.*, Barcelona, Spain, Jun. 2013, pp. 992–995.
- [17] L. Belsito, M. Ferri, F. Mancarella, L. Masini, J. Yan, A. A. Seshia, K. Soga, A. Roncaglia, "Fabrication of high-resolution strain sensors based on wafer-level vacuum packaged MEMS resonators", *Sens. Actuators A, Phys.*, vol. 239, pp. 90–101, Mar. 2016.
- [18] M. Ruggeri, G. P. Massarotti, L. Belsito, A. Roncaglia, Indirect Pressure Measurement on Hydraulic Components Through New MEMS Strain Sensors, *ASME/BATH 2017 Symposium on Fluid Power and Motion Control*, doi: 10.1115/FPMC2017-4355.
- [19] G.P. Massarotti, C. Ferrari, E. Codina Macia and M. Ruggeri. "General Lecture: Mems Sensors In Hydraulics, An Opportunity To Create Smart Components." (2020), 12th International Fluid Power Conference (12. IFK). Dresden, October 12 – 14, 2020. <https://doi.org/10.25368/2020.114>.
- [20] M. Ruggeri, L. Belsito, F. Bosi, P. Cominetti, M. Ferri, M. Ferri, A. Roncaglia, Advancements in Noninvasive Pressure Sensing in Hydraulic Components, *ASME/BATH 2023 Symposium on Fluid Power and Motion Control*, Sarasota FL, November 2023, doi:10.1115/FPMC2023-109140.

Bridging 1D System Modeling and 3D CFD for Hydraulic Slack Adjuster Design: A Decoupled Co-Simulation Approach

Vignesh Jeganathan^{1,*}, Chris Imholte², Ashutosh Pandey¹, Bipin Kashid²

¹*Simerics, Inc.*

²*Parker Hannifin Corporation*

**Corresponding Author, E-mail: vj@simerics.com*

Abstract

The design of safety-critical hydraulic braking subsystems, such as slack adjusters, necessitates the accurate prediction of complex transient interactions between multi-body kinematics and high-fidelity fluid dynamics. While traditional one-dimensional (1D) lumped-parameter models offer system-level insights, they fail to resolve localized phenomena like cavitation, flow separation, and transient pressure spikes. Conversely, fully coupled 3D computational fluid dynamics (CFD) is often computationally prohibitive for extensive design-space exploration. This paper proposes a technically robust, decoupled 1D-3D co-simulation methodology that leverages the strengths of both modeling paradigms to optimize hydraulic performance and reliability. In the proposed framework, a 1D system model is utilized to perform a comprehensive parametric study across a wide range of operating pressures and accumulator configurations, identifying critical transients and near-failure edge cases. These 1D-derived results are then mapped as time-dependent boundary conditions to a high-fidelity 3D CFD model within the Simerics MP environment. A novel contribution of this work is the implementation of a tailored modeling strategy for multi-body fluid-structure interaction (FSI), where contact formulations and damping factors—guided by 1D kinematic data—are used to stabilize the simulation of rapid cylinder collisions. The re-

sults indicate that the decoupled approach resolves complex flow and pressure interactions driven by component geometry, which are not captured by one dimensional analysis, yet preserves acceptable computational overhead. This methodology provides a scalable framework for industrial design teams to shorten development cycles and enhance the robustness of complex hydraulic systems through early-stage detection of failure modes.

Keywords: Slack Adjuster, Brakes, Computational Fluid Dynamics, Valve Body collision.

1 Introduction

The hydraulic slack adjuster is a multi-component device designed to maintain a consistent, predetermined residual pressure downstream of the device following the decay of an upstream input [1]. This residual pressure ensures that a finite trapped fluid volume persists within the braking caliper circuit, which in turn preserves a controlled offset distance between the brake pad and rotor interface [2]. The functional consequence of this offset is repeatable, reliable brake engagement and consistent pedal feel at the operator input interface [3].

The primary structural element of the device is a hydraulically balanced valve bodies, effectively an equal-area cylinder with custom end caps, that is spring-biased toward the downstream end cap [4]. This configuration divides the internal volume into two hydraulically distinct regions, as shown in figure. 1. The upstream region, referred to as the apply side, receives pressure input from a controlling brake valve. The downstream region, interfaces directly with the hydraulic braking caliper also referred as accumulator, due to model similarity. Under normal apply conditions, pressure is communicated across Body 1 via a check valve, which permits flow from the upstream brake valve input to the accumulator region whenever a positive pressure differential exists in that direction.

The trapping and release of accumulator volume is governed by a two-component sub-assembly consisting of a remote pressure-referenced sliding piece Body 3 and a floating component, Body 2. Body 3 uses atmospheric reference pressure on one face and upstream supply pressure on the opposing face. When upstream pressure rises sufficiently above atmospheric, the sliding piece displaces to the right, no longer interacting mechanically with the floating Body 2. Under normal conditions, Body 2 is seated against Body 1, sealing the accumulator region from the upstream region. A pressure differen-

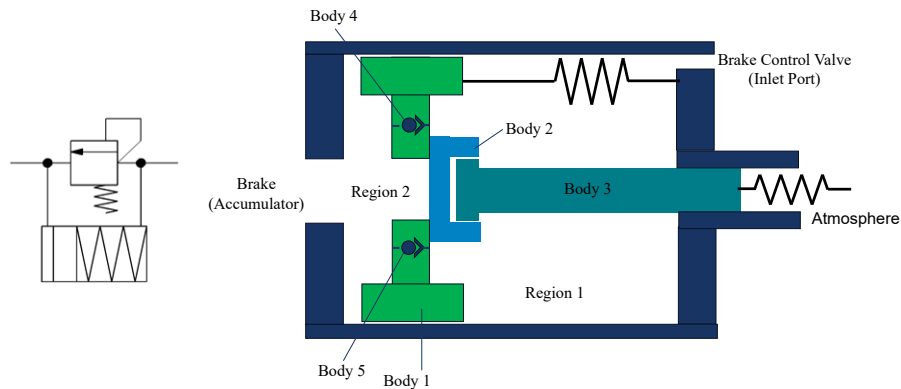


Figure 1 Body 1 denotes a rodless cylinder with equal areas, biased by a spring. Bodies 2 and 3 denote a modified form of a sequence valve. Body 4 consists of ball valves that permit one-way flow from region 1 to region 2. Collision and separation of these bodies 1,2 and 3 opens or closes fluid channels between the regions 1 and 2.

tial of sufficient magnitude unseats the Body 2, permitting accumulator fluid to return to the Brake Control Valve, which occurs during brake release. Once the Body 3 senses that upstream pressure has dropped equal to the spring force, it displaces back to the left, returning Body 2 to its fully seated position and sealing the accumulator volume. The resulting pressure difference across the Body 1 is then sufficient to produce Body 1 displacement to the right, with the system reaching force equilibrium.

Design-space exploration for this mechanism is governed less by the isolated performance of individual components and more by the timing and sequencing of interactions between them. The critical event is the sealing of trapped accumulator volume at a specified upstream pressure threshold, which initiates the Body 1 motion responsible for slack absorption. Two distinct failure modes bound the acceptable design space. If the Body 2 seals late, the trapped volume and corresponding pressure rise are insufficient to achieve full Body 1 stroke to the right; in the limiting case, no slack is absorbed and the braking device does not function as intended. Conversely, premature sealing prevents adequate reduction of accumulator pressure prior to brake release, resulting in residual caliper engagement and a separate class of functional failure where the brake rotors do not disengage. The design space is therefore defined by acceptable timing windows rather than single steady-state performance targets.

Within this timing-governed framework, the relevant design parameters are the spring rates and area ratios of the pressure-referenced sliding piece (Body 3), the spring rates and available displacement of the spring biasing Body 1, and the geometric relationship between the sliding piece and Body 2 that determines the mechanical interaction. These parameters interact such that no single component can be sized independently; the area ratio sets the pressure at which the sliding piece actuates, which in turn determines when Body 2 is driven to reseat, and the spring must provide sufficient restoring force over the available stroke to complete sealing under worst-case pressure decay conditions. Bounding this parameter space through 1D simulation prior to hardware fabrication is therefore essential for risk reduction.

For a given Body 1 bore area and required stroke, there exists a minimum sealed volume below which the residual trapped pressure rise is to deliver the net force required for full displacement. When the trapped volume falls below this threshold, the accumulator pressure decreases to equilibrium before Body 1 completes its stroke, and the lack of absorption is incomplete. The robust design therefore requires that the nominal trapped volume be sized with sufficient margin above this threshold to accommodate the combined worst-case contributions of leakage and mechanical load variation across the full operating envelope.

The 1D lumped-parameter model provides an effective and computationally efficient tool for system-level design-space exploration, timing assessment, and parametric sensitivity studies. However, the governing physics of the Body 2 sub-assembly include several locally three-dimensional phenomena that are not reliably captured within a lumped formulation, and which are consequential to the accurate prediction of device timing and dynamic stability [5].

The Body 2 features a unique surface profile and seat geometry that produces strong spatial gradients in pressure and velocity in the near-seat region due to geometry space limitations. The net axial hydraulic force acting on the Body 2 is highly sensitive to the instantaneous development of the flow jet, the location of flow separation from the seat geometry, and the degree of pressure recovery accumulator of the minimum flow area. These effects are collectively sensitive to the mass ratio of the moving element and to the precise geometric transition through small lift values. In a 1D representation, these phenomena are necessarily collapsed into a single effective flow area and a lumped jet force coefficient, which cannot capture the variation in net force with lift or the potential for flow-force-induced instability in the low-lift regime beyond a single component.

Additionally, the leakage behavior of the assembly is governed by multiple lift-dependent flow paths defined by the proximity of complex mating surfaces. The controlling restriction along these paths can shift between geometric features edges and clearance gaps as the Body 2 moves, and the resulting flow may be transitional with significant viscous influence at small clearances. 1D simulation without physical test data of Body 2 performance cannot represent the migration of the controlling restriction or the associated changes in effective leakage area with lift. The valve timing and fluid damping behavior are similarly influenced by the transient filling and emptying of local volumes adjacent to the Body 2 and by the unsteady pressure field generated during rapid seating events. As such, the moving boundaries, transient jet development, and viscous damping through narrow passages require spatially resolved simulation to predict reliably. Three-dimensional CFD resolves the pressure distribution across the Body 2 surface at each instant, enabling accurate computation of flow forces and leakage as functions of lift, pressure differential, and fluid state. This level of fidelity is necessary to bound the sensitivity of device timing to geometry-driven effects that the 1D model cannot resolve, and to provide confidence that the predicted operating mode, successful slack absorption versus failure, is robust to geometric and operating condition variation.

2 Decoupled Co-Simulation Framework

A fully coupled, time-resolved co-simulation between a system-level 1D model and a transient 3D CFD solver is computationally prohibitive for iterative design analysis. The indirect coupling framework adopted here separates the two modeling domains by leveraging the inherent time-scale and spatial-scale separation between system-level sequencing behavior and local component physics. The 1D model is used to generate transient boundary condition histories such as pressures, flow rates, and component state sequences across the operating envelope. These boundary conditions are then applied to targeted 3D CFD simulations of the Body 2 sub-assembly, which resolve the local flow physics that cannot be represented in the lumped formulation. The two models are not solved simultaneously; rather, results are exchanged in a defined, structured manner that preserves physical consistency while enabling each model to be applied where it is most effective.

This approach is motivated by industrial practicality. The 1D model can be exercised rapidly across large parameter spaces, making it suitable for design-space exploration, sensitivity studies, and identification of candidate

operating points that represent edge conditions or failure-adjacent scenarios. The 3D CFD model is then applied selectively to those conditions, providing high-fidelity resolution of the local physics at the cases most relevant to design risk. The result is a workflow that achieves the predictive capability of 3D CFD where it is needed without incurring the computational cost of full-envelope 3D simulation.

The boundary condition mapping procedure translates the time-resolved outputs of the 1D model into inlet and outlet pressure or flow rate histories suitable for driving the 3D CFD domain. Key outputs extracted from the 1D simulation include the upstream apply-side pressure trace, the accumulator caliper circuit pressure, and the predicted state sequence of the Body 2 and sliding piece specifically, the timing of Body 2 unseating, the duration of the open flow period, and the predicted reseal event. These traces establish the expected operating window for the transient CFD simulation.

Validation of the co-simulation framework does not seek pointwise quantitative agreement between 1D and 3D results across all outputs. Instead, consistency is assessed against a structured set of behavioral criteria that collectively confirm whether the two models predict the same operating mode and whether discrepancies identify specific physical mechanisms requiring higher-fidelity representation. The first criterion is general displacement agreement: whether the 1D and 3D models both predict Body 1 displacement, or both predict failure to displace, for a given set of boundary conditions. This high-level consistency check validates that the dominant force balance of pressure acting on Body 1 area against spring and mechanical load is captured correctly at the system level, and that the continuity and compressibility accounting in the 1D model is adequate for the purpose of mode prediction. The second criterion addresses leakage representation. In the 1D model, leakage across the Body 3 annular region is represented by an equivalent restriction characterized by an effective area and laminar leakage with varying length leakage assumption. The 3D CFD model resolves the actual evolving gap geometry and viscous pressure loss along the leakage path. Consistency in the rate and shape of accumulator pressure decay between the two models supports the adequacy of the 1D restriction parameterization.

The third criterion is Body 2 reseal behavior. Reseal is governed by the net force balance on the Body 2 as the upstream pressure decays to the atmospheric reference threshold. The 1D model predicts reseal using lumped pressures and simplified hydraulic force assumptions. The 3D CFD simulation reveals whether the near-seat flow field generates stabilizing or destabilizing axial forces at low lift and whether residual leakage through the

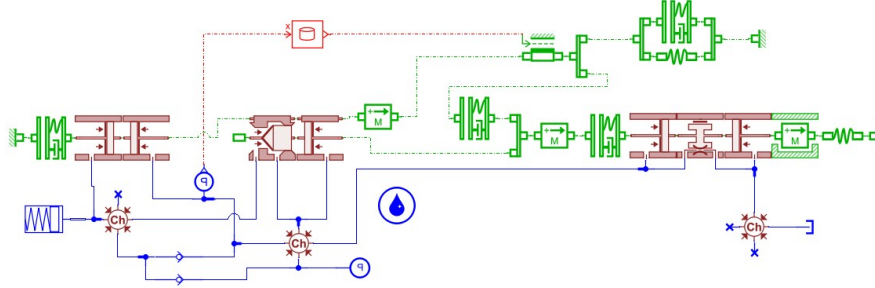


Figure 2 Configuration of the 1D simulation model in modeling software.

shrinking gap inhibits the pressure recovery necessary for complete seating. Agreement in reseal timing and completeness between the two models increases confidence in the sign and magnitude of the dominant modeled forces; disagreement identifies conditions under which flow-force nonlinearity or transient damping effects must be represented with higher fidelity to bound the design space reliably.

The fourth criterion is event sequencing and relative timing. The key event markers such as onset of Body 2 unseating, duration of open flow period, initiation of Body 1 motion, peak trapped-volume pressure, reseal time, and accumulator pressure recovery are compared between the 1D and 3D results. The 1D model provides efficient causal sequence prediction across parameter sets; the 3D model provides a higher-fidelity depiction of the transient pressure evolution associated with real leak-path geometry and jet development. Modest timing offsets between the two models are treated as diagnostic information rather than validation failures, indicating the degree to which lift-dependent leakage, transient damping, or flow-force nonlinearity is driving sequencing sensitivity and therefore must be resolved at higher fidelity to support robust design decisions. Upon iteration it results in a decoupled co-simulation loop of 1D simulation to 3D simulation, where the next iteration utilizes improved parameters for leakage, damping or other parameters.

3 1D simulations

The co-simulation framework described in Section 2 is exercised against a set of targeted case studies selected to expose operating conditions at or near the boundaries of acceptable device performance. 1D simulations are carried out using the simulation setup shown in figure 2. These cases were chosen

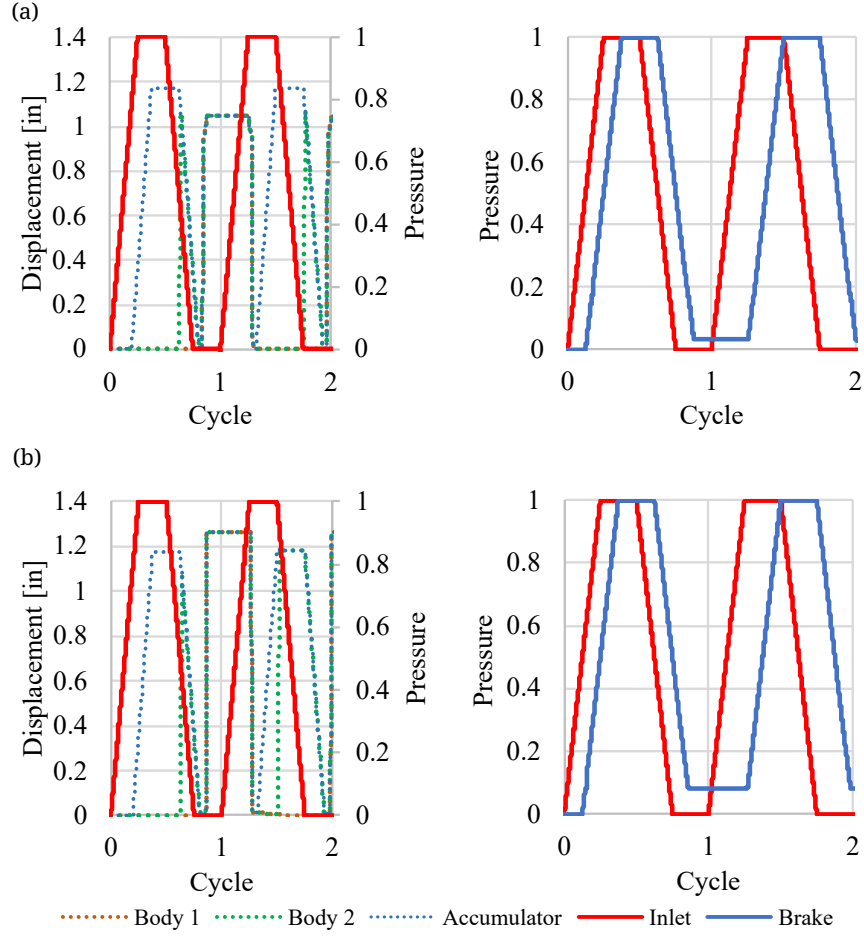


Figure 3 Valve body motion and accumulator (brake) pressure under different inlet pressure cycles for (a) the initial accumulator volume and (b) twice the accumulator volume. Time is non-dimensionalized using cycle time and pressure is normalized by the highest pressure used in the inlet.

specifically because they represent failure modes or edge conditions that are not fully observable in the 1D model alone, and for which the 3D CFD results provide either confirmation or refinement of the predicted system behavior.

3.1 Minimum Trapped Volume for Full Displacement of Body 1

As established in Section 2, there exists a minimum sealed accumulator volume below which the pressure remaining during the trapping event is insufficient to drive the Body 1 to full stroke. This case study independently examines the sensitivity of Body 1 displacement to trapped volume and minimum reapplication pressure. The 1D model was used to sweep the relevant parameters such as accumulator pressure decay rate, and minimum input pressure to identify the boundary conditions under which the predicted trapped volume approaches the minimum displacement threshold. These boundary condition sets are now prepared for implementation in the 3D CFD model to evaluate whether the locally resolved leakage behavior and the evolution of transient pressure align with the 1D predictions at the threshold condition. Of particular interest is whether the 1D equivalent restriction model adequately represents leakage at the low pressure differentials characteristic of the late stages of the trapping event, where small differences in leakage rate have the greatest influence on the final sealed volume. Results from this case study, per figure 2, inform the required volume margin in the nominal design and identify which operating condition variables contribute most significantly to the displacement risk. These findings directly support the sizing guidance established during design-space exploration and provide a quantified basis for design robustness decisions.

In addition, due to the stored energy within the slack adjuster Body 1 upon displacement, there is a minimum apply pressure required to return the Body 1 to the no displacement location. As seen in figure 4a, the slack adjuster has been sized to require 5 bar of brake valve pressure (barring losses) to effectively return the Body 1 to zero displacement (no slack volume). This minimum brake valve pressure input to completely return the Body 1 is a requirement driven through the brake valve subsystem which converts brake command into hydraulic actuation. Effectively sizing this pressure in relation to the constraint represents the minimum pedal displacement from an operator and as such is critical to determine.

3.2 Excessive Leakage Across the Body 3

The annular leakage path associated with the sliding piece Body 3 represents a critical interface in the device. Under nominal conditions, leakage through this path is bounded by the clearance geometry and fluid viscosity, and its effect on trapped volume integrity is accounted for in the 1D model through an equivalent restriction. However, under conditions of elevated clearance,

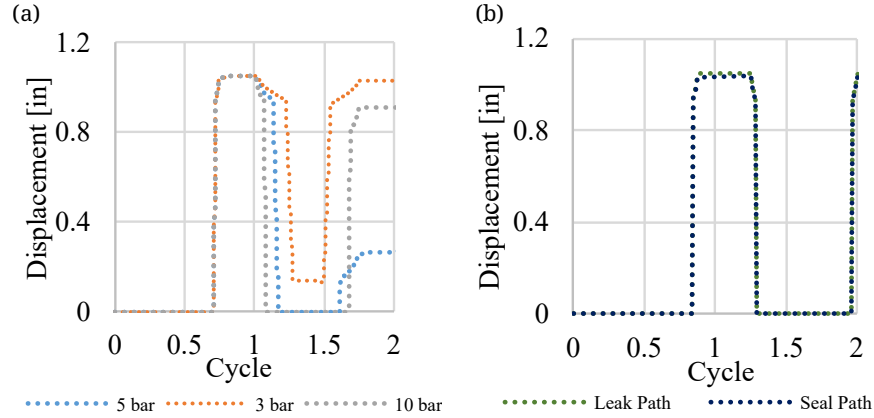


Figure 4 (a) Parametric study of constant caliper volume for various brake valve apply pressures. (b) Displacement of piston that retains slack volume for various leakage amount around the Body 3. Time is non-dimensionalized using cycle time.

arising from manufacturing tolerance stack-up, thermal expansion, or wear, the leakage rate can increase to the point where the force balance is disrupted by local effects or flow rate is of unacceptable magnitude.

In an ideal configuration, a frictionless seal would be represented such that no interface exists between the external atmospheric reference and the upstream pressure. A further development of the model is to incorporate either seal friction or annular leakage. For the present case, the seal was omitted, as illustrated in figure 4b, because a seal failure would exhibit a behavior analogous to that of significant annular leakage.

4 3D Simulations

After performing multiple 1D analyses, we identified a failure mode that must be investigated using Computational Fluid Dynamics (CFD). The modeling process includes defining suitable numerical setups to capture flow phenomena such as cavitation, turbulence, and leakage. Additionally, collision events are modeled carefully to determine valve timing with high accuracy. Significant effort has been devoted to achieving an optimal mesh and timestep size for the 3D CFD simulations.

4.1 Computational Fluid Dynamics model

In a CFD simulation, the Navier–Stokes equations that describe fluid motion are solved numerically to obtain the velocity and pressure fields within the domain.

The spatially averaged conservation equations for mass and momentum can be expressed in integral form as:

$$\frac{\partial}{\partial t} \int_{\Omega(t)} \rho d\Omega + \int_{\sigma} \rho(v - v_{\sigma}) \cdot n d\sigma = 0, \quad (1)$$

and

$$\frac{\partial}{\partial t} \int_{\Omega(t)} \rho v d\Omega + \int_{\sigma} \rho v((v - v_{\sigma}) \cdot n) d\sigma = \int_{\sigma} \tau \cdot n d\sigma - \int_{\sigma} P n d\sigma + \int_{\Omega} f d\Omega. \quad (2)$$

In the equations above, ρ denotes the density of the working fluid, v is the spatially averaged velocity vector, v_{σ} is the velocity of the moving reference frame, τ is the deviatoric stress tensor, P is the pressure, and f represents the body force, such as gravity. The symbols σ and Ω under the integral signs denote surface and volume integrals, respectively.

Turbulence is modeled using the standard $\kappa - \epsilon$ two-equation model together with a wall function, expressed as:

$$\begin{aligned} \frac{\partial}{\partial t} \int_{\Omega(t)} \rho \kappa d\Omega + \int_{\sigma} \rho((v - v_{\sigma}) \cdot n) \kappa d\sigma &= \int_{\sigma} \left(\mu + \frac{\mu_t}{\sigma_{\kappa}} \right) (\nabla \kappa \cdot n) d\sigma \\ &+ \int_{\Omega} (G_t - \rho \epsilon) d\Omega, \end{aligned} \quad (3)$$

and

$$\begin{aligned} \frac{\partial}{\partial t} \int_{\Omega(t)} \rho \epsilon d\Omega + \int_{\sigma} \rho((v - v_{\sigma}) \cdot n) \epsilon d\sigma \\ = \int_{\sigma} \left(\mu + \frac{\mu_t}{\sigma_{\epsilon}} \right) (\nabla \epsilon \cdot n) d\sigma + \int_{\Omega} \left(c_1 G_t \frac{\epsilon}{\kappa} - c_2 \rho \frac{\epsilon^2}{\kappa} \right) d\Omega. \end{aligned} \quad (4)$$

Here, κ denotes the spatially averaged turbulent kinetic energy, ϵ the spatially averaged dissipation rate of turbulent kinetic energy, μ_t the turbulent (eddy) viscosity, c_1 and c_2 are closure constants of the model, and G_t is the production term of turbulent kinetic energy.

To represent cavitation and aeration, the cavitation model of Singhal et al. [6] is employed in Simerics-MP+. This model incorporates real liquid properties, cavitation, aeration, and liquid compressibility. The cavitation vapor distribution is given by the following formulation:

$$\frac{\partial}{\partial t} \int_{\Omega(t)} \rho f_v d\Omega + \int_{\sigma} \rho ((v - v_{\sigma}) \cdot n) f_v d\sigma = \int_{\sigma} (D_{f_v} + \frac{\mu_c}{\sigma_{f_v}}) (\nabla f_v \cdot n) d\sigma + \int_{\Omega} (R_e - R_c) d\Omega \quad (5)$$

Here, f_v denotes the vapor volume fraction of the working fluid, D_{f_v} is the diffusivity of the vapor mass fraction, and σ_{f_v} is the turbulent Schmidt number. In this work, D_{f_v} and σ_{f_v} are taken to be equal to the mixture viscosity and unity, respectively. The vapor generation rate R_e and the condensation rate R_c are defined as:

$$R_e = C_e \frac{\sqrt{K}}{\sigma_l} \rho_l \rho_v \left(\frac{2(p - p_v)}{3\rho_l} \right)^{1/2} (1 - f_v - f_g) \quad (6)$$

$$R_c = C_c \frac{\sqrt{K}}{\sigma_l} \rho_l \rho_v \left(\frac{2(p - p_v)}{3\rho_l} \right)^{1/2} f_v \quad (7)$$

with model constants $C_e = 0.02$ and $C_c = 0.01$. The mixture density is then obtained from:

$$\frac{1}{\rho} = \frac{f_v}{\rho_v} + \frac{f_g}{\rho_g} + \frac{1 - f_v - f_g}{\rho_l} \quad (8)$$

The non-condensable gas (NCG) can be treated either as a purely free gas or as a combination of free and dissolved gas. Simerics-MP+ provides several models that account for both contributions of NCG; further details are given in the software manual [7]. For this simulation, the equilibrium dissolved gas model is adopted, where it assumes that the mass fraction of the total non-condensable gas remains constant, but a part of it is dissolved into the liquid to instantly satisfy the local equilibrium condition.

The governing equations are discretized using the finite volume method and solved with a pressure-based algorithm. Spatial discretization uses a second-order upwind scheme, while time integration is performed with a second-order explicit scheme. The pressure-velocity coupling is resolved using the SIMPLE-S (Semi-Implicit Method for Pressure Linked Equations-Simerics) algorithm [8, 9].

4.2 Translational dynamics model for collisions involving multiple bodies

Simerics-MP+ incorporates a rigid-body motion model that neglects fluid–structure interaction. This class of motion is governed by the linear momentum conservation equation for translational rigid-body dynamics.

The resulting governing equation from linear momentum conservation is

$$m \frac{d^2 s}{dt^2} = F_{hydrodynamic} - F_{damping} - F_{spring} - F_{friction} + F_{penalty} \quad (9)$$

Within Simerics-MP+, this equation is solved for the displacement of the moving body simultaneously with the flow and cavitation governing equations.

Here, m denotes the mass of the moving body, and s is its displacement under the net force. $F_{hydrodynamic}$ is the hydrodynamic load arising from pressure and shear stresses. $F_{damping}$ is a damping force that can originate from a spring (if present) or any other external damping device. F_{spring} is the spring force, generally opposing the motion, and $F_{friction}$ is the frictional resistance due to contact between the moving and stationary components. $F_{penalty}$ is an additional force activated upon detection of body–body contact and is expressed as

$$F_{penalty} = kx - cv_{rel} \quad (10)$$

where k is the spring constant, typically chosen as a relatively large positive value, and x is the overlap between bodies, usually on the order of nm . The damping term, given by the product of the damping coefficient c and the relative velocity v_{rel} , serves to reduce jitter or oscillations during impact events.

4.3 Model details

The computational mesh used in this work is shown in figure 5a. At the inlet of the slack adjuster, the pressure is prescribed to vary cyclically over time (see figure 5c). This periodic pressure excitation drives the motion of the valve bodies, leading to their interaction and impact with one another as well as with the three adjacent walls depicted in figure 5a. The configuration also comprises a spring-loaded accumulator, which is charged during phases of elevated inlet pressure, and a drain port that allows fluid to flow in or out depending on the Body 3 displacement. A zoomed-in representation of the

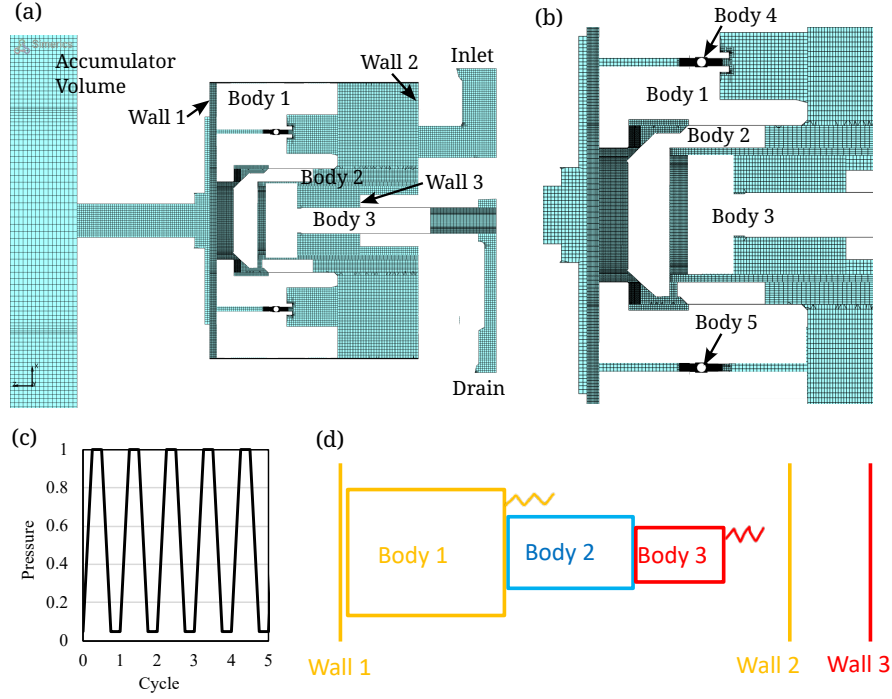


Figure 5 (a) Full CFD mesh showing the accumulator, inlet and drain ports and the valve bodies. (b) Magnified view showing valve bodies and the corresponding deforming zones. (c) Cyclical pressure applied at the inlet. (d) Free-body diagram of the valve bodies represented in 1D for collision modeling.

valve bodies is provided in figure 5b. The ball valves (Body 4 and Body 5) allow flow exclusively from the inlet toward the accumulator. As a result, during the high-pressure inlet phase they facilitate charging of the spring-loaded accumulator, while during the low-pressure inlet phase they prevent any reverse flow from the accumulator back into the inlet.

To simulate collisions among valve components, we represent the 3D bodies and walls using a 1D abstraction, as shown in figure 5d. During operation, the Body 1, Body 2, and Body 3 may impact each other. The Body 1 and Body 3 are preloaded by springs. The Body 1 is constrained by wall 1, wall 2 and the Body 2. The Body 2 is constrained by the Body 1 and the Body 3. The Body 3 is constrained by the Body 2 and wall 3. Because the positions of the valve bodies are mutually dependent, we employ the penalty method

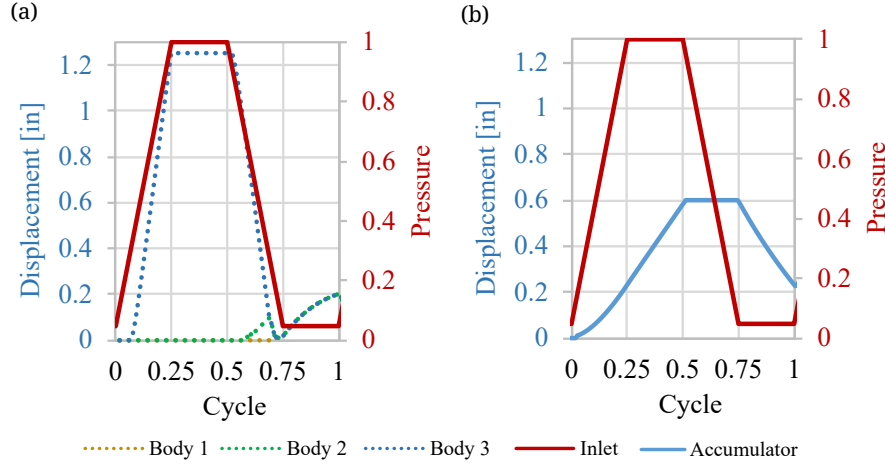


Figure 6 (a) Valve bodies' displacements from their resting positions and (b) accumulator displacement as a function of time for the initial cycle. Inlet pressure applied during the simulation is shown in the secondary-y for reference. Time is non-dimensionalized using cycle time and pressure is normalized by the highest pressure used in the inlet.

(see eqn. 10) to decouple them into individual free-body diagrams, each with its corresponding force balance (see eqn. 9).

4.4 Results and discussion

In this section, we analyze the outcomes of the 3D CFD simulation to gain a detailed understanding of the system using the design finalized during the 1d analysis. As described in the previous section, a pressure pulse is applied at the inlet pressure boundary. This pressure varies cyclically and reflects the actual operating conditions of the slack adjuster.

During the initial cycle of inlet pressure variation, as the inlet pressure increases, Body 3 is driven toward wall 3, as illustrated in the displacement plot in figure 6a, because the fluid force exceeds the restoring force of the spring attached to Body 3. This behavior can be observed more distinctly in the contour plots in figure 7a,b,c. Over this period, the accumulator becomes fully charged, which is indicated by its displacement rising to the maximum value, as shown in figure 6b.

During the inlet pressure ramp-down phase of the first cycle, the spring force acting on Body 3 exceeds the fluid pressure, causing it to move away from wall 3. This behavior is quantified in figure 6a and illustrated in fig-

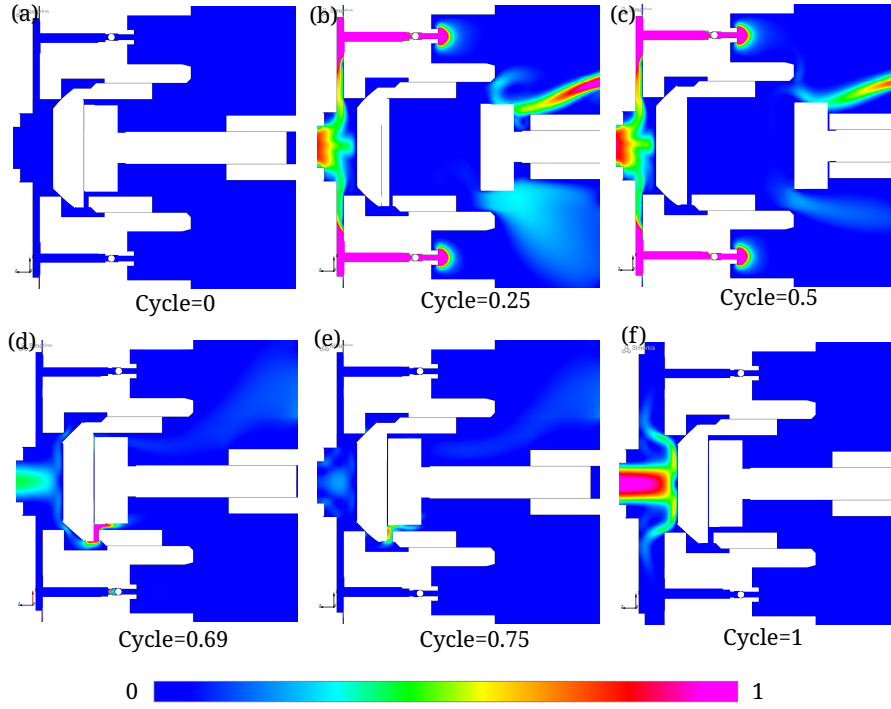


Figure 7 (a-f) 3D contour of velocity magnitude showing the valve body movements for different times in the initial cycle. The velocities are in ms^{-1} .

ures 6d and e. During this interval, the accumulator discharges (see figure 6b) because its pressure is higher than that at the inlet. As the ball valve restricts direct flow from the accumulator to the inlet, the fluid instead drives Body 2, separating it from Body 1 and creating a new flow path from the accumulator to the inlet through the gap that opens between bodies 1 and 2. This is reflected in figure 6a, where Body 2 begins to move approximately Cycle=0.6, and is further depicted in the contour plot figure 7d. This sequence of events puts bodies 2 and 3 on a collision trajectory. At around Cycle=0.69, they collide and Body 3 forces Body 2 back toward Body 1, thereby closing the flow channel formed between bodies 1 and 2. figure 6e provides a detailed visualization of this event. Once this leakage path is eliminated, the outflow from the accumulator begins to displace all bodies at approximately Cycle=0.75, continuing until the end of the cycle. The final positions of the valve bodies at the end of the first cycle are shown in figure 7f.

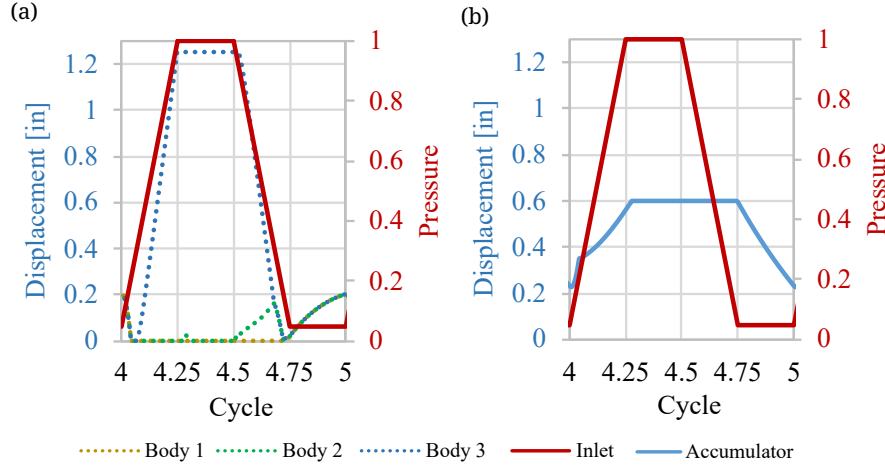


Figure 8 (a) Valve bodies' displacements from their resting position and (b) accumulator displacement as a function of time for the final cycle. Inlet pressure applied during the simulation is shown in the secondary-y for reference. Time is non-dimensionalized using cycle time and pressure is normalized by the highest pressure used in the inlet.

After the first cycle, the accumulator is primed and the behavior of the valve bodies and the accumulator in the subsequent cycles is periodic. Hence, we look at the results of the last inlet pressure cycle we simulated. Unlike the first cycle, the valve bodies and accumulators are not in their resting position, but offset by some distance from the wall 1, which coincides with the end state of the first cycle. This can be observed by comparing the valve body and accumulator positions in the initial cycle figure 6 and the final cycle figure 8. As the inlet pressure ramps up, the increasing inlet pressure combined with the spring force pushes the valve bodies 1, 2 and 3 closer to the wall 1 i.e. toward their resting positions like evident in the figure 8a. Like the first cycle, once the inlet pressure increases enough to overcome the spring of the Body 3, the Body 3 starts to move towards the wall 3 as illustrated in the figures 8b,c. Unlike the first cycle (figure 6), the accumulator gets fully charged very early in the high pressure phase in the final cycle (figure 8b). Consequently, Body 2 starts moving earlier as seen in the figure 8a.

During the ramp-down stage of the final cycle, Body 3 returns to its original position as the spring force overtakes the fluid force. As Body 3 moves back, it collides with Body 2, pushing it toward Body 1 and thus closing the leakage path between Bodies 1 and 2. The contour plots in figure 9d,e,f

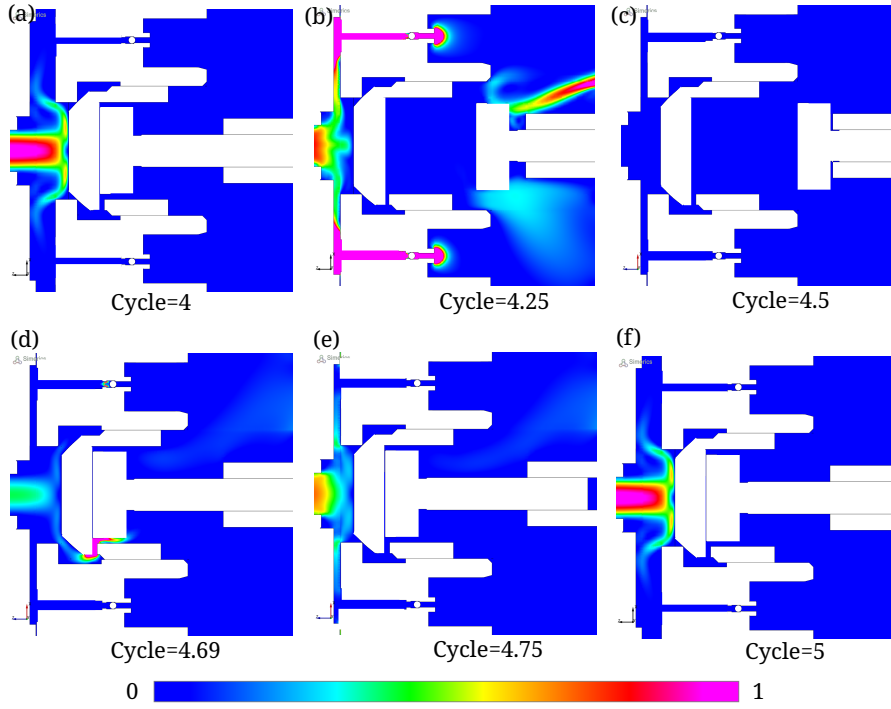


Figure 9 (a-f) 3D contour of velocity magnitude showing the valve body movements for different times in the final cycle. The velocities are in ms^{-1} .

qualitatively illustrate this behavior, and figure 8 shows the corresponding quantitative results. This mechanism is analogous to that observed in the initial cycle, with the key difference that in the later cycles Body 2 travels a greater distance from its starting position because the accumulator is charged more quickly. As in the first cycle, all the valve bodies are displaced toward wall 3 by a certain amount. After this, the entire sequence repeats periodically.

5 Comparison of 1d and 3d simulations

It should be emphasized that the quantitative results reported for the 1D and 3D simulations are not in agreement. For example, the valve body displacements obtained from the 1D simulations, shown in figure 3, differ from the displacements predicted by the 3D simulations, which are presented in fig-

ures. 6 and 8. The 3D methodology solves the full flow field and explicitly accounts for turbulence and cavitation, effects that are not included in the 1D model. As a result, the opening and closing times of the valve bodies can differ significantly between the two approaches. In a complex valve system such as this, even small variations in timing can lead to large differences in the overall displacements.

Nonetheless, the global system behavior is comparable in both approaches. For instance, accumulator priming during the first cycle is captured in both the 1D and 3D simulations. Another example is Body 3 driving Body 2 to close the leakage path between the valve bodies, followed by the subsequent motion of all valve bodies; this sequence is observed in both the 1D and 3D results, as illustrated in figures. 7 and 8, respectively.

6 Industrial Impact

Typical new-product development (NPD) workflows in the off-highway hydraulics industry can stretch for many months or years, and require 1-3 prototyping and “testbed” builds to validate novel elements. These prototypes and testbeds consume resources at an accelerated rate relative to increased novelty in design. Conservatively, organizational resource impact may be 50,000 USD or more for a prototype or testbed with significant new component design validation. In addition, some phenomena are difficult or impossible to observe during physical testing. In the present case, equipment, time, instrumentation, and personnel limitations put full validation of unconstrained edge-case poppet motion out of reach. Typically, valuable time is consumed on adjusting validation goals and plans to mitigate for resource constraints and missing or compromised data, which in turn weakens support of needed qualitative and quantitative design inferences, leading to mixed results or extensive post-processing.

In the study presented here, a fraction of typical prototyping and test resources were needed, and these resources were targeted on risk areas that would be exceedingly difficult to observe and manipulate in physical testing. Rapid post-processing and direct comparisons of simulation results fully supplanted typical pre-launch prototyping cycles. This team anticipates the launch program will proceed directly to testing First Article parts from a production-intent supply chain.

7 Conclusion

A well-planned simulation strategy should enable ambitious concept development even in mature hydraulic components and systems. A decoupled 1D–3D approach is especially effective: 1D models support rapid ideation, subsystem/system studies, and fast DoE to explore edge cases and failure drivers, while 3D simulations provide high-fidelity insight into component-level physics and application behavior.

Using each tool where it fits in the design cycle reduces development time and minimizes late-cycle surprises during testing. The available simulation tools can replace portions of early prototyping - when risk and design churn are highest - while enabling teams to evaluate concepts that would be difficult to assess with traditional methods. Model fidelity can be increased incrementally as the design matures, focusing effort on the highest-risk elements with clear requirements and validation criteria.

Finally, linking 3D visuals and results with 1D system behavior improves communication across cross-functional teams and brings Subject Matter Experts into design reviews and Design Failure Mode and Effect Analysis discussions earlier, leading to more efficient and effective pre-production verification testing.

References

- [1] T. H. Engle, “Hydraulic slack adjuster and brake circuits therefor,” May 11 1976. US Patent 3,955,370.
- [2] F. W. Berwanger, “Aircraft brake piston hydraulic adjuster assembly,” Jan. 23 1996. US Patent 5,485,902.
- [3] R. E. Landgraf, “Automatic brake adjuster mechanism,” June 1 1971. US Patent 3,581,849.
- [4] M. A. Salam, *Hydraulic Cylinders*, pp. 257–286. Singapore: Springer Singapore, 2022.
- [5] J. F. Blackburn, G. Reethof, and J. L. Shearer, “Fluid power control,” (*No Title*), 1960.
- [6] A. K. Singhal, M. M. Athavale, H. Li, and Y. Jiang, “Mathematical basis and validation of the full cavitation model,” *Journal of Fluids Engineering*, vol. 124, no. 3, pp. 617–624, 2002.
- [7] Simetrics Inc., “Simetrics-MP+ user manual,” 2024.
- [8] A. Pandey, J. Wang, D. Maiti, and Y. Jiang, “Performance and validation of a segregated pressure-based solver for computations of low and high-speed compressible flows,” in *AIAA SCITECH 2023 Forum*, p. 2274, 2023.
- [9] S. Taghizadeh, K. C. Ng, J. Horen, and S. Dhar, “A 3d computational fluid dynamics and acoustics simulation approach for noise mitigation prediction in gerotor pumps,” tech. rep., SAE Technical Paper, 2024.

1D System Modeling and 3D CFD Co-Simulation Methodology for Analyzing Transient Behavior in Hydraulic Valves

Priyanka Viswanath^{1,*}, Nathan Rosenbaum², Chris Imholte², Ashutosh Pandey¹ and
Bipin Kashid²

¹*Simerics Inc.*

²*Parker-Hannifin*

**corresponding author: pv@simerics.com*

Abstract.

Hydraulic valves are key components in fluid power systems and are required to operate reliably over wide pressure, flow, and temperature ranges, often under fast-changing load conditions. As hydraulic systems become more compact, complex interactions between valves and the surrounding sub-system can lead to undesirable phenomena such as noise and dynamic instabilities. These effects can degrade system performance, damage components, and negatively impact operator experience. Identifying the root cause of such behavior is challenging and frequently relies on repeated prototyping and extended testing, increasing development time and delaying product launch. This paper presents a decoupled co-simulation methodology for modeling transient flow behavior and instabilities in hydraulic valves. The approach combines one-dimensional (1D) sub-system modeling with three-dimensional (3D) Computational Fluid Dynamics (CFD) analysis using Simerics-MP+. The 1D models establish the operating envelope and are validated against test data to define the boundary conditions for high-fidelity 3D CFD simulations. Targeted CFD analysis enables the identification of operational deviations, cavitation, jets, flow separation and ensuing vortices, which are critical in identifying local instabilities. These detailed insights are difficult to obtain through 1D analysis or to capture experimentally. By maintaining a decoupled framework, the methodology ensures computational efficiency while preserving realistic system behavior. The results demonstrate reduced prototyping cycles and accelerated valve development, providing a practical and scalable approach for resolving complex dynamic issues in modern hydraulic valve designs.

Keywords. Hydraulic valves, CFD, 1D system modeling, valve dynamics, transient flow behavior, instabilities, cavitation.

1. INTRODUCTION

Fluid power systems rely on hydraulic valves to precisely regulate pressure and flow under demanding and rapidly varying operating conditions. As system architectures become more compact and performance requirements more stringent, valve dynamics become increasingly sensitive to interactions with the surrounding circuit. These interactions can manifest as oscillations, cavitation, and flow-induced instabilities that compromise reliability and operator comfort. In hydraulic components, cavitation in particular can lead to vibrations, pressure pulsations, noise, and surface erosion [1], while unsteady flow through valve passages generates pressure fluctuations that propagate as structural excitation throughout the system [2]. Computational Fluid Dynamics (CFD) analysis has been applied to characterize internal flow behavior in various hydraulic valve types, including pressure control valves [3], directional control valves constructed from logic elements [4], flow force reduction in hydraulic valves [5], and noise mitigation in relief valves using hybrid CFD-acoustic approaches [6]. Despite these advances, diagnosing transient instabilities and failure modes under realistic system boundary conditions remains challenging, and development programs frequently address them through iterative build-and-test cycles that are both time-consuming and costly.

Simulation has become an indispensable tool in hydraulic valve development. Lumped parameter models in which valve flow passages are represented as orifices, volumes, and springs are widely used for system-level analysis owing to their low computational cost [1]. However, the simplifying assumptions inherent to this approach limit its ability to resolve spatially distributed phenomena such as localized pressure gradients, jet formation, flow separation, cavitation and instabilities. Gasche et al. [7] demonstrated this directly in the context of compressor suction valves, showing that while a one-dimensional (1D) model predicts global force and displacement well, it cannot capture local flow detail essential for design refinement. Three-dimensional (3D) CFD analysis resolves these effects with high fidelity and can reveal internal flow patterns that are difficult or impossible to obtain experimentally [2]. Dynamic valve simulations using moving mesh approaches and coupled valve motion models have been demonstrated using the commercial CFD code Simerics-MP+ in related applications [8, 9], establishing the solver's capability for transient valve-fluid interaction problems. However, applying three-dimensional CFD indiscriminately across a full operating envelope is computationally prohibitive, and coupled 1D–3D frameworks have been proposed across a range of hydraulic system applications to address this [10, 11]. Most recently, Schmid et al. [12] demonstrated a directly analogous methodology for pressure relief valves in a hydropower system, where 1D model calibration revealed irregular discharge characteristics that could not be explained at the system level, and targeted 3D CFD analysis identified cavitation and geometric flow restrictions as the governing mechanisms; findings that were subsequently used to refine the 1D model. The application of such decoupled multi-scale approaches to industrial hydraulic valves under transient operating conditions, including failure mode diagnosis and design iterations, remains an active area of development.

This paper presents such a methodology, applied to two poppet-style hydraulic valves developed by Parker-Hannifin: a Poppet-style Logic Element (PLE) and an Electro-Proportional Flow (EPF) control valve. System-level 1D models first characterize normal operation and establish validated boundary conditions. 3D CFD in Simerics-MP+ then

targets the cases where system-level analysis falls short. Both valves were identified as exhibiting dynamic behavior that could not be adequately explained or reproduced through 1D system modeling alone. For the PLE, high-frequency poppet oscillations observed experimentally at low flow rates under elevated spring loads are investigated. For the EPF, a pressure-dependent failure mode is diagnosed through analysis of the internal pressure zones, leading to a targeted design modification that is validated in simulation and subsequently confirmed through physical testing. Together, the two case studies demonstrate how the decoupled 1D–3D framework can be used to simulate realistic valve behavior, thereby accelerating design iterations, and reducing prototyping cycles in the development of modern hydraulic valves.

2. MODELS

Two hydraulic valves developed by Parker-Hannifin are considered in this study: a Poppet-style Logic Element (PLE) and an Electro-Proportional Flow (EPF) control valve. The PLE is a 2-way pilot-operated cartridge valve designed for directional flow control of hydraulic systems at flows and pressures that exceed the limits of conventional cartridge valves. A cross-sectional view is shown in Figure 1a, where port 3 is the pilot port. When piloted by low-flow control elements, the PLE act as a high-capacity switch, allowing a wide variety of flow path changes analogous to multi-way spool valves, while preventing inter-port leakage that a spool would otherwise allow. The PLE has a 2:1 pilot ratio such that the area on which the pressure acts in the pilot chamber is twice that of the port 1 seat area and the port 2 differential area between the seal and the seat. Since the areas at ports 1 and 2 are equal, the valve is biased closed by a spring in the pilot chamber and opens only when the pressure at either port 1 or port 2 exceeds the load of the bias spring.

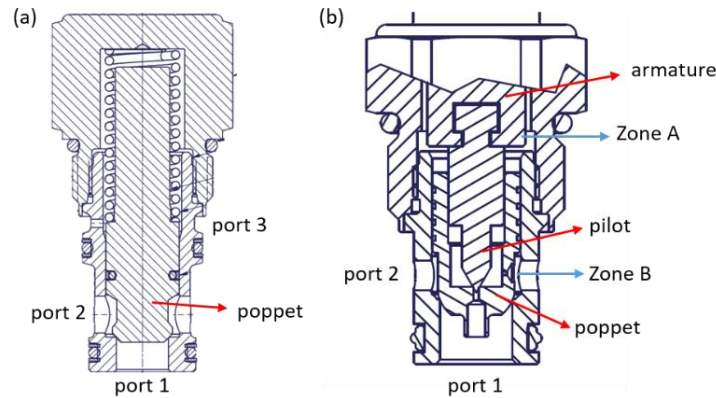


Figure 1: Cross-section views of the valve models. (a) Poppet-style Logic Element (PLE), with port 3 as the pilot port. (b) Electro-Proportional Flow (EPF) control valve, showing the pilot, poppet, and key zones.

The EPF is a poppet-style normally closed electro-proportional flow control valve used in load-holding applications to control flow into or out of a cylinder, while blocking leakage that would allow loads to drift when the valve is not engaged. This dual function simplifies hydraulic circuits by combining the functions of a spool valve for directing flow and a check valve for preventing leakage. A cross-sectional view is shown in Figure 1b. In the de-

energized state, the internal springs hold the pilot against the poppet and the poppet against an internal seat, sealing the flow path from port 2 to port 1. Pressure builds behind the poppet (Zone A), adding to this closing force. When the solenoid is energized, the pilot lifts from the poppet allowing the pressure in Zone A to vent to port 1. The resulting pressure at port 2, acting on the differential area between the poppet outer diameter and seat diameter (Zone B), overcomes the reduced closing force and lifts the poppet off the seat, allowing controlled flow from port 2 to port 1. When the solenoid is de-energized, the pilot reseats against the poppet, and the combined action of the return spring and the recovering pressure in Zone A pushes the poppet back against the seat.

3. MODELING APPROACH

A co-simulation methodology is employed that combines 1D system modeling with 3D CFD analysis. The 1D models are developed first to capture system-level behavior and run across the full range of operating conditions, with certain assumptions refined iteratively. For edge cases and worst-case operating scenarios, 3D CFD provides higher-fidelity insight into local flow behavior, instabilities, extent of cavitation, jet formation, and operational deviations that are difficult to resolve with 1D models alone. The boundary conditions and parameters used to model the valve motion in the 3D CFD models are derived directly from the 1D results, ensuring that the CFD simulations reflect realistic system behavior while maintaining computational efficiency.

3.1. 1D

1D simulations are developed in Amesim, with some general assumptions made, including but not limited to:

- Physical parts do not include non-conformances.
- Certain flow passages are sufficiently non-restrictive as to act as one volume.
- Certain volumes are sufficiently restrictive as to be treated as an orifice rather than a volume.

Parameters relating to fluid inertia were left at the default values provided by Amesim. For the PLE, the seal friction, being a specialty seal, could be incorporated into the friction of the poppet mass, rather than as an independent pressure dependent model element. Key factors in achieving close agreement with real world valve performance include the selection of which flow passages are treated as volumes versus restrictive orifices, the inclusion of the cavity surrounding the cartridge in the manifold block in which the cartridge is installed, and the application of varying types of friction.

The EPF model includes two additional assumptions:

- The solenoid function operates normally, and as such, its behavior could be assumed as a direct force input, rather than considering how its components influence the conversion of electrical power to the physical force acting on the moving components of the valve.
- The nature of the link between the pilot and solenoid armature is such that the two components could be considered as one mass.

3.2. 3D CFD

3.2.1. Governing Equations

3D CFD simulations are performed in Simerics-MP+. The integral form of the unsteady conservation of mass and momentum equations is solved [13].

Cavitation is modeled using Constant Gas Mass Fraction model available in Simerics-MP+, which accounts for both liquid-vapor phase change and the presence of non-condensable gases. In this model, the mass fraction of non-condensable gases is held constant and is treated as an ideal gas [13, 14].

Valve motion is modeled using a translational one-degree-of-freedom (1-DOF) formulation, applied either as a prescribed position or a force balance. In the prescribed position approach, valve displacement is specified as a user-defined function of time. In the force balance approach, the valve position is determined by solving a scalar linear momentum equation:

$$m \frac{d^2 s}{dt^2} = F_{hydrodynamic} - F_{damping} - F_{spring} \quad (1)$$

where m is the mass of the solid body and s is the magnitude of the solid body's position vector. The hydrodynamic force comprises pressure and shear contributions from the relative motion between the fluid flow and the surfaces of the solid body which are in contact with the flow:

$$F_{hydrodynamic} = \sum F_{pressure} + \sum F_{shear} \quad (2)$$

The damping force is a retarding force proportional to the velocity determined by the motion of the solid body and a user-specified damping coefficient D :

$$F_{damping} = D \frac{ds}{dt} \quad (3)$$

The spring force depends on the spring preload $F_{preload}$, spring constant k , and displacement of the spring, d :

$$F_{spring} = F_{preload} + kd \quad (4)$$

3.2.2. Simulation Setup

Hydraulic oil properties are used for all simulations. The setup differs between the two valves, reflecting differences in their operating modes and internal complexity. All results and boundary conditions are presented in normalized form to protect proprietary design information while preserving the underlying flow physics.

For the PLE, two operating conditions are considered: flow from port 1 to port 2 (OP1) and flow from port 2 to port 1 (OP2). A volumetric flux inflow boundary condition is applied at the inlet, ramped from 0 to 1 unit and back to 0 unit. The outlet has a specified pressure boundary condition at atmospheric pressure. Although physical test ramps span 30–60 seconds, CFD ramps are performed over 1 second with a time step of approximately 0.15 millisecond. Sensitivity studies confirmed that the shorter ramp duration produces equivalent results, significantly reducing computational cost. The time step size used

ensures a minimum of 100 time steps per cycle, where the cycle frequency corresponds to the natural frequency of the valve body. This resolution aligns with established best practices for accurately capturing valve dynamics. Poppet motion is modeled using a single 1-DOF force balance model, with poppet mass, spring constant, preload, and damping coefficient specified as user inputs. A closure model is applied to handle complete valve seating, without having to compress the corresponding mesh volumes to zero. The flow in the cells between the valve body and valve seat is cut off when gap between them is below a certain specified value. In the PLE, this gap is set to 25 μm . The pilot port and chamber are excluded from the PLE model, as the case simulated represents a worst-case scenario in which no pilot pressure signal is applied. In this condition, the large opening at port 3 allows oil to escape freely as the poppet lifts, rendering pilot chamber damping negligible.

Flow in the EPF is from port 2 to port 1, with pressure boundary conditions specified both at the inlet and the outlet. Three inlet pressures are simulated namely P_{low} , $P_{intermediate}$ and P_{high} , with the outlet set to atmospheric pressure. Three translation 1-DOF models are used for the three bodies that can move independently. The pilot and armature are modeled with a prescribed displacement ramped linearly over time. They are treated as a single combined mass consistent with the 1D model. The poppet motion is governed by a force-balance, with its position bounded by the pilot position. Closure models are applied to both the poppet and pilot, with a gap limit of 100 μm . A time step of approximately 0.05 millisecond is used.

4. MESH

Simerics-MP+ uses a binary-tree meshing algorithm to create a mesh comprising primarily hexahedral elements, with tetrahedral elements used where hexahedral elements cannot adequately resolve local geometry, primarily on the surface of the volumes. Additionally, a deforming mesh strategy is used to model the motion of the valves, which preserves mesh topology throughout the simulation, eliminating the need for remeshing. This ensures solution continuity without facing interpolation errors and removing the computational overhead associated with dynamic mesh refinement. The stationary and deforming fluid volumes are meshed separately. The deforming volumes are meshed using the Valve Template offered in Simerics-MP+, which automatically assigns boundary conditions and optimizes the mesh to accommodate both deformation and valve closure.

Cross-sectional views of the mesh for each valve are shown in Figure 2. The PLE mesh comprises approximately 450,000 cells (Figure 2a), and the EPF mesh approximately 922,000 cells (Figure 2c). A mesh independence study was conducted using a finer grid resolution, which exhibited negligible differences in results. Hence, these baseline mesh densities were used for computational efficiency. The mesh can be compressed to a gap of a few microns to represent the physically closed valve position, as shown for the PLE in Figure 2b and the EPF in Figure 2d. In the EPF, the leakage gap between the poppet and the cage is meshed using the Annulus Template, which generates a high quality orthogonal mesh in gaps on the order of microns. The motion of the valve is dictated by the translation 1-DOF model calculations. Flow communication between separate meshed volumes is handled using Mismatched Grid Interfaces (MGIs) in Simerics-MP+, which allow flux to pass through a shared interface between volumes regardless of node alignment. The interface can be created between stationary or moving boundaries. For moving boundaries, it is automatically recalculated at each time step.

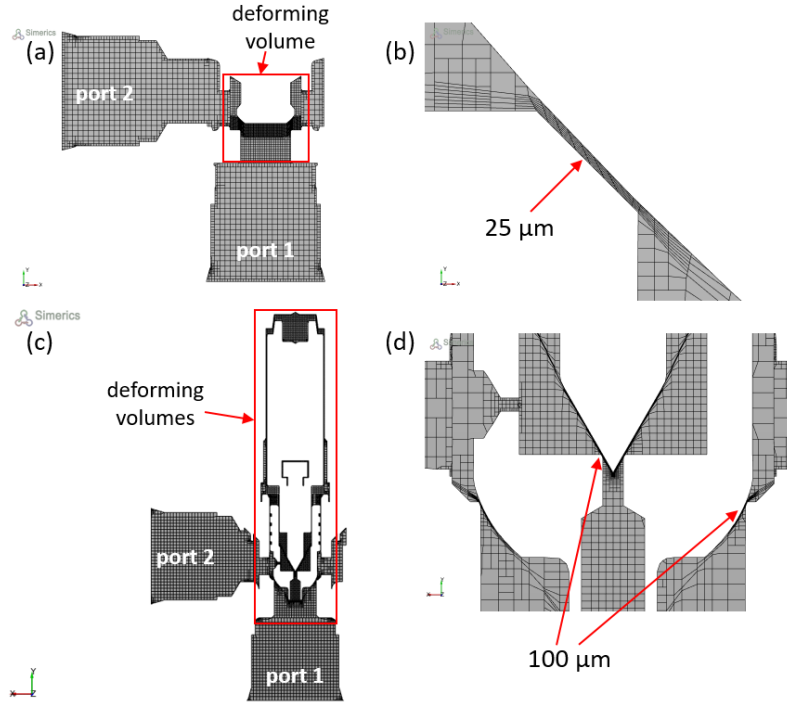


Figure 2. Mesh resolution in the PLE and EPF. (a) Cross-sectional view of the PLE mesh. (b) Close-up of the PLE mesh where the poppet physically touches the valve seat, showing a mesh compressed into a 25 μm gap. (c) Cross-sectional view of the EPF mesh. (d) Close-up of the mesh where the poppet physically touches the valve seat and the pilot touches the poppet, showing a mesh compressed into in a 100 μm gap.

The EPF presents an additional meshing consideration due to the three independently moving valve bodies. The pilot and armature travel together under electromagnetic actuation while the poppet moves separately due to hydraulic forces. As the travel distances and dynamics of these bodies are independent, their mesh deformations are handled separately, which is handled by the meshing capabilities of Simerics-MP+.

5. RESULTS AND DISCUSSION

5.1. PLE

In testing, the PLE exhibits unstable behavior at higher bias spring loads. With no pilot pressure applied, as the flow increases from either port 1 or 2, the poppet oscillates at high frequency until the flow rate is sufficient to push the poppet fully against its stop. A similar instability recurs as flow decreases and the poppet returns toward the seat. 3D simulation was employed to gain greater insight into how fluid behavior could influence the lift forces acting on the poppet, with the objective of identifying potential poppet designs that maximize lift force to overcome the opposing hydrodynamic forces resisting valve opening. The following results are presented in normalized form as:

$$\theta^* = \frac{\theta}{\theta_{ref}} \quad (5)$$

Where, θ is any physical quantity and θ_{ref} is a reference or maximum value of that quantity.

The simulation results are validated against test data. Figure 3a shows pressure drop versus flow rate for operating condition OP1, with the volumetric flux ramped from 0-1 unit and back to 0 unit. Figure 3b shows the corresponding data for operating condition OP2. The region of instability observed in tests at low flow rates is captured in the 3D CFD results for both OP1 and OP2, though the predicted oscillation amplitudes are higher than measured. The amplitude can be tuned by adjusting the damping coefficient, however, the objective of this study was to capture the extent of the instability region rather than its amplitude, and the current inputs are appropriate for that purpose. Once the instabilities subside, both the test and CFD results show a similar pressure drop between the 0-1 and 1-0 unit ramps. Figure 3c shows the 1D simulation results, in which both stiction and coulomb friction are included. The 1D model also captures the instability region, though the oscillations decay at a lower flow rate.

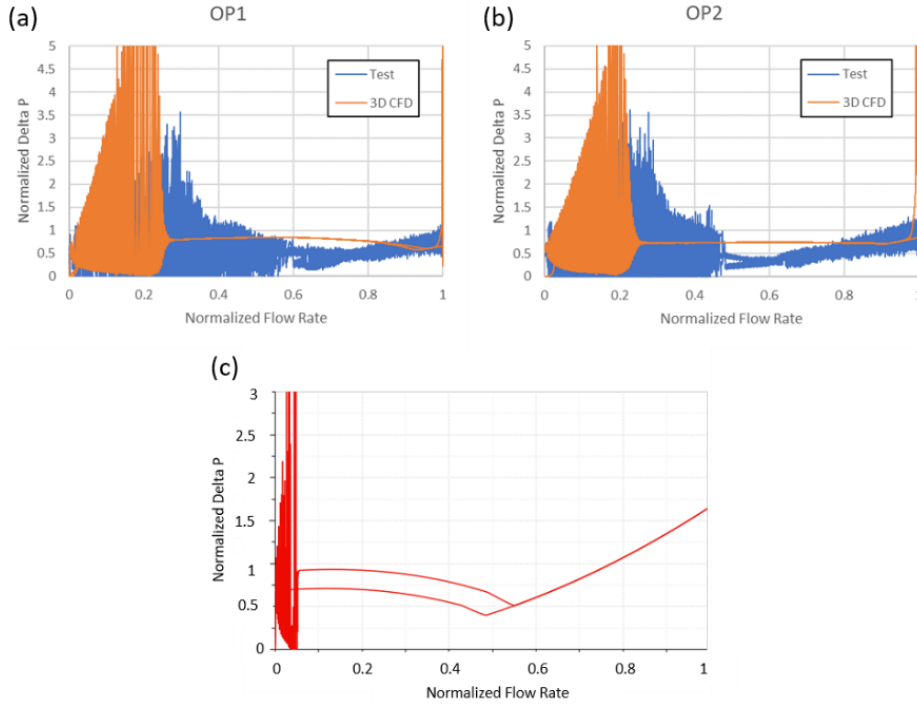


Figure 3: Validation against test data. (a) Test and 3D CFD data showing the pressure drop over flow rate for operating condition OP1. (b) Test and 3D CFD data showing the pressure drop over flow rate for operating condition OP2. (c) 1D simulation showing the instabilities in pressure drop in the low flow rate region.

Figures 4a and 4b show valve lift versus flow rate for OP1 and OP2 respectively, with each plot showing both the 0-1 as well as 1-0 unit ramps. The oscillations in displacement are lower during the decreasing ramp from 1-0 unit, in both OP1 and OP2. Once the oscillations die down, the lift profiles converge closely until the poppet begins to bottom out, where minor differences are observed. The poppet reaches its travel stop at flow rates above approximately 0.9 unit in both conditions. While the overall behavior in OP1 and OP2 is similar, displacement oscillations during the 0-1 unit ramp are marginally higher in OP1.

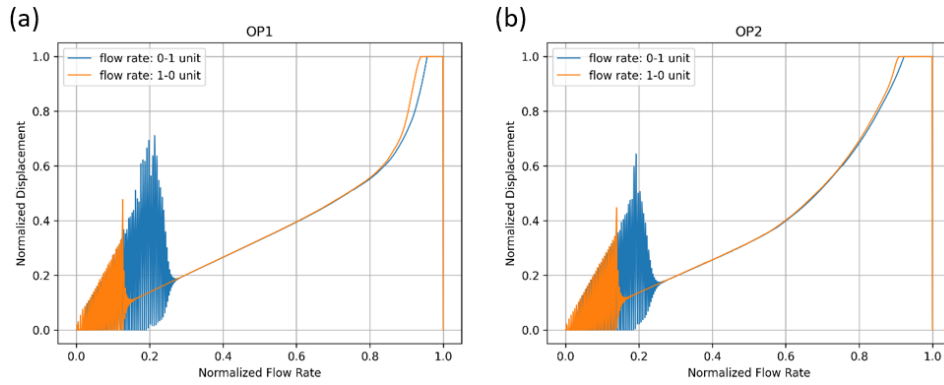


Figure 4: Valve lift versus flow rate. 3D CFD data showing the poppet lift versus flow rate with both the ramp up from 0-1 unit as well as the ramp down from 1-0 unit (a) for OP1, (b) for OP2.

Figures 5a and 5b show the hydrodynamic force acting on the poppet as a function of flow rate, for OP1 and OP2 respectively. The variation in fluid force is larger during the increasing ramp from 0-1 unit in both OP1 and OP2, with peak values of approximately 1 unit, nearly double those observed during the decreasing ramp from 1-0 unit. Once the oscillations subside, the hydrodynamic force is consistent in both the increasing and decreasing ramps. The valve instability is triggered at lower flow rates, which corresponds to low valve lift positions where pressure drop is highest. During the flow rate ramp up from 0-1 unit, the accelerating fluid could increase the kinetic energy imparted to the system, providing negative damping that could induce high amplitude oscillations that also take longer to dissipate. Conversely, when the flow rate is ramped down from 1-0 unit, the decelerating fluid could provide positive hydrodynamic damping, which suppresses oscillation amplitudes and delays the onset of instability. This delay could further be aided by the inertia of the fluid and the momentum of the moving poppet. Differences between OP1 and OP2 are pronounced in the oscillatory region, with slightly higher fluid forces in OP1 during the ramp up from 0-1 unit.

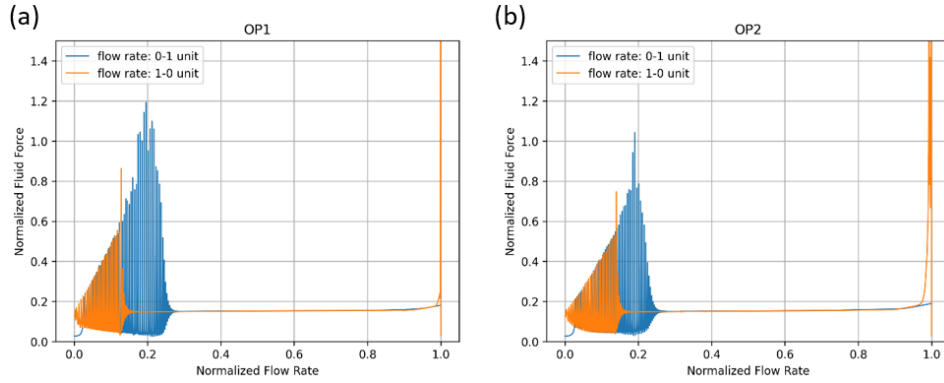


Figure 5: Hydrodynamic force versus flow rate. 3D CFD data showing the hydrodynamic force acting on the valve versus flow rate with both the ramp up from 0-1 unit as well as the ramp down from 1-0 unit (a) for OP1, (b) for OP2.

Examination of the pressure contours shows that the inlet port becomes pressurized up to a threshold level, above which poppet oscillations subside and stable lift begins. Figures 6a and 6c show this condition for operating conditions OP1 and OP2 respectively. As flow rate continues to increase toward 1 unit, the inlet port progressively depressurizes, as seen in Figures 6b and 6d at maximum flow. In condition OP2, where the poppet reaches its stop at a slightly lower flow rate than in OP1, pressure builds again at the inlet port once the poppet is fully bottomed-out, which is apparent in Figure 6d when compared to Figure 6b at the same flow rate.

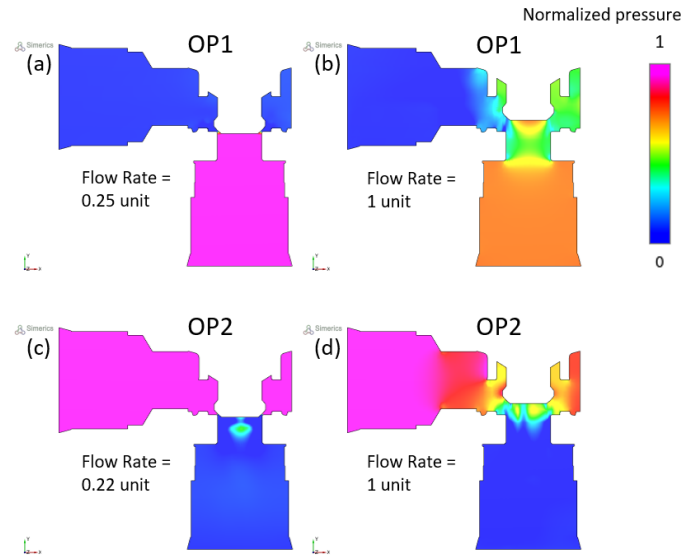


Figure 6: Cross-section showing the pressure contours in the PLE. (a) OP1 at the onset of stable lift. (b) OP1 when the poppet has bottomed-out and the flow rate reaches 1 unit. (c) OP2 at the onset of stable lift. (d) OP2 when the valve has bottomed-out and the flow rate reaches 1 unit.

Velocity contours at the same flow rate conditions are shown in Figure 7. Figure 7a shows operating condition OP1, where flow enters the narrow gap between the poppet and valve seat at a high velocity as the poppet begins stable lift, eventually forming a jet reaching 0.4–0.6 unit at the maximum flow rate of 1 unit, as seen in Figure 7b. A stagnation point is visible on the poppet face at port 1 in Figure 7b. Operating condition OP2 is depicted in Figure 7c, where the jet forms directed toward the outlet port, reaching a peak core velocity of approximately 1 unit at a flow rate of 1 unit. A stagnation point at a similar location on the poppet face is observed in OP2, as the jet exits through port 1.

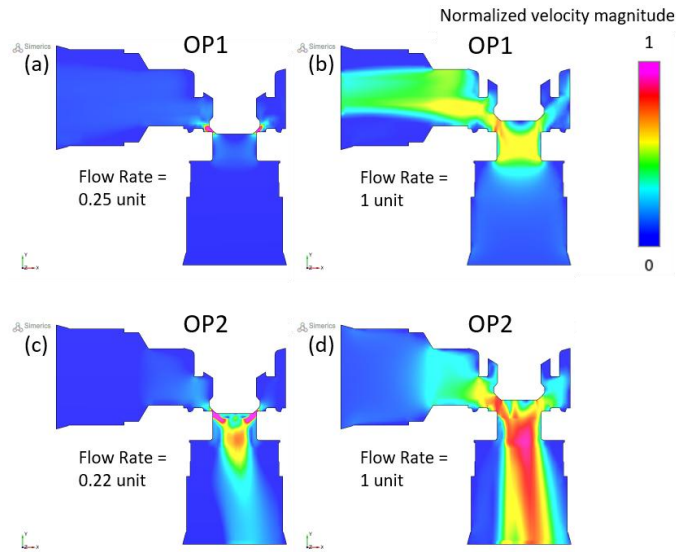


Figure 7: Cross-section showing the velocity contours in the PLE. (a) OP1 at the onset of stable lift. (b) OP1 when the poppet has bottomed-out and the flow rate reaches 1 unit. (c) OP2 at the onset of stable lift. (d) OP2 when the valve has bottomed-out and the flow rate reaches 1 unit.

Cavitation is quantified by the average volume fraction of the total gas, which accounts for vapor, dissolved gas, and free non-condensable gas phases combined. Figure 8 shows isosurfaces of regions where the total gas volume fraction exceeds 0.1, colored by volume fraction magnitude. Figure 8a shows cavitation bubbles in operating condition OP1 which first appears around the poppet–seat interface as the valve begins to open. The largest cavitation region occurs when the maximum flow rate of 1 unit is reached, as seen in Figure 8b, extending into the outlet port 2. The highest volume fraction of gas is concentrated near the entrance to port 2. In OP2, the initial cavitation bubble forms in a similar location, as seen in Figure 8c, and also reaches its maximum extent at 1 unit where it extends into the outlet port 1, as shown in Figure 8d. The cavitation bubble in OP2 extends into port 1 at maximum flow, with the highest gas fraction concentrated near the poppet face. The differences in cavitation extent and distribution between OP1 and OP2 reflect the distinct flow paths and jet orientations characteristic of each operating condition.

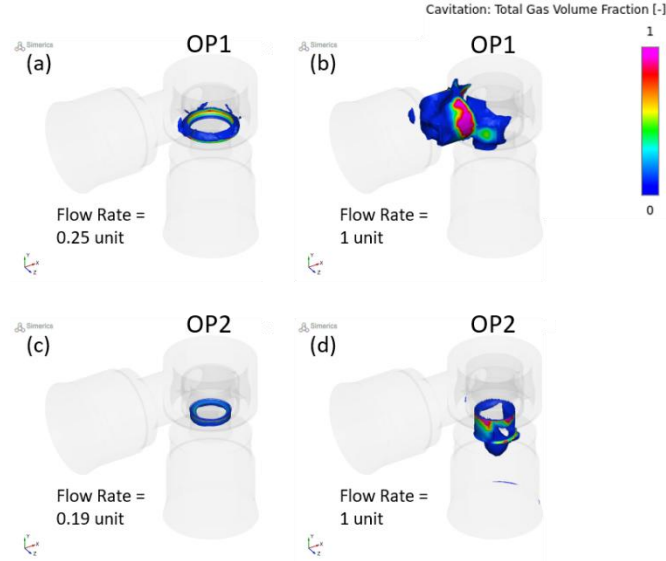


Figure 8: Regions where total gas volume fraction > 0.1 in the PLE. (a) OP1 at the onset of stable lift. (b) OP1 when the poppet has bottomed-out and the flow rate reaches 1 unit. (c) OP2 at the onset of stable lift. (d) OP2 when the valve has bottomed-out and the flow rate reaches 1 unit.

This methodology is being applied to evaluate multiple design variations of the poppet, with the objective of reducing instability at low flow rates. Key metrics include the hydrodynamic force acting on the poppet and the evolution of pressure and velocity fields as the poppet lifts and seats across a range of flow rates. Together, these diagnostics enable rapid screening of poppet design variants, identifying promising designs and eliminating underperforming ones based on computed lift force, pressure, and velocity fields, before any physical prototype is committed to manufacture or test.

5.2. EPF

In early testing, the valve did not perform as expected, failing to open above certain inlet pressures. Test data indicated that the solenoid was successfully lifting the pilot, but the poppet was not lifting off the seat. Several 1D models were developed that reasonably represented the expected behavior of the valve, but this particular failure mode could not be replicated. A 3D CFD model was developed to resolve fluid effects not captured by the 1D analysis. The CFD model reproduced the pressure dependency of the performance, revealing that above a certain inlet pressure, the pressure in Zone A does not drop sufficiently for the poppet to lift. A specific feature was found to control this behavior, and was proposed as a design change. The original design is referred to as Design 1 and the new proposed design as Design 2 in the results that follow. Similar to the PLE, the results are presented in a normalized form.

Figure 9 shows the displacement of the armature and the poppet at different inlet pressures for both Design 1 and Design 2. The armature displacement is identical across both designs. In Design 1, the poppet lifts at P_{low} but fails to lift at $P_{intermediate}$. In Design 2, the poppet lifts successfully at $P_{intermediate}$, closely tracking the armature and pilot displacement. A

lag between armature lift and poppet response is also visible in Design 1 at P_{low} , which is negligible in Design 2.

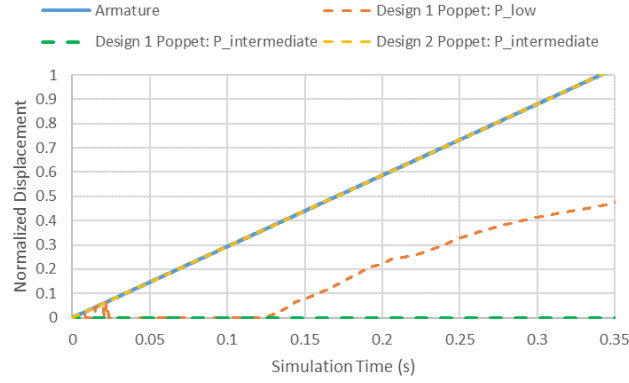


Figure 9: Valve Lift. Simulation data showing the valve lift for the armature and the poppet at different inlet pressures in both Design 1 and Design 2.

Figure 10 shows pressure contours in the EPF at key operating conditions. Figure 10a shows the pressure contours in Design 1 at an inlet pressure of P_{low} . The pressure drop in Zone A in this case is sufficient to allow the poppet to open. At $P_{intermediate}$, however, the poppet remains seated even as the pilot lifts, which can be observed in Figure 10b, indicating that the pressure in Zone A does not drop low enough to overcome the closing force. Figure 10c shows Design 2 at $P_{intermediate}$, where the pressure in Zone A is substantially lower than in Design 1 at the same condition, enabling the poppet to lift. The proposed design modification successfully resolved the operational deviation, and simulations confirmed that the poppet in Design 2 lifts reliably at inlet pressures up to P_{high} .

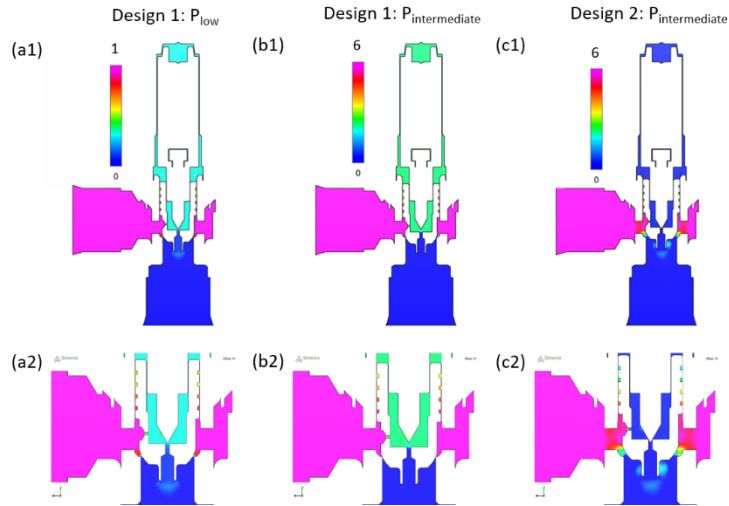


Figure 10: Cross-section showing the pressure contours in the EPF. (a1) Design 1 at inlet pressure P_{low} , where the poppet lifts successfully. (a2) Detailed view of the poppet and pilot lift region for Design 1 at P_{low} . (b1) Design 1 at inlet pressure $P_{intermediate}$, where the poppet fails to lift. (b2) Detailed view of the poppet and pilot lift region for Design 1 at $P_{intermediate}$. (c1) Design 2 at inlet

pressure $P_{intermediate}$, where the poppet lifts successfully. (c2) Detailed view of the poppet and pilot lift region for Design 2 at $P_{intermediate}$

Figure 11 shows pressure histories at three probe locations for both designs at an inlet pressure of $P_{intermediate}$. As the pilot and armature begin to lift, the pressure drop across these regions is approximately 0.4 unit greater in Design 2 than in Design 1, providing the additional pressure differential needed for the poppet to overcome the opposing spring forces and lift. The pressure drop in Design 1 is also more gradual, further reducing the available opening force.

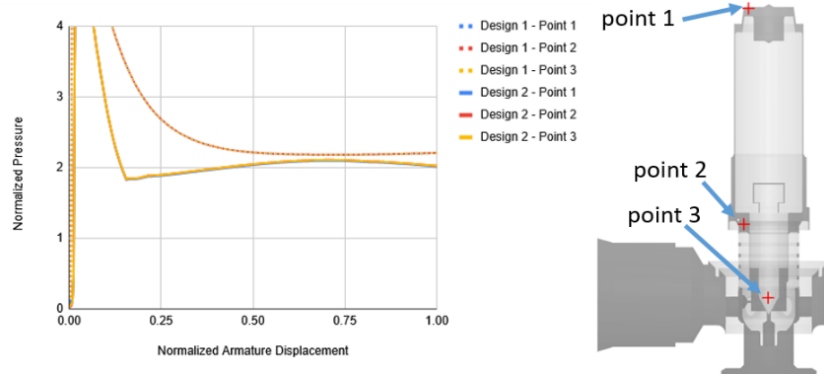


Figure 11: Pressure at probe locations. Pressure is monitored at three probe points placed in key regions in the EPF over increasing armature displacement, showing a higher pressure in Design 1 compared to Design 2.

Figure 12 shows the velocity contours in the EPF. Figure 12a shows the formation of a jet in Design 1 as the poppet lifts at an inlet pressure of P_{low} , with a core velocity of approximately 0.43 unit. At an inlet pressure of $P_{intermediate}$, though the poppet does not lift, there is some flow from port 2 to port 1 through orifices in the poppet as seen in Figure 12b. The design change in Design 2 allows the poppet to lift at $P_{intermediate}$, with Figure 12c showing the formation of a jet with a core velocity of about 2.14 unit as the poppet lifts, consistent with the significantly higher operating pressure compared to the P_{low} condition.

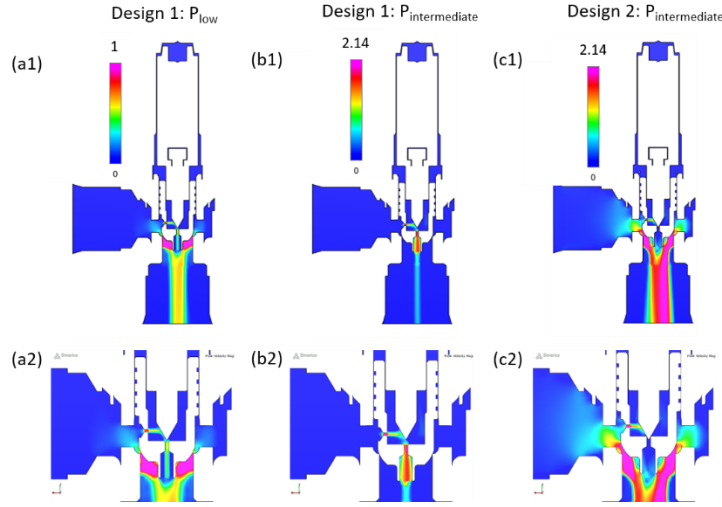


Figure 12: Cross-section showing the velocity contours in the EPF. (a1) Design 1 at inlet pressure P_{low} , where the poppet lifts successfully. (a2) Detailed view of the poppet and pilot lift region for Design 1 at P_{low} . (b1) Design 1 at inlet pressure $P_{intermediate}$, where the poppet fails to lift. (b2) Detailed view of the poppet and pilot lift region for Design 1 at $P_{intermediate}$. (c1) Design 2 at inlet pressure $P_{intermediate}$, where the poppet lifts successfully. (c2) Detailed view of the poppet and pilot lift region for Design 2 at $P_{intermediate}$

Figure 13 shows regions where the total gas volume fraction exceeds 0.1, capturing the combined contribution of vapor, dissolved gas, and free non-condensable gas. Cavitation bubbles form primarily as the poppet begins to lift and extend into outlet port 1. Figure 13a shows Design 1 at an inlet pressure of P_{low} and in Figure 13b, the inlet pressure is at $P_{intermediate}$ for the same design. As the poppet does not lift at this pressure, cavitation is confined to the poppet orifices through which flow leaks. Figure 13c shows Design 2 at $P_{intermediate}$, where the larger gas volume fraction compared to the P_{low} case is consistent with the higher operating pressure and jet velocity.

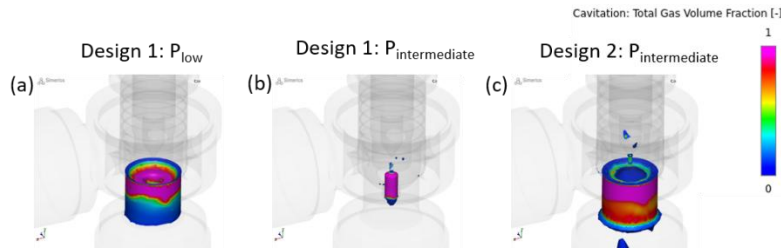


Figure 13: Regions where total gas volume fraction > 0.1 in the EPF. (a) Design 1 at inlet pressure P_{low} , where the poppet lifts successfully. (b) Design 1 at inlet pressure $P_{intermediate}$, where the poppet fails to lift. (c) Design 2 at inlet pressure $P_{intermediate}$, where the poppet lifts successfully.

3D CFD results are validated with test data. The test data in Design 1 in Figure 14a shows a clear increase in flow rate with solenoid current at P_{low} , while the $P_{intermediate}$ case produces less than 0.1 unit flow rate, signaling flow through the poppet orifices only. CFD results, plotted against armature displacement on the same axes, follow the same trends for both pressure conditions, confirming that the simulation correctly captures the poppet's

failure to lift at elevated pressure. Figure 14b shows the corresponding data for Design 2 at P_{high} . Both test and simulation demonstrate that the design modification enables full poppet lift at high operating pressures.

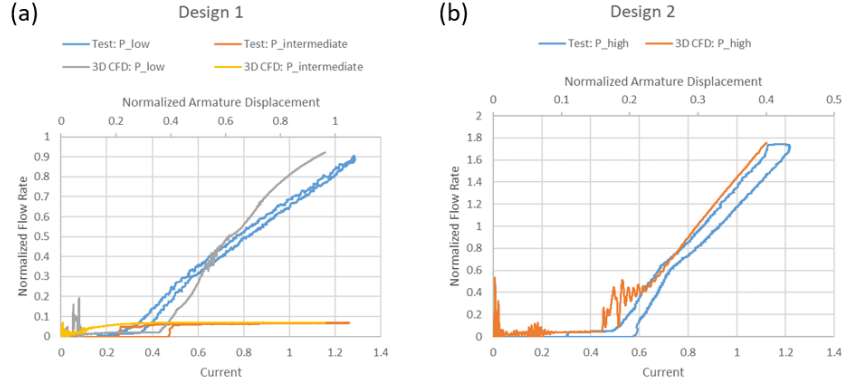


Figure 14: Validation against test data. (a) Test data showing flow rate versus current and CFD data showing flow rate versus armature displacement for Design 1. Poppet fails to lift at an inlet pressure of $P_{intermediate}$, which was also observed in the CFD simulation. (b) Test data showing flow rate versus current and CFD data showing flow rate versus armature displacement for Design 2. Both test and simulation show that the poppet in the new design, Design 2, can lift even at a high inlet pressure of P_{high} .

The validated CFD results provided the confidence to manufacture a prototype, which was subsequently tested successfully. Informed by the CFD pressure analysis across the separate zones, the 1D model was revisited to assign greater significance to the less predictable frictional and viscosity related parameters that had previously been under-constrained. With these adjustments, the 1D model was able to replicate the observed failure behavior successfully across a range of inlet pressures from P_{low} up to P_{high} .

6. SUMMARY

This study presents a decoupled co-simulation methodology for analyzing transient flow behavior and instabilities in hydraulic valves, combining one-dimensional (1D) system modeling with high-fidelity three-dimensional (3D) Computational Fluid Dynamics (CFD) simulations in Simerics-MP+. The methodology was demonstrated on two valves developed by Parker-Hannifin: a poppet-style Logic Element (PLE) and an Electro-Proportional Flow (EPF) control valve. In the decoupled framework, 1D models characterize normal operating conditions and establish model parameters iteratively, while 3D CFD is used to investigate worst-case scenarios and failure modes that cannot be resolved at the system level. For the PLE, CFD reproduced experimentally observed poppet oscillations at low flow rates and enabled rapid computational screening of design variants prior to physical prototyping. For the EPF, CFD identified a pressure-dependent failure mode that 1D modeling could not replicate, leading to a targeted design modification that was validated in simulation and confirmed through physical testing across a range of inlet pressures. Together, these results demonstrate that the decoupled 1D–3D methodology reduces prototyping cycles and accelerates valve development by providing physical insight that is difficult to obtain

through 1D analysis or experimental measurements alone. The approach is computationally efficient, preserves realistic system behavior, and is directly applicable to the resolution of complex dynamic issues in modern hydraulic valve designs.

7. REFERENCES

- [1] Valdés, J. R., Rodríguez, J. M., Monge, R., Peña, J. C., & Pütz, T. (2014). Numerical simulation and experimental validation of the cavitating flow through a ball check valve. *Energy Conversion and Management*, 78, 776–786.
- [2] Yang, Q., Zhang, Z., Liu, M., & Hu, J. (2011). Numerical simulation of fluid flow inside the valve. *Procedia Engineering*, 23, 543–550.
- [3] Wu, D., Li, S., & Wu, P. (2015). CFD simulation of flow-pressure characteristics of a pressure control valve for automotive fuel supply system. *Energy Conversion and Management*, 101, 658–665.
- [4] Lisowski, E., & Rajda, J. (2013). CFD analysis of pressure loss during flow by hydraulic directional control valve constructed from logic valves. *Energy Conversion and Management*, 65, 285–291.
- [5] Heraković, N., Duhovnik, J., & Šimic, M. (2015). CFD simulation of flow force reduction in hydraulic valves. *Tehnicki Vjesnik - Technical Gazette*, 22(2), 453–463.
- [6] Taghizadeh, S., Oravec, J., & Dhar, S. (2025). Noise Mitigation Analysis in Relief Valves Using a Hybrid CFD-Acoustics Approach. *Proceedings of the ASME 2025 International Mechanical Engineering Congress and Exposition*.
- [7] Gasche, J. L., Demétrio, A., De, S., Dias, L., Domingues Bueno, D., Ferreira Lacerda, J., Gasche, J. L., Dias, A. D. S. L., Bueno, D. D., & Lacerda, J. F. (2016). Numerical Simulation of a Suction Valve: Comparison between a 3D Complete Model and a 1D Model. *International Compressor Engineering Conference*. Paper 2461.
- [8] Dhar, S., Ding, H., Lacerda, J., & Lacerda, J. (2016). A 3-D Transient CFD Model of a Reciprocating Piston Compressor with Dynamic Port Flip Valves. *International Compressor Engineering Conference*. Paper 2471.
- [9] Chaudhari, N., Ding, H., Gao, H., & Mohapatra, C. K. (2024). Dynamic Reed Valve in Rolling Piston Compressor: A 3-Dimensional Transient CFD Simulation 3-Dimensional Transient CFD Simulation. *International Compressor Engineering Conference*.
- [10] Wang, C., Nilsson, H., Yang, J., & Petit, O. (2017). 1D–3D coupling for hydraulic system transient simulations. *Computer Physics Communications*, 210, 1–9.
- [11] Zhang, P., Cheng, Y., Xue, S., Hu, Z., Tang, M., & Chen, X. (2023). 1D-3D coupled simulation method of hydraulic transients in ultra-long hydraulic systems based on OpenFOAM. *Engineering Applications of Computational Fluid Mechanics*, 17(1).
- [12] Schmid, J., Bussard, C., Nicolet, C., Forest, S., Stojanovic-Roth, S., & Saugy, J. N. (2025). Investigation of Pressure Relief Valve Performance from 1D Hydraulic

Transient Model Calibration to Advanced CFD Analysis. *IOP Conference Series: Earth and Environmental Science*, 1561(1), 012033.

- [13] Ding, H., Visser, F. C., Jiang, Y., & Furmanczyk, M. (2011). Demonstration and validation of a 3D CFD simulation tool predicting pump performance and cavitation for industrial applications. *Journal of Fluids Engineering, Transactions of the ASME*, 133(1).
- [14] Singhal, A. K., Athavale, M. M., Li, H., & Jiang, Y. (2002). Mathematical basis and validation of the full cavitation model. *Journal of Fluids Engineering, Transactions of the ASME*, 124(3), 617–624.

Analysis of Efficiency and Axial Compensation in Continuous-Contact Helical Gear Pumps through Multiphysics Modeling and Experimental Validation

Parth Tawarawala^{1,*}, Harsh Kulkarni¹, Dinghao Pan¹, Andrea Vacca¹

¹Maha Fluid Power Research Center, School of Mechanical Engineering, Purdue University, Lafayette, IN, USA

E-mail: ptawaraw@purdue.edu

*Corresponding Author

Abstract.

This work presents a multi-physics, strongly coupled numerical model for the study of continuous-contact helical gear pumps (CCHGP), specifically focusing on the numerical analysis of their axial compensation and the prediction of volumetric and hydromechanical efficiencies. The proposed work involves a multi-domain multi-physics modeling approach that combines lumped-parameter modeling of the fluid domain and body dynamics, with consideration of all degrees of freedom and range of motion of the floating bodies, and a CFD-based numerical model for the lubricating interfaces of the continuous-contact helical gear pump, built using an in-house C++-based framework, Multics. The paper shows that the model predicts the pump efficiency trends and outlet pressure ripple in good agreement with experimental measurements on a reference unit, while considerably improving upon state-of-the-art models, and reports a breakdown of the major loss sources in the pump. The paper also demonstrates model capabilities in predicting gear micro-motions and dynamics. The complete model can be used to perform predictive numerical simulations of existing designs or to optimize them to improve performance.

Keywords. Continuous-contact Helical Gear Pump; Multiphysics modeling; Axial Compensation dynamics; Efficiency prediction; Experimental Validation

1. INTRODUCTION

The Continuous-Contact Helical Gear Pump (CCHGP), shown in Figure 1.1, is a positive-displacement machine that has gained interest in the fluid power community for its low-noise operation, first introduced and studied in [1][2][3]. As noted in [4], specific geometric configurations of CCHGPs achieve a theoretical flow ripple of zero, thereby significantly reducing noise during operation. As hydraulic machines are being developed for new applications, such as electrification, pump noise must be addressed. This is particularly true for the External Gear Pump (EGP). CCHGPs share structural similarities with EGPs while addressing one of their biggest disadvantages; thus, they must be studied in detail for applications in hydraulic actuation and fluid transport.

The CCHGP, like the EGP, consists of a drive and driven gear, journal bearings to compensate for radial loading on gears, and lateral pressure plates for axial sealing and compensation. The transverse gear profile is a special compound profile called the continuum profile. However, there are critical differences as well. During meshing, the tips and roots of continuum gears seal completely, creating a pressure-differential sealing surface

in the axial direction, a feature absent in EGPs. The helical gear design introduces unbalanced axial loads and tilting moments that affect axial gap heights and bushing dynamics. Furthermore, CCHGP units often utilize pressurized pistons to compensate for the gears' axial imbalances. These pistons introduce leakage paths from the high-pressure port to the low-pressure drain, altering bearing dynamics and increasing volumetric losses. These complexities prevent the direct application of EGP design principles to CCHGPs, and dedicated tools are required to support the CCHGP's research and development.

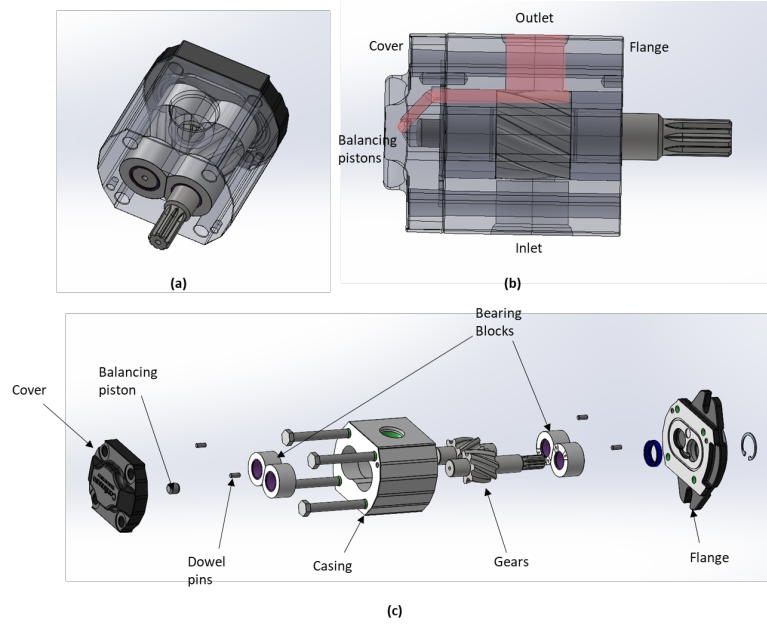


Figure 1.1. Reference CCHGP. (a) 3D CAD of the pump; (b) axial compensating system of the pump; (c) exploded view of the components.

Previous analytical models have correlated gear profiles with performance parameters such as compactness and flow ripple [5][6], but they fail to predict losses due to leakage, compressibility, and cavitation. Lumped Parameter Models [4] can capture the physics of these fluids while enabling very fast evaluation times. However, these models struggle to accurately capture the compensation mechanisms and lubricating interfaces. Similarly, 3D CFD [7] methods often simplify body dynamics and lubrication interfaces, limiting their utility for predicting failures of compensation mechanisms and associated losses.

State-of-the-art numerical multi-physics models, such as the Multics suite developed by the authors' lab [8], address these limitations by coupling compressibility, leakage, cavitation, body dynamics, contact, and thermal effects. While the Multics-HYGEsim model for EGPs [9][10] is a strong candidate for direct application here, the CCHGP's unique gear morphology and complex axial compensation require a dedicated multi-physics model development and investigation. Furthermore, the loss distribution in CCHGPs is expected to differ from that of EGPs and is thus described in a dedicated section in this paper.

Earlier studies [4][11] established an understanding of CCHGP operation but neglected the dynamics of the axial gear and bushing body, as well as piston-interface leakage. The model

in [12] calculates static axial forces but neglects the dynamic motion and behavior of the compensation system and the lateral interface. The axial compensation system controls lateral gap heights, which directly affect volumetric and torque losses; thus, accurately modeling its dynamic behavior is essential. This paper addresses these gaps by incorporating axial compensation dynamics, meshing friction, and piston leakage into a strongly coupled multi-physics model. This detailed simulation also enables precise analysis of the breakdown and distribution of energy loss, which is validated against experimental results in this paper. Such insights enable direct comparison with EGP loss mechanisms [10]. This model enables hydraulic machine experts who might retrofit EGP housings with continuum gearsets to optimize tolerances and fluid-dynamic features for CCHGPs.

The primary novel contributions of this paper are: (i) The implementation of a strongly coupled multi-physics model for a CCHGP; (ii) Detailed modeling of axial compensation dynamics, film contact and friction losses, meshing friction forces, and axial piston leakage, with results compared against state-of-the-art methods; and (iii) Experimental validation of the model's efficiency and ripple predictions. The remainder of this paper describes the reference CCHGP geometry (Section 2), details the multi-physics model (Section 3), presents results and validation (Section 4), and concludes with key findings.

2. CCHGP GEOMETRY DESCRIPTION

The reference CCHGP (Figure 1.1) consists of the following components:

1. **Drive gear:** Connected via spline to the prime mover.
2. **Driven (slave) gear:** In mesh with the drive gear.
3. **Top bushing:** Located on the gear spline side.
4. **Bottom bushing:** Located opposite the spline side.
5. **Axial piston:** Situated within the casing bottom cover for axial compensation.

The CCHGP employs a compound circular arc-involute ‘continuum’ profile. Unlike standard truncated involute profiles used in EGPs, this geometry ensures a continuous transition of the contact point throughout the meshing cycle, eliminating sudden transitions as new tooth pairs engage [1]. The conjugate root and tip geometry provide almost complete sealing in both tangential and axial directions, reducing the displacement chamber dead volume to near zero. Because this transverse profile yields a low contact ratio (CR), a helix angle is used to achieve a total CR greater than 1; this helix also allows a smooth transition between low and high pressures. Furthermore, the helix causes the root-tip sealing surface to move axially during rotation, creating a secondary compression mechanism. These features yield a theoretical kinematic ripple of zero [4]. However, factors such as axial micromotions, leakage, and compressibility result in a small but non-zero real-world ripple.

Similar to a standard involute EGP's construction, floating bushings are present on either side of the gearset to maintain axial sealing. Each bushing is hydrostatically balanced by a pressurized chamber on its outer axial face, which counters the fluid pressure from the gear side. Because the gear-side forces fluctuate while the balancing force remains static, the bushings' position and orientation vary continuously. Due to the helix angle, the top and bottom bushings experience asymmetric loading, which must be accounted for in the pump's design and modeling. Additionally, each bushing houses two journal bearings, one for each gear that supports the radial loads acting on the gears and shafts. The materials and surface

roughness of these components are generally the same as those used for a standard EGP. The reference unit has split bushings in which the two halves of the bushing compensate each gear independently. Some CCHGP designs instead use a single bushing for the same.

To compensate for the axial load imbalance on the gears caused by helical asymmetry, the reference unit uses axially oriented pistons in the casing bottom cover. The far end of these pistons is pressurized by HP port fluid (see Figure 1.1(b)). These pistons push the axial face of the gear shaft. This compensating force is static, whereas the gears' axial loading is dynamic due to gear motion and film forces. Multiple design variations exist for this axial compensation of gears, using 2 pistons (one for each gear) or 1 (for the drive gear), with some designs even substituting pressurized fluid for pistons to compensate the gears directly. The reference unit consists of one piston pushing on the drive gear. The piston floats freely with clearance, and thus there is leakage from the piston's far-side chamber towards the bushing, into the journal bearings, and eventually into the drain. In this case, the slave gear is axially compensated by the gear-meshing force, which varies over time. This complex axial motion requires detailed modeling of the axial forces on the gears. While the current model analysis is presented for a single-piston design, its versatility allows the study of these different compensation mechanisms with minor adjustments.

3. MULTI-PHYSICS CCHGP MODEL

The CCHGP Multics model is a C++-based, multi-domain, multi-physics model comprising fluid, lubrication, and solid domains. These domains have individual solvers that communicate with each other at each solver step. The three domain solvers solve their respective physics and also exchange information with one another, as described in Figure 3.1. The novelty of this paper lies not in introducing individual solvers, but in demonstrating how they can be integrated to model the performance aspects of a CCHGP. For further details on individual model solvers and their strong coupling, readers are encouraged to refer to [8]. In Figure 3.1, the model physics in black were presented in earlier publications [11], while the features marked in red are the novel contributions of this paper. The solver implementations specific to CCHGP are described briefly in this section.

3.1. Fluid domain modeling

The fluid domain is modeled using a lumped-parameter approach that forms a hydraulic circuit of chambers and flow connections, consisting of 3 component types [4].

Chambers: These are lumped volumes of fluid with uniform state and physical properties. Their governing equation is the isothermal pressure build-up equation (equation (3.1)).

$$\frac{dp_{CV}}{dt} = \frac{K_{CV}}{V_{CV}} \left(\frac{\sum m_i}{\rho_{CV}} - \frac{dV_{CV}}{dt} \right) \quad (3.1)$$

This equation is isothermal. It can account for cavitation and aeration, but it is static, with no time delays or hysteresis effects due to bubble formation/collapse. The model can account for pressure-dependent variations in fluid properties by importing tabulated values, which are interpolated at each solution step. In the CCHGP, the chamber of interest is the Tooth Space Volume (TSV), which is the space between a tooth pair, the outer casing, and the axial sealing surface. The fluid inside the TSV compresses to high pressure to achieve positive displacement. The formulation of TSVs is described in detail in [4].

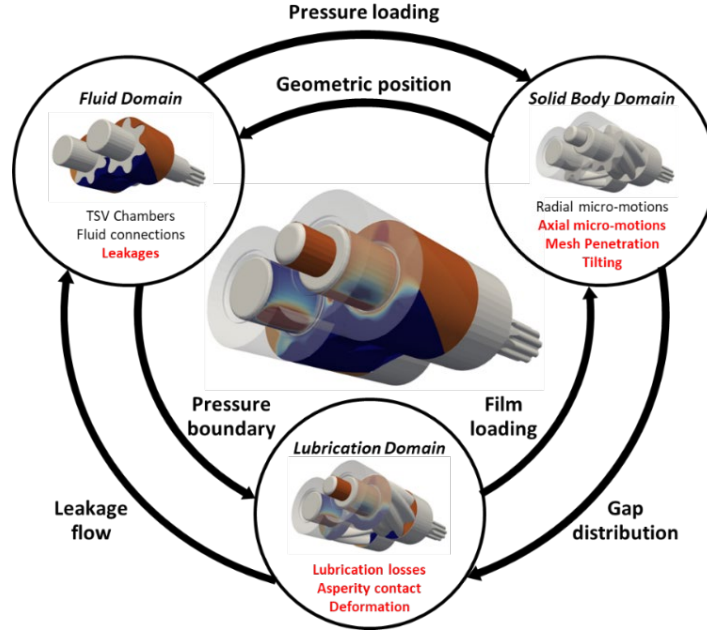


Figure 3.1. Multi-Physics model overview for CCHGP

Orifice connections: The TSVs are connected to each other and to the outlet/inlet ports through fluid connections modeled as orifices, with physics governed by the modified turbulent orifice equation (equation (3.2)).

$$Q_{i,j} = C_q \Omega \sqrt{\frac{2(p_i - p_j)}{\rho}} \text{sign}(p_i - p_j) \quad (3.2)$$

Piston leakage: The piston interface introduces an additional leakage, modeled as a hydraulic circuit. Figure 3.2 shows the leakage path and circuit for a single-piston pump. The flow connections here are modeled as lubrication films as described in Section 3.2.

Tip leakages: The leakage due to radial clearance between gear tooth tip and casing is important because: (a) it leads to volumetric losses in the pump; (b) it controls the pressurization timing of TSVs. For standard involute EGMs, this flow is modeled as a parallel-plate flow [10]. However, since the tip leakage path of the CCHGP is a curved constriction, it is modeled using the approach presented in [13]. The instantaneous local tip gap is calculated from the gears' radial motion and tilting, as explained in Section 3.3.

Geometric preprocessing: The geometric properties used here: chamber volume (V_{cv}) and orifice flow area (Ω) vary with time, but depend only on pump construction, not operating conditions. A geometric preprocessor calculates and stores this information for interpolation during the simulation, to save time. This geometric preprocessing for CCHGPs is described extensively in [4].

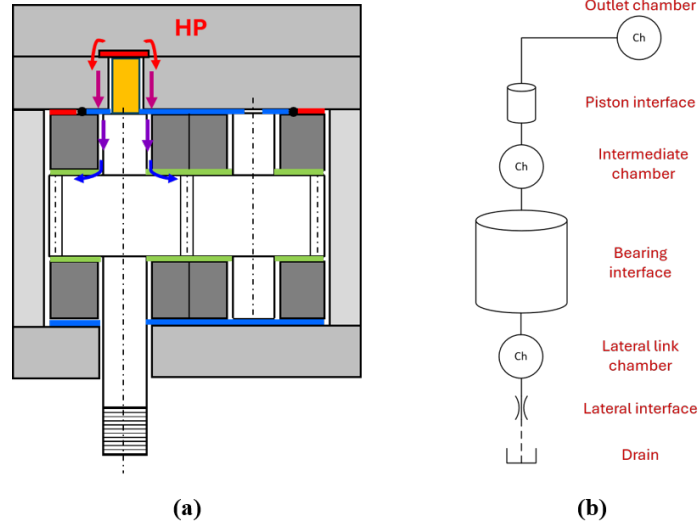


Figure 3.2. (a) Piston leakage path inside the pump, (b) Model hydraulic circuit

3.2. Lubrication interface modeling

Lubricating interfaces are responsible for load bearing and hydrodynamic support. Two consequences of these interfaces, due to their often-small gaps, are volumetric losses from flow leakage and torque losses from shear and contact. These effects were simplified in previous CCHGP models but are modeled here using a numerical 2D CFD approach. The governing equation for fluid-structure interaction effects in lubricating interfaces is the Universal Mixed Reynolds Equation [8] (Equation (3.3)).

$$\nabla \cdot \left(\phi_p \left(\frac{K_T h^3}{12\mu} \nabla \rho \right) \right) = \nabla \cdot \left(\rho \vec{v}_m (\phi_R R_q + \phi_c h) \right) + \nabla \cdot \left(\rho \phi_s R_q \left(\frac{\vec{v}_t - \vec{v}_b}{2} \right) \right) + \frac{\partial \rho (\phi_R R_q + \phi_c h)}{\partial t} \quad (3.3)$$

This equation is isothermal. It can account for both static cavitation and variations in fluid properties with pressure. This model also accounts for the effects of mixed lubrication and surface asperities on pressure build-up and effective gap height.

The source term in this equation is the instantaneous, local gap height at the interface, $(\bar{h}(x, y, t))$. This gap height is influenced by 3 factors:

Rigid body linear and angular motion of the bodies: The motion of the bodies is integrated and solved at each time step and is described in detail in Section 3.3.

Deformation of the solid body surface: This uses an analytical approximation in which the solid bodies are modeled as infinite half-planes [8]. More sophisticated numerical techniques exist [8] that drastically increase computation time. Therefore, to study performance trends of the axial compensation, an analytical approximation of deformation is used.

Surface roughness and asperities: Mixed-lubrication effects are considered using the method proposed by Patir and Cheng [14]. This method defines flow factors based on statistical characteristics of surface roughness to quantify their effect on flow rate. In the case of surface asperity contact, local contact pressure and solid asperity friction are also

calculated from the local gap height using the formulation proposed by [15]. This component is critical for modeling contact in conditions such as over-compensated bushings.

The details of the exact calculations can be found in previous publications by the authors' lab [8] and are therefore not repeated here. Similar to EGPs, there are two specific geometries of lubricating films: (a) polar films for the lateral interface between gear and bushing axial surfaces and (b) cylindrical films for the journal bearings. Additionally, the piston and casing form a cylindrical film with its own dynamics and leakage, which are modeled as an independent Reynolds film with chamber boundary conditions from the piston leakage circuit introduced in Section 3.1.

3.3. Solid domain modeling

The solid bodies float freely in the pump, exhibiting micro-motions. These micro-motions control the instantaneous local gap heights at the lubricating interfaces, thereby directly affecting load-bearing capacity, leakage, and torque losses.

The dynamics of each floating body are governed by Newton's 2nd law of motion for translation and rotation. The equations are solved by adding them to the global system of ODEs and then integrating the resulting system. However, it is important first to understand and calculate the net forces ($\sum F_i$) and moments ($\sum M_i$) acting on each specific component.

Drive and slave gears: The gears are capable of 4 types of movement, corresponding to their 6 degrees of freedom.

- (i) Radial/transverse micro-motions of the gears inside the bearing (x, y).
- (ii) Tilting of the gears with respect to the nominal bearing orientation (θ_x, θ_y).
- (iii) Axial micro-motion of the gears (z).
- (iv) Relative rotation/penetration of the gears (θ_z).

The previous CCHGP model [11] neglects motions (iii) and (iv), while simplifying (ii) due to the absence of coupling between body dynamics and lubricating interfaces. This paper relaxes these assumptions by modeling all of the gears' DOFs and quantifying their effects. This is crucial to accurate performance estimation of the axial compensation.

The forces acting on the gears in radial and axial directions are shown in Figure 3.3. These can be classified into 4 sources: (a) TSV pressure forces; (b) Lubricating gap pressure and shear forces; (c) Piston axial compensation forces; (d) Meshing contact and friction forces.

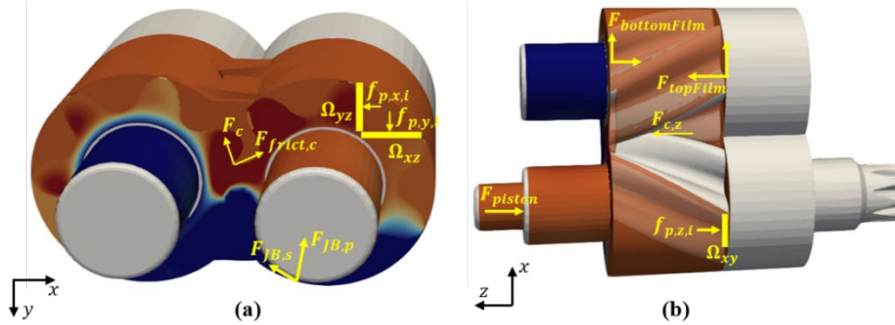


Figure 3.3. Forces acting on gears (a) radially, and (b) axially

TSV pressure loading depends on the TSV projection areas on the gears and their moment arms in the (x, y, z) directions. These quantities are calculated and stored by the geometric preprocessor [4].

The lubricating interfaces (journal bearing and lateral gap) exert both normal and tangential forces relative to the solid body surface. Normal forces arise from fluid-film pressure or contact, and tangential forces comprise film shear and asperity friction. Interface pressure calculations are handled by the lubrication interface solver (Equation 3.3). The film friction loss calculations are described in Equation 3.4.

$$\vec{F}_{fric} = \int \vec{\tau} \cdot d\vec{A} + \int P_c \left(-\frac{\vec{v}_t - \vec{v}_b}{|\vec{v}_t - \vec{v}_b|} \right) \cdot d\vec{A} \quad (3.4)$$

The first term represents viscous shear, while the second represents asperity contact friction.

The pistons apply an axial compensating force to the gear, modeled as a simple static force acting independently on each gear.

The gear meshing contact force is modeled as a spring-damper system based on the relative penetration between the gears. It is assumed that contact occurs only in the involute section of the drive flanks. The equation of force is given by Equation 3.5.

$$F_{contact} = k\delta + c\dot{\delta} \quad (3.5)$$

The teeth meshing penetration (δ) is the superposition of 2 effects: relative angular rotation of the gears about the z-axis and axial micro-motions of the gears. The penetration velocity ($\dot{\delta}$) is obtained by taking the time derivative of the instantaneous penetration (Equation 3.6).

$$\delta = r_{base}\Delta\theta_z \cos(\beta) + \Delta z \sin(\beta) \left(-\text{sign}(\Phi_{helix}^{drive}) \right) \quad (3.6)$$

The stiffness and critical damping ratio of the contact are calculated using Hertzian theory.

Once the force is calculated, the moment on the gears due to contact forces can be determined from the 3D geometric location of the contact point.

Lastly, the friction forces (Equation (3.7)) and, subsequently, the moments are calculated using a coefficient of friction, with the correlation for EHL line contact based on [16].

$$F_{friction} = \mu_{EHL} F_{mesh} \quad (3.7)$$

The friction coefficient depends on fluid viscosity, which can change by orders of magnitude due to density changes in the contact region [16]. The model can input tabulated variations in fluid properties and therefore accurately model this behavior during meshing.

The drive gear is assumed to be rotating with a constant z-axis rotation velocity, and any torque imbalance is overcome by the prime mover. The slave gear is free to spin around its z-axis, with its angular position affecting the penetration. All other degrees of freedom for both gears are allowed, and their linear and angular positions are independently determined by integrating and solving the body-dynamics ODEs.

An important distinction between EGPs and CCHGPs is the tooth tip torque loss. As demonstrated in [10], tooth-tip losses contribute significantly to the overall loss of EGPs. In the case of the CCHGP, the hydrodynamic shear stress from tip-leakage flow acts on a very

small area, contributing very little to the overall torque loss. Although this torque loss is modeled, it is not reported because its effect is negligible.

Top and bottom bushings: The lateral bushings control clearances with the gears' lateral surfaces by pressurizing the balancing chambers from behind, mimicking the pressure distributions of the gear TSVs and the lateral lubricating film. However, the compensation force is static, whereas the gear-side force varies over time. This leads to dynamic periodic motion in the bushing. The bushing is expected to be in contact with the casing in the LP region, resulting in a frictional force during its motion, which is applied as an external force on the bushing. To simplify the body dynamics, the current model assumes the split bushing halves move together without relative separation or tilt. Modeling the independent half bushings is out of scope for this paper.

4. RESULTS AND DISCUSSION

This section will describe the simulations conducted on a reference CCHGP unit. This is a 20 cm³/rev 7-tooth pump with one-piston compensation. For confidentiality reasons, the design dimensions of the tested pump have not been disclosed. However, the tested pump was opened, and precision metrological measurements were taken to recreate the pump for simulations.

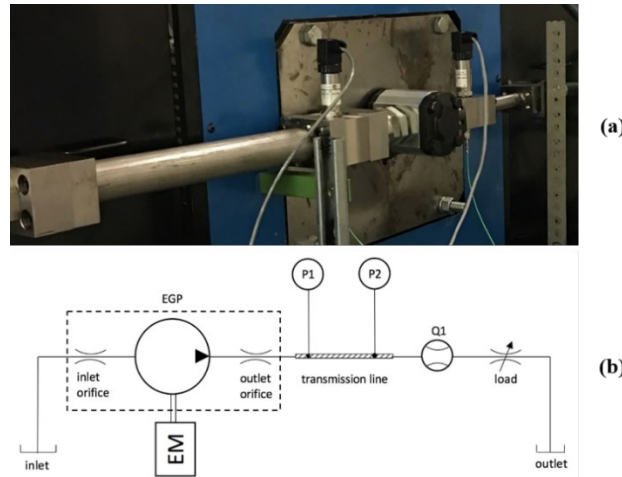


Figure 4.1. (a) Maha Multi-Purpose Test Rig with CCHGP, (b) Test circuit.

Sensor	Model
Outlet Pressure	Wika S-10
Inlet Pressure	Wika A-10
Pressure Ripple	PCB Piezotronics 108A02
Flow rate	VSE VS 4 Gear flowmeter
Shaft torque	HBM T10FS Torquemeter

Table 4.1. Sensors used for experimental measurements

To validate the model experimentally, tests were conducted using a steady-state test setup at the authors' lab, as shown in Figure 4.1. The focus of these experiments is to demonstrate

how the model captures macroscopic trends in efficiency under certain reference operating conditions as supported by experimental validation. In the future, the authors aim to incorporate additional results for a larger range of operating conditions. The simulations and experiments were conducted at a fixed temperature of 50 °C, with ISO VG 46 hydraulic oil as the fluid. The sensors used in the experiments are listed in Table 4.1.

4.1. Efficiency results

The steady-state efficiency trends of the pump were compared across experiments, the current simulation model, and the baseline simulation model proposed in [11] at different pressures and a fixed speed of 2000 RPM. The differences in the model assumptions are summarized in Table 4.2. The results are plotted and compared in Figure 4.2.

Parameters/Assumptions	Baseline	Proposed
Gear axial (Z) motion & compensation	Fixed	Solved dynamically
Bushing axial (Z) motion	Fixed	Solved dynamically
Lateral film gap heights	Assumed symmetric	Solved independently
Film contact & mixed lubrication	Ignored	Solved
Deformation	Ignored	Solved
Gear meshing penetration & friction	Ignored	Solved
Piston interface leakage	Ignored	Solved

Table 4.2. Model assumption comparison between the baseline and the proposed.

The volumetric efficiency prediction has improved for some operating conditions while preserving trends; however, some discrepancies remain. The current model assumes a single-bushing design with simplified DOFs and deformation. Furthermore, the meshing flow connections are modeled as simple orifices, and the effects of gear tilting and manufacturing tolerances are ignored, all of which may affect the predicted volumetric efficiency. Future versions of the model can aim to address these assumptions.

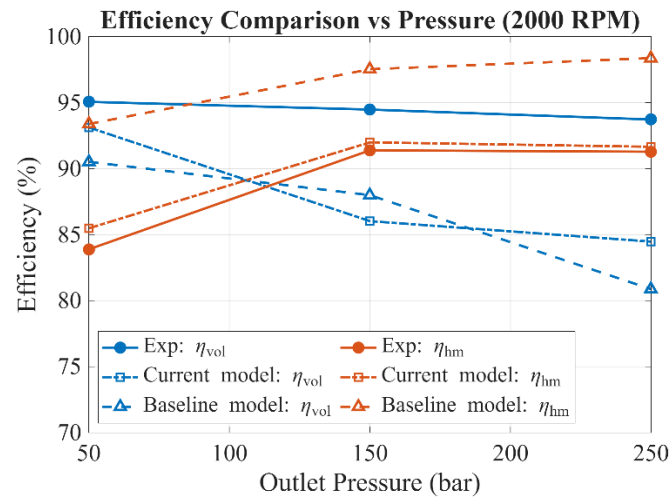


Figure 4.2. Comparison of hydromechanical (in red) and volumetric (in blue) efficiencies for the CCHGP between experiments and simulation models.

The hydromechanical efficiency prediction has improved significantly compared to the baseline model. To investigate this further, the mechanisms of torque loss are analyzed. There are 3 major torque-loss sources in the CCHGP: lateral lubrication, meshing, and journal bearing losses. The currently proposed model in this paper can model these losses, and their breakdown is presented here. The torque loss at the pump shaft seal is ignored in this model. The torque loss breakdown is presented in Figure 4.3. From the figure, it is clear that while the gear meshing and JB film loss predictions are largely consistent across models, the lateral film torque loss differs significantly. The baseline model severely underpredicts the lateral film losses compared to the current model.

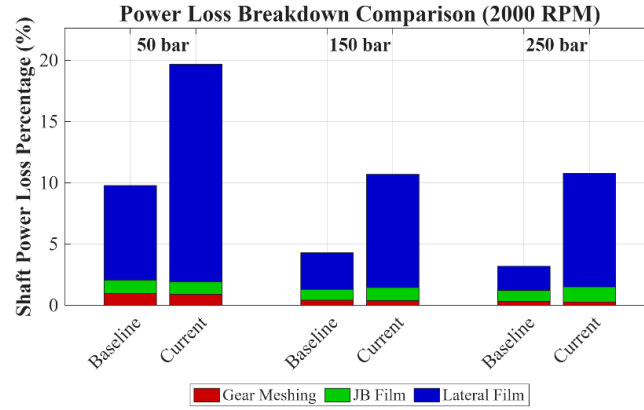


Figure 4.3. Distribution of torque losses as predicted by each model.

From Table 4.2 it can be seen how these model assumption relaxations contribute to better model predictions. Solving gear and bushing motion dynamics allows accurate estimation of the lateral gap height distribution, and a mixed-lubrication model in the films enables more accurate estimation of asperity friction losses. To demonstrate this, a more detailed breakdown of gap height distribution is needed.

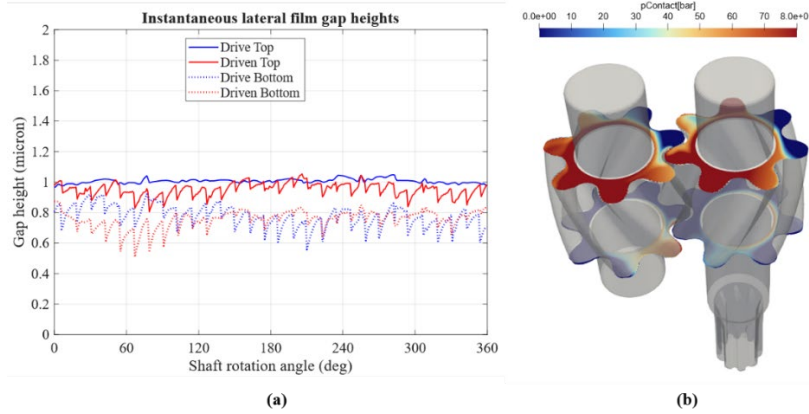


Figure 4.4. (a) Instantaneous minimum lateral gap heights predicted by the model ($\Delta P = 250 \text{ bar}$, $N = 2000 \text{ RPM}$), (b) Contact pressure in lateral films

The baseline model assumes a fixed, uniform rigid gap height at the lateral interface. For this simulation, the gap height is set to 1.5 microns to model full hydrodynamic lubrication, as assumed in [11]. However, in the proposed model, all 4 bodies (2 gears and 2 bushings) are allowed to move axially and deform. The resulting gap heights thus fall within the mixed-lubrication regime (see Figure 4.4(a)), leading to a buildup of contact pressure at the lateral interface (see Figure 4.4(b)). Surface asperity contact and friction increase the opposing torque exerted by the lateral interface, thereby increasing hydromechanical losses. The contribution from surface asperity friction can be calculated as the difference between the lateral film losses of the Proposed model with contact and those of the Baseline model without contact. It can be seen here that under all 3 operating conditions, the asperity friction loss dominates the lateral film loss.

The current model can not only model this interaction to more accurately capture losses in the CCHGP but also predict the effectiveness of a compensation mechanism arising from body dynamics. This can be useful for conducting a sensitivity study on the bushing balance areas while designing them for CCHGP.

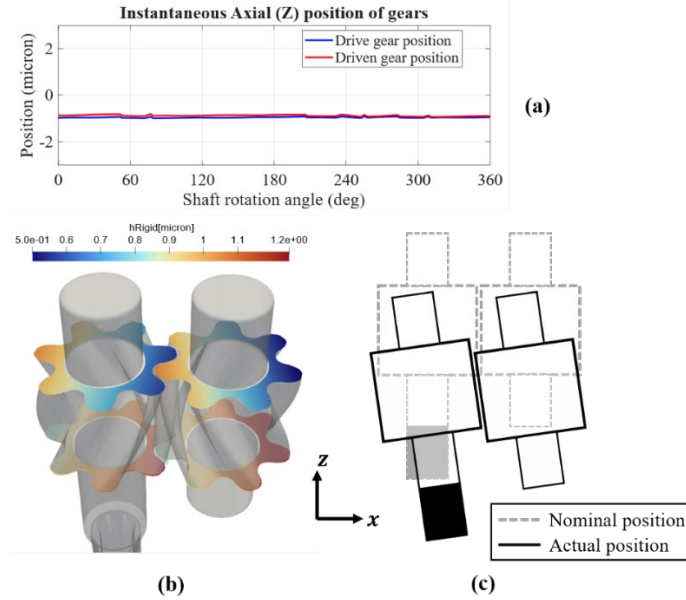


Figure 4.5. (a) Axial position of gears; (b) Lateral rigid gap heights showing tilting, (c) Comparison of the gears' nominal and actual positions (exaggerated)

4.2. Gear micro-motions

The model offers detailed insights into pump physics, some of which are presented here. Figure 4.5 shows the position of the gears as predicted by the model. It can be seen clearly that the gears tend to move axially towards $-z$ (towards the driveshaft) and away from the piston side, as expected from the orientation of the pressure force. Furthermore, the gap height distribution shows that both gears tilt in the same direction. Thus, the casing length must be designed to accommodate the gearset's range of motion, and any gear profile micro-corrections must also account for the magnitude of tilting.

4.3. Pressure ripple results

The model can also predict the pressure ripple, which can be compared to experimental results. For the experiments, a PCB Piezotronics Model 108A02 ICP Pressure Sensor was utilized to measure the outlet pressure ripple. The pressure ripple comparison between experiments and simulations for two operating conditions is shown in Figure 4.6. The results show good agreement between the simulated and experimental pressure ripples in amplitude and frequency. The difference can be attributed to the CCHGP's pressure ripple order-of-magnitude being much smaller than that of other pumps, making it more sensitive to factors such as external excitations, manufacturing defects, and sensor noise than most other measurements.

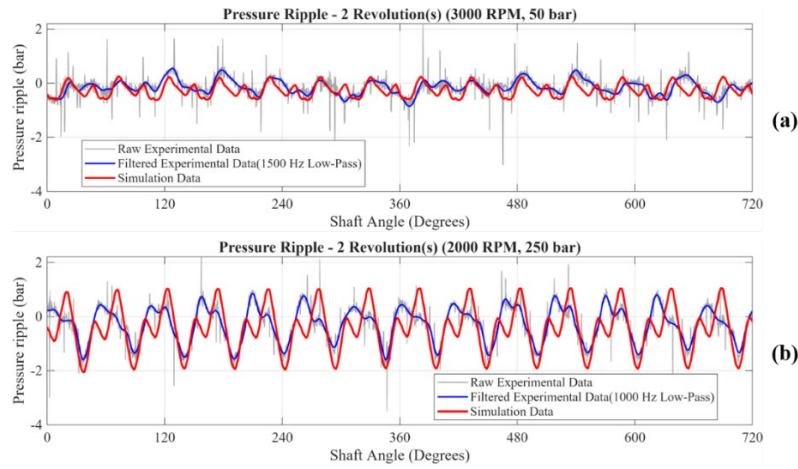


Figure 4.6. Comparison of pressure ripples from simulations and experiments for (a) $\Delta P = 50 \text{ bar}$, $N = 3000 \text{ RPM}$, (b) $\Delta P = 250 \text{ bar}$, $N = 2000 \text{ RPM}$

5. CONCLUSIONS

This paper presents a multi-domain, multiphysics numerical model for analyzing a Continuous Contact Helical Gear Pump. The novel additions to the model are presented, and the model was applied to a reference CCHGP unit as a case study, with experimental validation. The results show considerable improvement in predicting steady-state efficiency trends compared to state-of-the-art models. The hydromechanical efficiency trends now match experiments much more closely than state-of-the-art models. The volumetric efficiencies do not match as closely and will require a more in-depth investigation and modeling effort. A detailed breakdown of torque losses and their contribution to hydromechanical efficiency was also presented. Furthermore, the pressure ripple predicted by the model also shows good agreement with experiments. This model has applications in the design of axial compensation bushings and pistons for a CCHGP, as it can accurately predict losses at the lateral interface and the dynamics of the floating bodies. In the future, the authors intend to validate the model for more operating conditions. Furthermore, as with EGPs, the deformation of the bushing and casing can be captured more accurately using a numerical influence matrix approach. The authors also plan to conduct a detailed CFD analysis of leakage and flow in the meshing zone to understand volumetric losses better and

improve the model's prediction of volumetric efficiency. The authors also plan to validate the simulation's prediction across multiple speeds in the future. The complete model is highly versatile and can provide users with detailed insight into pump operation to develop novel CCHGP designs.

6. REFERENCES

- [1] K. Seki and K. Mitome, "A new continuous contact low-noise gear pump," *Journal of Mechanical Design*, vol. 105, no. 4, pp. 736–741, Dec. 1983, doi: 10.1115/1.3258544.
- [2] M. A. Morselli, "Rotary positive-displacement pump with meshing gear wheels without encapsulation, and gear wheel for such a positive-displacement pump," US 6,769,891 B2, Aug. 03, 2004
- [3] M. A. Morselli, "Mechanical and Hydraulic Noise of Gear Pumps: The Drastic Evolution Marked by 'Continuous Contact' Pumps."
- [4] X. Zhao and A. Vacca, "Analysis of continuous-contact helical gear pumps through numerical modeling and experimental validation," *Mech. Syst. Signal Process.*, vol. 109, pp. 352–378, Sep. 2018, doi: 10.1016/j.ymssp.2018.02.043.
- [5] H. Jang and S. Ji, "Numerical Study on the Instantaneous Theoretical Flow Rate of the Continuous Contact Gear Pump with a New Geometrical Approach," *J. Mar. Sci. Eng.*, vol. 12, no. 12, Dec. 2024, doi: 10.3390/jmse12122332.
- [6] C.-K. Chen and S.-C. Yang, "Geometric modelling for cylindrical and helical gear pumps with circular arc teeth," *Proc. Inst. Mech. Eng. C J. Mech. Eng. Sci.*, vol. 214, no. 4, pp. 599–607, 2000, doi: 10.1243/0954406001523939.
- [7] X. Zhao, A. Vacca, and S. Dhar, "Numerical Modeling of a Helical External Gear Pump With Continuous-Contact Gear Profile: A Comparison Between a Lumped-Parameter and a 3D CFD Approach of Simulation," in *Proceedings of the BATH/ASME 2018 Symposium on Fluid Power and Motion Control.*, ASME, 2018.
- [8] T. Ransegnola, "A Strongly Coupled Simulation Model Of Positive Displacement Machines For Design And Optimization," PhD Thesis, Purdue University, 2020.
- [9] A. Pawar, A. Vacca, and M. Rigosi, "Comparative Analysis of External Gear Machine Performance Considering Deformation and Thermal Effects," *International Journal of Fluid Power*, vol. 25, no. 4, pp. 465–492, 2024, doi: 10.13052/ijfp1439-9776.2543.
- [10] A. Pawar, V. Harish, B. Manne, A. Vacca, and M. Rigosi, "Analysis of Torque Efficiency of External Gear Machines Considering Gear Teeth Roughness," *Mech. Mach. Theory*, vol. 199, Sep. 2024, doi: 10.1016/j.mechmachtheory.2024.105675.
- [11] X. Zhao and A. Vacca, "Multi-domain simulation and dynamic analysis of the 3D loading and micromotion of continuous-contact helical gear pumps," *Mech. Syst. Signal Process.*, vol. 163, Jan. 2022, doi: 10.1016/j.ymssp.2021.108116.

- [12] Y. fei Wu, P. qi Ge, and W. bo Bi, “Analysis of axial force of double circular arc helical gear hydraulic pump and design of its balancing device,” *J. Cent. South Univ.*, vol. 28, no. 2, pp. 418–428, Feb. 2021, doi: 10.1007/s11771-021-4612-2.
- [13] R. Rituraj and A. Vacca, “Investigation of flow through curved constrictions for leakage flow modelling in hydraulic gear pumps,” *Mech. Syst. Signal Process.*, vol. 153, May 2021, doi: 10.1016/j.ymssp.2020.107503.
- [14] N. Patir and H. S. Cheng, “An Average Flow Model for Determining Effects of Three-Dimensional Roughness on Partial Hydrodynamic Lubrication,” *J. Tribol.*, pp. 12–17, 1978.
- [15] S. C. Lee and N. Ren, “Behavior of elastic-plastic rough surface contacts as affected by surface topography, load, and material hardness,” *Tribology Transactions*, vol. 39, no. 1, pp. 67–74, Jan. 1996, doi: 10.1080/10402009608983503.
- [16] B. J. Hamrock, S. R. Schmid, and B. O. Jacobson, *Fundamentals of Fluid Film Lubrication*. CRC Press, 2004. doi: 10.1201/9780203021187.

CFT Cartridge Flow Meters: Enabling Continuous In-Circuit Hydraulic Diagnostics

Dale G. Dietel

Alpena, MI, United States

ddietel@DGDFluidPower.com

March 2026

Abstract

Hydraulic system diagnostics have historically been constrained by the inability to measure flow directly within integrated hydraulic circuits. Conventional flow meters typically require external plumbing, additional fittings, and manual installation during diagnostic procedures, increasing system complexity and discouraging continuous flow monitoring.

This paper presents the **CFT Cartridge Flow Meter**, a turbine-based flow measurement architecture that is fully integrated within a standard cartridge valve cavity. By embedding sensing, signal processing, and mechanical flow measurement elements inside the cartridge envelope, the CFT makes flow a permanently measured variable without altering the existing hydraulic architecture.

This paper describes the design philosophy, mechanical and sensing architecture and key engineering solutions that enable low loss repeatable measurement in confined geometries. It then outlines system level benefits such as earlier fault detection, remote monitoring of distributed hydraulic systems and closed loop control based on real time flow feedback. The results demonstrate that cartridge-integrated flow measurement provides a practical and scalable foundation for predictive maintenance and digitally connected hydraulic equipment.

KEY WORDS—Hydraulic diagnostics, flow measurement, cartridge valves, turbine flow meters, condition monitoring, in-circuit sensing.

I. INTRODUCTION

Flow is one of the most critical parameters in hydraulic systems, yet it remains among the least accessible during normal operation. While pressure and temperature are commonly monitored, flow measurement has traditionally relied on external devices that are connected temporarily or installed outside the primary hydraulic circuit. This approach limits diagnostic resolution, delays fault detection, and discourages continuous monitoring.

The **CFT Cartridge Flow Meter** addresses these limitations by redefining flow measurement as an intrinsic function of the hydraulic circuit rather than an external diagnostic accessory. In one embodiment, the CFT fits entirely within a standard cartridge valve cavity, enabling permanent, protected, and real-time flow measurement without requiring manifold redesign or additional plumbing. The cartridge-integrated architecture further provides a platform extensible to additional cavity formats and sensing modalities. This paper first contrasts conventional external flow meters with cartridge integrate sensing and then details the CFT mechanical and electronic architecture, addresses key engineering challenges and concludes with system level benefits and directions for future development.

II. LIMITATIONS OF TRADITIONAL FLOW MEASUREMENT

Conventional hydraulic flow meters typically require external plumbing loops, additional fittings and hoses, and dedicated installation space. In many applications, flow meters are installed only during troubleshooting and removed afterward due to cost, leak risk, or space constraints.

These limitations introduce several disadvantages:

1. Increased potential for leakage and contamination
2. Higher installation and maintenance costs
3. Poor suitability for mobile or hazardous environments
4. Incompatibility with continuous monitoring strategies

As a result, flow data is often unavailable during normal system operation, preventing early fault detection and masking inefficiencies that could otherwise be identified through trend analysis.

III. CONCEPT ORIGIN AND DESIGN PHILOSOPHY

The development of the CFT Cartridge Flow Meter originated from a fundamental question: *Why has in-circuit hydraulic flow measurement long been considered impractical?* Addressing this question required re-examining long-standing assumptions related to space constraints, sensor survivability, and measurement accuracy within hydraulic manifolds.

The resulting design philosophy prioritizes integration, robustness, and serviceability. Rather than maximizing laboratory-grade accuracy at the expense of deployability, the CFT architecture emphasizes repeatable, real-time measurement within the constraints of practical hydraulic systems. This approach supports a cartridge-style diagnostic device that preserves standard cavity dimensions while remaining extensible to additional form factors and performance requirements.

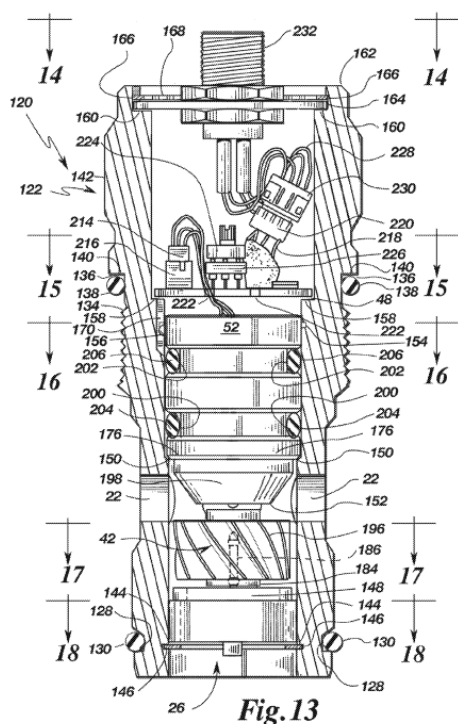
IV. MECHANICAL AND SENSING ARCHITECTURE

A. Cartridge Integration

In one embodiment, the CFT is contained within the envelope of a standard cartridge valve cavity. This integration eliminates external flow paths and preserves existing hydraulic architecture, allowing installation in both new and legacy manifolds. The cartridge format provides mechanical protection for sensitive components and enables modular replacement during service operations.

Fig. 1 illustrates an internal architecture of the CFT Cartridge Flow Meter, showing how the turbine impeller, sensing elements, and electronics may be integrated within the cartridge body while maintaining uninterrupted primary flow through the hydraulic circuit. The cartridge-integrated approach is extensible to multiple cavity sizes while preserving in-circuit flow measurement functionality.

U.S. Patent Jan. 27, 2026 Sheet 8 of 11 US 12,535,344 B2



U.S. Patent Jan. 27, 2026 Sheet 10 of 11 US 12,535,344 B2

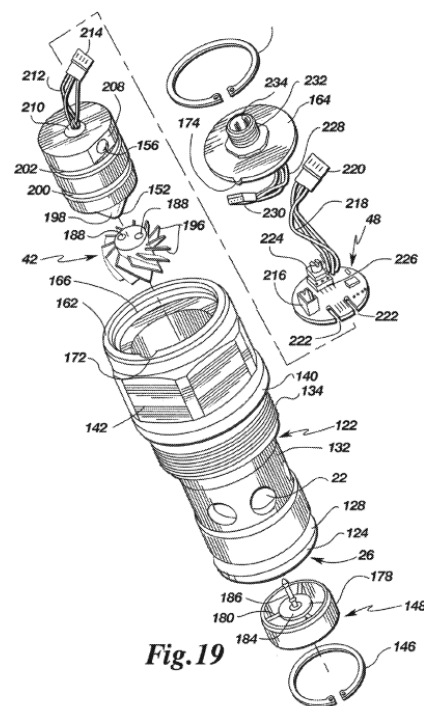


Fig. 1. Internal architecture of a cartridge-integrated flow measurement device. The cross-section illustrates a turbine impeller positioned within the primary flow path, Hall-effect sensing elements embedded in the cartridge body, and a compact electronics package contained within a protected cartridge envelope. Adapted from U.S. Patent No. 12,535,344, Fig. 13 & Fig. 19. Additional Patents: US Patent No. 11,614,351 & US Patent No. 11,892,331

B. Turbine and Impeller Design

At the core of the CFT architecture is a turbine impeller configured to rotate in response to hydraulic flow. In various embodiments, impeller geometry may be optimized to balance pressure drop,

sensitivity at low flow rates, and stability across a wide operating range. This optimization enables diagnostic insight while minimizing hydraulic disturbance.

Scaling of impeller geometry and flow passages enables adaptation to different flow ranges and cavity sizes, supporting performance optimization across diverse system requirements.

C. Hall-Effect Sensing and Electronics

Flow induced impeller rotation is detected using a noncontact Halleffect sensing architecture integrated within the cartridge body. Small magnetic features integrated into the impeller pass in close proximity to stationary Hall effect sensor elements embedded in the cartridge, generating a series of electrical pulses proportional to impeller speed and thus volumetric flow rate for the given geometry. Because the sensors are magnetically coupled to the impeller rather than mechanically linked, they are fully isolated from the hydraulic fluid and do not introduce additional wear interfaces. This configuration is well suited to high pressure, high vibration environments and compact manifolds where space for traditional transducers is limited. The conditioned pulse stream is processed by embedded electronics that convert impeller speed into calibrated flow outputs for real-time monitoring and control. The same electronics platform supports multiple interface options, including 4–20 mA current loop, analog voltage ranges, and J1939 CAN-bus communication, allowing the CFT to connect directly to legacy PLCs, modern ECUs, and networked vehicle architectures.

Additional detail on the Hall effect sensing principle, sensor configurations, and magnetic feature geometry is provided in Appendix 1.

V. ENGINEERING CHALLENGES AND SOLUTIONS

The development of cartridge-integrated flow measurement required addressing several engineering challenges, including minimizing pressure loss within confined geometry, ensuring sensor stability in high-pressure and high-vibration environments, achieving repeatable and linear outputs and protecting electronics from hydraulic fluid exposure.

These challenges are addressed through mechanical optimization, robust sensor placement, and integrated electronics packaging. Various aspects of the architecture have been formalized through U.S. patents covering mechanical, sensing, and system-level innovations.

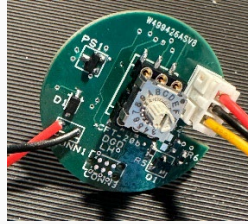


Fig. 2. Embedded electronics used in the 4–20 mA and analog voltage (VDC) versions of the CFT Cartridge Flow Meter. The electronics condition and process Hall-effect pulse data to generate calibrated flow-proportional analog outputs. A **16-position rotary switch** allows selection of the optimum flow range for each application, enabling flexible scaling and compatibility with a wide range of hydraulic system requirements.

VI. SYSTEM-LEVEL BENEFITS

The cartridge integrated flow measurement architecture enables diagnostic and control capabilities that are difficult or impractical to achieve with traditional external flow meters. By making flow a permanently measured variable inside the manifold, the CFT Cartridge Flow Meter provides continuous visibility into how hydraulic energy is actually being delivered to critical functions, rather than inferring performance only from pressure or actuator motion. Continuous in circuit flow data supports earlier detection of inefficiencies and component wear. Gradual changes in flow at constant command can indicate internal leakage, pump degradation, or valve erosion well before these issues appear as lost speed or failed cycles. This enables predictive maintenance strategies in which service actions are scheduled based on measured degradation trends instead of fixed time intervals or unexpected failures. In distributed or inaccessible hydraulic systems, such as mobile equipment and remote industrial installations, permanent flow monitoring also improves operational safety and uptime. Flow data can be logged and analyzed over time, used to verify proper function after software or hardware changes, and checked remotely to confirm that circuits are operating within expected ranges. When combined with pressure and temperature information, in circuit flow measurement supports more accurate system models and control algorithms, improving overall efficiency and dynamic response. For closed-loop hydraulic control, direct flow feedback enables functions that are not feasible when only pressure or position is measured. Examples include actively limiting flow to protect sensitive actuators, enforcing ramp profiles during startup and shutdown, and adapting control parameters as components age. Because the CFT architecture fits within standard cartridge cavities, these capabilities can be added without redesigning manifolds or introducing external plumbing, allowing both new designs and existing platforms to adopt continuous flow diagnostics with minimal disruption.

A. Predictive maintenance examples

- In a hydraulic power unit driving multiple actuators, continuous in circuit flow measurement at the pump outlet can reveal a slow increase in leakage long before operators notice reduced speed or pressure loss. A gradual rise in flow at a fixed command and pressure setpoint indicates internal

wear, allowing maintenance to schedule pump replacement during planned downtime instead of responding to an unexpected failure.

- In a manifold controlling a critical cylinder on production machinery, a CFT installed in the cylinder supply line can detect subtle changes in flow dynamics as valve clearances change or spool edges erode. Deviations from the learned “normal” flow profile for each stroke can be used to flag emerging valve faults, enabling technicians to replace or service the valve before it sticks or causes a process interruption.
- On mobile equipment connected via J1939, multiple cartridge integrated flow meters can stream flow data to the vehicle ECU and telematics platform alongside existing pressure and temperature signals. Fleet operators can correlate changes in commanded versus measured flow with duty cycle and environment, identifying machines that are trending toward cavitation, filter blockage, or abnormal internal leakage and prioritizing those assets for inspection

VII. OUTPUT INTERFACES AND SYSTEM INTEGRATION

To support integration across a wide range of hydraulic platforms, the CFT Cartridge Flow Meter is available with multiple electrical output options. These outputs allow the same cartridge integrated flow hardware to interface with legacy control systems, modern electronic controllers, and networked vehicle architectures commonly used in mobile equipment.

In current implementations, the CFT platform supports **three primary output formats**: analog current, analog voltage, and digital CAN-bus communication. Each output format addresses distinct system requirements and integration preferences.

A. Analog Current Output (4–20 mA)

This format is particularly attractive for plant and maintenance engineers retrofitting existing PLCbased systems where long cable runs and electrical noise are common.

B. Analog Voltage Outputs (1–10 V and 0.5–5 VDC)

These outputs align with electronic control units and embedded controllers on compact machinery, where short harnesses and high resolution analog to digital conversion are available.

C. J1939 CAN-Bus Digital Output

In addition to analog signaling, the CFT is available with a **J1939 CAN-bus digital output**, supporting direct integration into networked mobile equipment architectures. J1939 is widely adopted in off-highway, agricultural, construction, and commercial vehicle applications, providing a standardized, robust communication framework for distributed electronic systems.

The use of J1939 offers several system-level advantages, including:

- **Reduced wiring complexity**, as multiple devices share a common bus
- **Improved noise immunity** through differential signaling
- **Standardized message structures** that simplify system integration
- **Scalability**, enabling multiple CFT devices to coexist on the same network
- **Direct compatibility with modern vehicle ECUs and telematics systems**

By transmitting flow data digitally over the CAN-bus, the CFT can support real-time monitoring, coordinated control strategies, and remote diagnostics without additional analog signal conditioning. In networked systems, flow information can be shared across controllers, logged for trend analysis, or transmitted off-board for fleet-level monitoring and predictive maintenance.

The availability of J1939 output further aligns the CFT platform with current and emerging mobile equipment architectures, positioning cartridge-integrated flow measurement as a native element within digitally connected hydraulic systems.

As a result, a cartridge integrated CFT can appear as just another node on the vehicle network, streaming flow data to ECUs and telematics systems alongside existing pressure, speed, and position sensors.

VIII. APPLICATIONS AND IMPACT

Cartridge-integrated flow measurement is applicable to mobile equipment, industrial automation, distributed hydraulic networks, and safety-critical or inaccessible systems. Permanent flow visibility fundamentally changes how engineers observe system health, diagnose faults, and optimize performance across the equipment lifecycle.

IX. FUTURE WORK

Future development of the CFT Cartridge Flow Meter will focus on expanding sensing capability and optimizing performance through architectural scalability, while preserving the cartridge-integrated diagnostic philosophy.

In future embodiments, **temperature sensing may be integrated across CFT variants**, enabling synchronized flow and temperature measurement at the point of use. Hydraulic fluid temperature directly influences viscosity, internal leakage, and volumetric efficiency, and its integration within the cartridge enables improved diagnostic correlation, compensation algorithms, and system modeling. Temperature sensing may be implemented using one or more sensor technologies integrated within the cartridge body and interfaced through existing or expanded electrical connections.

Future work also includes support for **additional cartridge cavity sizes**. Scaling the cartridge-integrated flow measurement architecture to multiple cavity formats enables optimization of impeller geometry, pressure drop, and measurement resolution across broader flow ranges and operating conditions. Multiple cavity sizes further support adaptation to diverse manifold standards and performance requirements.

Collectively, these developments extend the CFT platform toward a **multi-parameter, scalable in-circuit diagnostic architecture**, supporting future implementations that integrate flow, temperature, and additional sensing functions without external instrumentation.

X. CONCLUSION

The CFT Cartridge Flow Meter demonstrates that real-time, in-circuit flow measurement can be practical, reliable, and scalable within standard hydraulic manifolds. By integrating a turbine flow

element, noncontact Hall effect sensing, and embedded electronics into a standard cartridge valve cavity, the CFT eliminates external flow loops' potential leak paths and makes flow a permanently measured variable alongside other key system health data like pressure and temperature. Continuous in-circuit flow data enables earlier fault detection, supports predictive maintenance and remote monitoring, and allows closed-loop hydraulic control based on direct flow feedback rather than indirect symptoms. The same cartridge integrated architecture provides an extendable platform for multiparameter sensing, including future integration of fluid temperature and additional diagnostics, positioning flow cartridges as native elements of smarter, more connected hydraulic systems

REFERENCES

- [1] U.S. Patent No. 12,535,344, *Cartridge-Integrated Flow Measurement System*, 2025.
- [2] R. W. Fox, A. T. McDonald, and P. J. Pritchard, *Introduction to Fluid Mechanics*, 9th ed. Hoboken, NJ, USA: Wiley, 2016.
- [3] ISO 1219-1, *Fluid Power Systems and Components—Graphical Symbols*, International Organization for Standardization.
- (4) U.S Patent No. 11,614,351
- (5) U.S. Patent No. 11,892,331

Appendix:

1.

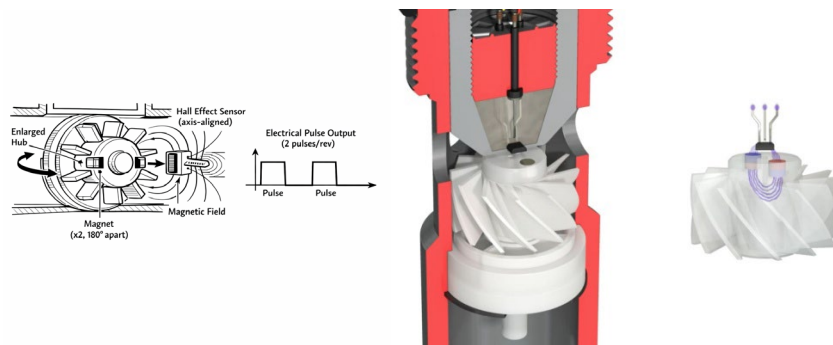


Fig. 3 - Axis-aligned Hall-effect sensor. Two magnets embedded in the hub and positioned 180° apart pass a stationary Hall-effect sensor aligned with the impeller axis, producing two electrical pulses per revolution. The resulting pulse frequency is proportional to impeller rotational speed and volumetric flow rate. Refer to U.S. Patents: No. 11,614,351 / No. 11,892,331 / No.12,535,344

2.



Here's a simplified sketch of the test stand circuit:

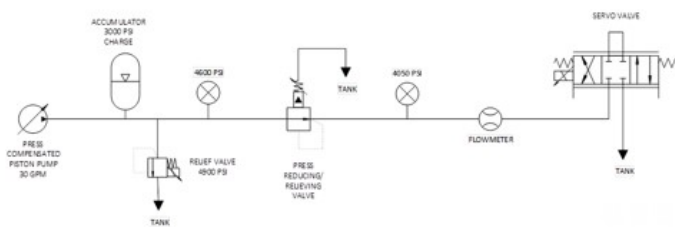


Fig. 4 - CFT Cartridge Flow Meter – shown prior to and after installation in a hydraulic manifold

Ultra Compact High-Speed Axial Piston Pumps for Electrified Mobile Machinery: Optimizing and Testing Tribological Gaps for 10,000 rpm

Roman Iwantysyn and Jürgen Weber

*Institute of Mechatronic Engineering, Technische Universität Dresden,
Helmholtzstrasse 7a, 01069 Dresden,
E-mail: roman.iwantysyn@tu-dresden.de, fluidtronik@mailbox.tu-dresden.de*

Abstract.

High-speed electric drives are being adopted in mobile machinery, with automotive e-motors commonly operating up to 10,000 rpm and beyond. To exploit this speed range in electro-hydraulic powertrains, hydrostatic units must be qualified and redesigned for ultra-high rotational speed, where tribological limits are often the limiting factor, when boosting the inlet pressure. In the present work, a wobble-plate type axial piston pump platform was selected due to advantage of mass independence in the rotating kit. This rotating kit can be operated at 61.1 cm³/rev (fixed 12° wobble angle) or 94.6 cm³/rev (18°). The baseline pump is rated at 2,400 rpm in open-circuit operation, where the self-priming speed limit was confirmed by measurements as the primary limit.

To isolate the tribological speed limit, the inlet was pressurized and a temperature and particle monitoring instrumentation was implemented. Temperatures were monitored at bearings, piston–cylinder gap, and slipper leakage locations. Together with online particle monitoring these metrics were used as an early warning indicator. With this methodology, stable overspeed operation up to 4,000 rpm was achieved at various pressures. At 3,800 rpm and 300 bar, a rapid thermal rise at the slipper interface and increased particle activity indicated imminent failure, and the test was terminated intentionally before catastrophic damage occurred. Post-test inspection confirmed that the pump remained intact, enabling failure mode analysis without obscured failure chains.

The paper presents the test methodology, the sensor-based stop criteria, and a simulation-guided redesign workflow for high-speed operation. Improvements of the slipper design are demonstrated by simulation and measurement, followed by a thermally adaptive, meso-structured slipper concept targeted at 10,000 rpm. In parallel, the requirements for a next-generation high-speed test rig are derived to enable testing at approximately 360 kW (61.1 cm³/rev, 10,000 rpm, 350 bar), which would upgrade this mobile pump to a power density of 17 kW/kg, outperforming all existing pumps on the market.

Keywords. Digital Twin, Thermal Modeling, Tribology, Efficiency.

1. INTRODUCTION

The electrification of mobile working machines is past its exploration stage, and increasing requirements are being imposed on hydrostatic powertrains. Electric motors have proven themselves as superior prime movers to diesel engines in terms of efficient, robustness and controllability. The technological progress both in battery and electric machine design has far from peaked, which has been primarily driven by high-volume automotive production. Power densities above 5 kW/kg have been reported for current traction motor technologies, with examples such as BYD or Tesla stating up to 5.5 kW/kg in mass production vehicles [1], [2]. In motorsport-oriented and aerospace-oriented machines, substantially higher peak power densities have been published; Lucid reported 11 kW/kg for an electric drive unit [3], and Evolito reported 18.5 kW/kg for a 240 kW peak motor at 13 kg. In parallel, prototype axial-flux machines have been demonstrated with short-term peak values far above today's series-production levels (e.g., YASA reporting up to 59 kW/kg peak in a prototype [4]). These values are not continuous ratings, but the trend indicates that further increases in electric motor power density are expected rather than stagnation.

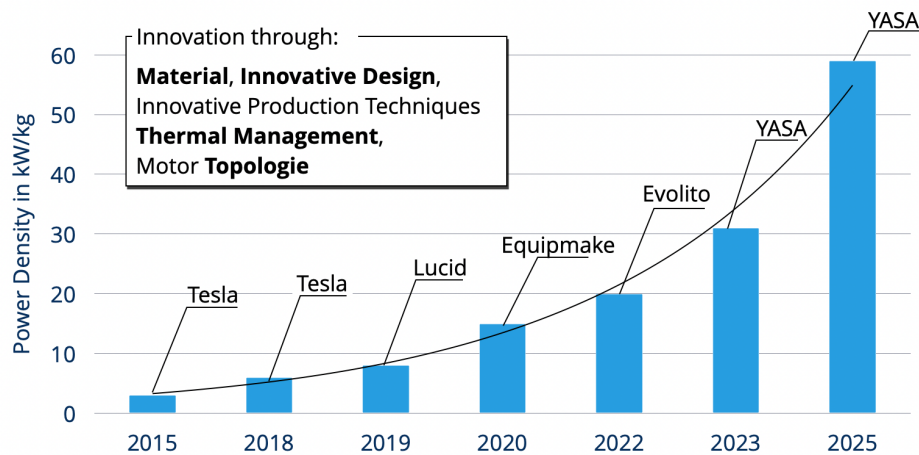


Figure 1.1. Power density trend in modern electrical motors

The main drivers for this development are mass and packaging constraints in mobile machinery and, even more strongly, in aviation. Because battery systems are heavier and larger than fuel tanks for comparable usable energy, weight and volume savings in all other subsystems are being prioritized. Across the reported high power density electric machines, one shared enabler is high rotational speed. In series-production vehicles, high efficiency regions are typically located at high machine speeds (e.g., 12,000–16,000 rpm reported for a Tesla Model 3 power unit efficiency map), and similar or higher speed levels are commonly used in modern traction and propulsion concepts. Similar high-speed motors are currently being designed for mobile construction machines [5], [6].

On the hydraulic side, several levers are available to increase power density, but progress has been limited in widely deployed products. In axial piston machines further increasing the displacement angle has been used successfully, but geometric limits and secondary

effects (longer piston stroke, increased sliding speeds, and stability constraints) restrict further gains. Higher operating pressure can raise power density, and high-pressure pumps exist; for example, Danfoss specifies closed-circuit pump pressure capability in the ~400 to 500+ bar class [7], while specialist solutions for significantly higher pressure levels (1000bar+) are available [8], [9]. However, system-level implementation at very high pressure is constrained by fittings, hoses, manifolds, and long-life requirements, particularly for medium to high flow rates.

Consequently, increasing rotational speed is considered a direct and scalable pathway to higher hydraulic power density, and it aligns with the E-motors provide. The main challenges of higher speed are concentrated inside the pump or motor itself, while the rest of the hydraulic system primarily experiences increased flow, which is generally easier to scale than pressure.

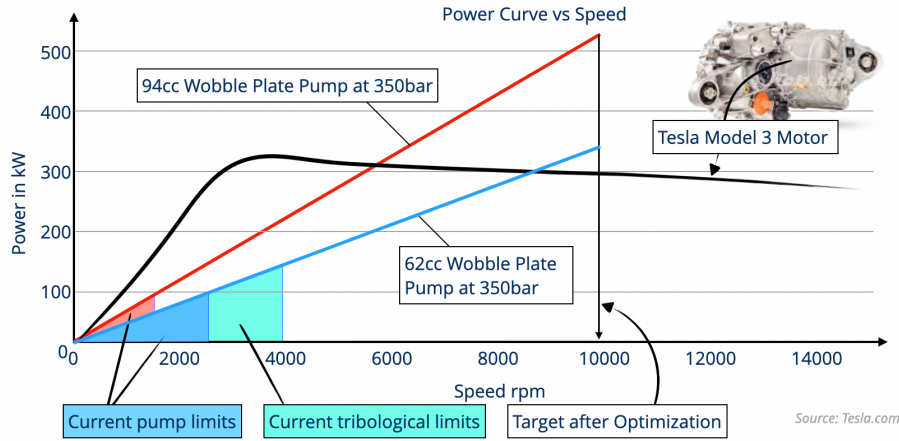


Figure 1.2. Power curves of current and targeted operation limits of a 62cc and 94cc pump, compared with a Tesla Model 3 motor [1].

In this work, the challenges associated with high-speed operation are addressed with emphasis on the tribological speed limit of fluid film interfaces for a 62cc wobble plate type axial piston pump. Methods for identifying this limit experimentally, supported by simulation-based redesign of critical components for maximum speed up to 10000 rpm, are presented along with first validations.

2. PRIOR WORK AND RESEARCH GOAL

The maximum rotational speed of hydrostatic machines can be grouped into three primary limitation classes:

1. **Fluid-dynamic limits** – e.g., self-priming capability and cavitation during the suction stroke.
2. **Body-force limits** – e.g., inertial and centrifugal forces acting on the rotating group, including cylinder-block tilt and piston stick phenomenon.

3. **Tribological limits** – e.g., fluid-film stability, hydrodynamic lift-off, and loss of contact control in lubricated interfaces.

Most hydrostatic pumps in mobile machinery are open-circuit pumps, where the inlet is supplied directly from a tank. In this configuration, the inlet pressure is usually only slightly above atmospheric and is strongly influenced by reservoir head and inlet line design. During the suction stroke, the pressure at pump inlet is reduced relative to the tank due to line losses (bends, restrictions) and viscous losses in the pipes and hoses. Because pressure and velocity are coupled, increasing flow velocity increases the local pressure drop for a given geometry, which may reduce the local pressure at the kidney region below the cavitation pressure. This mechanism has been discussed in the literature [10], [11], and it explains the characteristic volumetric efficiency drop above a certain speed in self-priming operation.

2.1. Fluid-dynamic speed limit: cylinder-block filling and cavitation

Manring expressed the limiting speed for incomplete filling using a Bernoulli-based formulation (Eq. 22 in [10]) as:

$$\omega^2 < \frac{2(P_i - P_{cav})}{\rho r_p^2 \left(1 + \left[\left(\frac{A_p}{A_k} \right)^2 - \left(\frac{N A_p}{\pi A_i} \right)^2 \right] \tan^2(\beta) \right)} \quad \text{Eq. 1}$$

This expression highlights that the filling limit is dominated by fluid density ρ , piston pitch radius r_p , piston area A_p , kidney area A_k , intake area A_i , number of pistons N , displacement angle β , and inlet pressure P_i . For open-circuit pumps, the most practical method to increase the limit is typically an increase of inlet pressure via charge pumps, often implemented using impellers.

In addition to inlet pressurization, mitigation approaches reported in the literature include increasing kidney areas A_k [11], reducing displacement angle, and reducing pitch radius r_p (e.g. by inclining the pistons), because these measures reduce characteristic velocities and thus reduce the associated pressure drop.

2.2. Body-force speed limit: cylinder-block tipping and rotating-group inertia effects

The body-force limitation for axial piston pumps is mainly governed by mass properties and pitch radius. The cylinder-block tipping mechanism is driven by inertial forces of the rotating and reciprocating components, producing a tipping couple that must be counteracted by the cylinder-block spring and pressure-induced clamping forces. Manring derived the corresponding limiting speed (Eq. 5 in [10]):

$$\omega^2 < \frac{2F_{Sp}R_{Block}}{N(M_p + M_s)r_p^2 \tan(\beta)} \quad \text{Eq. 2}$$

The relationship shows that higher speed capability is obtained by increasing spring force F_{Sp} or by reducing piston/slipper mass (M_p, M_s), pitch radius, and displacement angle. The detailed physical contributors to block tipping and the design criteria to prevent tipping were discussed in detail by [12], [13], including the interaction of pressure forces and inertial forces. Zhang has studied impact of piston mass on the maximum speed [14], while Achten

has proposed a counter force using a lever [13], however this type of solution is only suitable for the floating cup principle.

It should be noted that block tipping is not the only body-force-dependent limitation. At high speed, piston and slipper tilting tendencies increase, which may promote local metal-to-metal contact. In the slipper interface this typically manifests as accelerated wear, while at the piston–bore interface it may result in sticking or seizure, potentially leading to catastrophic failure called “piston stick”. These failure modes are also driven by inertial loading acting on the rotating group.

2.3. Tribological speed limit: slipper lift-off, film stability, and separation risk

At high speed, the tribological operating point may be destabilized by changes in hydrodynamic lift, local heating, and altered micro-motion of the slipper and piston. Excessive slipper lift-off may increase leakage or cause adverse force states at the ball joint. If a hold-down mechanism is present, opposing forces may be generated across the joint, potentially increasing play or causing partial separation. For the cylinder block / valve plate interface, increased hydrodynamics can lead to block lift, which results in a catastrophic failure due to instabilities in the displacement chamber.

Manring summarized the slipper tipping tendency in a limiting expression (Eq. 30 in Manring 2014), where the combined effects of tilting moments, spring force distribution, and pitch radius appear explicitly:

$$\omega^2 < \frac{F_{SP}}{Nr_p \left(M_s \frac{X_s}{R_s} + (M_p + M_s) \tan(\beta) \right)} \quad \text{Eq. 3}$$

According to Eq. 3 the tribological speed can hence be improved by increasing the spring force acting on the slipper F_{SP} , or decreasing the pitch radius r_p , number of pistons N , swash plate angle β , or the masses of the components. To counteract block lift at high speeds, it the main counter measure is reducing the pitch radius or increasing the spring force acting on the block. Zhang summarized in detail some measures that were taken in the past to improve tribological stability, but no single approach can be mentioned that would significantly improve this speed limit, other than again reducing mass [15].

2.4. Wobble plate type pump principle

Instead of increasing speed capacity primarily by mass reduction, an alternative approach is pursued in this work: the mass dependence is reduced structurally by changing the kinematic principle of the axial piston unit. In a wobble-plate arrangement, the inclined plate rotates with the shaft while the pistons and slippers undergo primarily translational motion, and the cylinder block remains stationary. As a consequence, the sealing gap between cylinder block and valve plate is eliminated and replaced by a check-valve commutation to the low-pressure and high-pressure ports.

Figure 2.3 illustrates the working principle of the check-valve-based wobble-plate pump used in this work. The investigated unit has a nominal displacement of 60 cm³/rev and uses a 12° inclined plane with up to nine pistons that are spring-loaded to maintain slipper contact with the wobble plate. The shaft is supported by two radial bearings and one axial bearing. In this design the wobble plate is the only rotating part, and the piston and slipper motion is

purely translatory. There is no valve plate, reducing the number of sealing gaps to two, and removing the need for speed depended grooves designs.

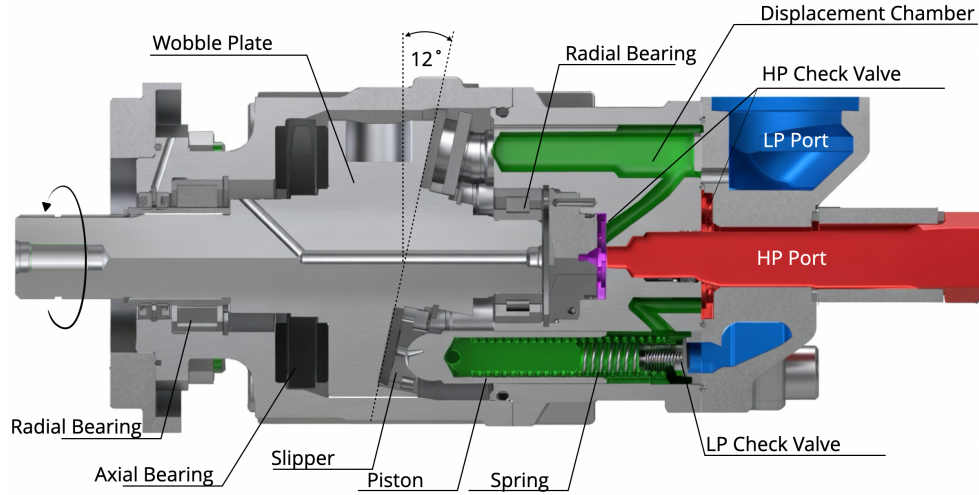


Figure 2.3. Working principle of check valve based wobble plate pump

Each displacement chamber is hydraulically isolated and connected to high and low pressure via dedicated check valves. This concept offers two key functional advantages:

1. **Pressure equalization during commutation** is governed by the check valves and therefore is less sensitive to geometric port timing, which reduces the tendency for pressure spikes at high speed.
2. **Modular piston count** is enabled because each bore is isolated; the unit may be operated with reduced piston count (e.g., six pistons) to reduce displacement while maintaining the same kinematic hardware.

The typical disadvantages of wobble plate pumps are fixed displacements and the inability to run in motoring mode. However, modern fast-switching-valves could overcome these limitations, as has been shown in digital displacement pumps [7]. With the dawn of speed variable displacement control, there is a new market for highly compact fixed displacement pumps.

The pump was tested in the configuration where all pistons delivered into a single high-pressure port.

2.5. Modified speed-limit expressions for the wobble-plate principle

By removing cylinder-block rotation, the tangential velocity component at the kidney is removed from the Bernoulli-based filling limit. Under this assumption, Eq. 1 simplifies to:

$$\omega^2 < \frac{2(P_i - P_{cav})}{\rho r^2 \tan^2(\beta) \left(\left[\left(\frac{A_p}{A_k} \right)^2 - \left(\frac{NA_p}{\pi A_i} \right)^2 \right] \right)} \quad \text{Eq.4}$$

Where the denominator is reduced when compared to Eq.1. For the same geometry and inlet pressure, this modification increases the theoretical filling speed limit because the tangential component of the kidney velocity is not present. In addition, the kidney opening A_k can be selected larger because it is no longer required to contribute to cylinder-block clamping as in a conventional valve-plate design. Combined, these effects indicate that the wobble-plate architecture should achieve higher self-priming speeds than a conventional rotating-block axial piston pump.

Because the cylinder block is stationary, the centrifugal tipping mechanism described by Eq. 2 is not present, and the associated speed limitation disappears for this architecture. The wobble plate itself remains a rotating body, and it can be easily designed to be rotationally balanced.

For the slipper lift-off / hold-down limitation, the centrifugal contributions are removed under the same assumption, leaving a simplified form where the primary limiting term is the slipper hold-down spring force:

$$\omega^2 < F_{SP} \quad \text{Eq.5}$$

2.6. Surface structures as a complementary path for conventional machines

Although the wobble-plate principle enables higher speed capability in theory, most industrial axial piston units are not of wobble-plate type. Therefore, surface structures on pistons and slippers are considered as an additional pathway to extend tribological stability for traditional architectures.

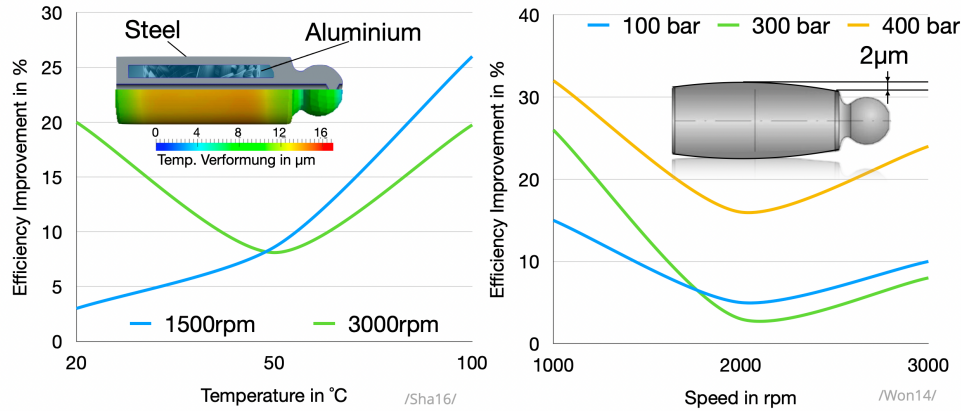


Figure 2.4. Piston surface structures improve efficiency also at high speeds

Figure 2.4 summarizes published examples where piston surface structures improved efficiency under high pressure and high speed conditions. [16], [17] reported that defined piston surface shaping can increase efficiency at elevated pressure and speed, with improvements exceeding 20% depending on operating point (right graph in Figure 2.4). The effect was attributed to improved piston micro-motion and the ability to reduce clearance while still permitting necessary piston tilt. Such surface shapes are typically only a few micrometers in height and therefore require precise manufacturing.

[18] proposed a thermally adaptive piston concept in which a hollow piston is filled with a material of higher thermal expansion (e.g., aluminum), producing a temperature-dependent convex shape. Efficiency improvements near 20% at high speed (left graph in Figure 2.4) were reported at elevated temperature, which is relevant because critical speed conditions are typically accompanied by increased local heating.

Slipper surface structures have primarily been reported for low-speed robustness and torque reduction, as [19] published a measured 80% reduction in torque. However, the same stabilization of the fluid film is expected to be beneficial at high speed. [20] showcased simulated a reduction of 60% of the slipper power losses at the corner power of the pump.

Combining surface structures on both piston and slipper with the wobble plate principle should lead to even further speed gains, as will be studied in this research.

2.7. Research goal

The objective of the present research is to establish the tribological maximum speed of the 60 cm³/rev wobble-plate pump platform and to demonstrate design and test methods to increase this limit. In the longer term, surface structures on both piston and slipper are intended to be combined with the wobble-plate architecture to extend the tribological speed capability toward 10,000 rpm, exceeding typical pre-charged axial piston pump limits by more than 100% at comparable displacement (see Figure 2.5). The present paper reports the first step in this endeavor by (i) establishing a methodology to identify the tribological speed limit and (ii) demonstrating initial improvement measures to the slipper.

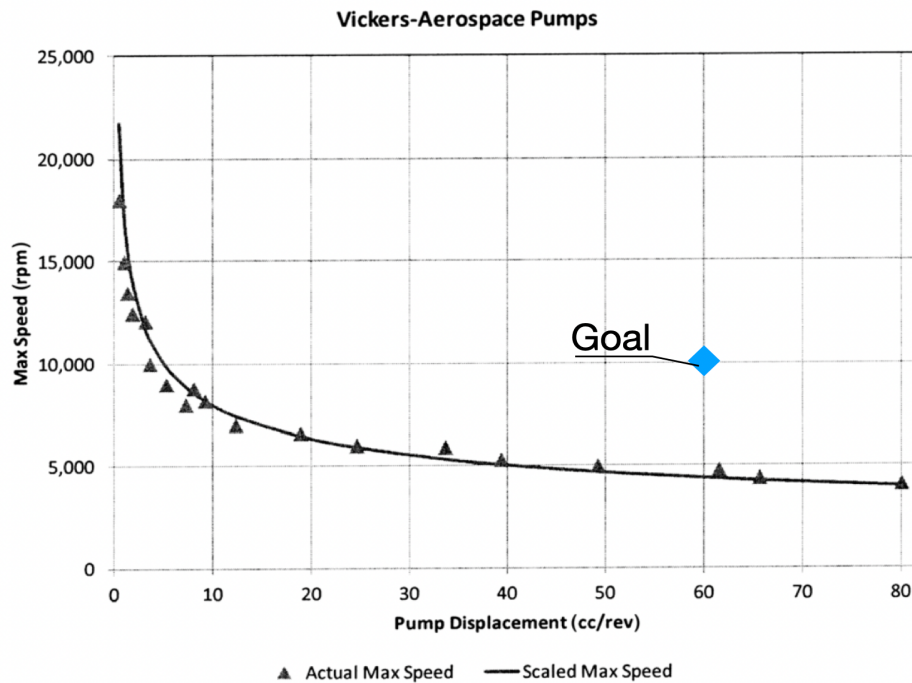


Figure 2.5. Typical Speed limits of pre-charged axial piston pumps [10].

3. MEASUREMENTS

This chapter presents the measurement setup, highlights key measurement results, and establishes the current tribological speed limit of the unmodified wobble-plate pump. Measurement results for the improved slipper design are reported in Chapter 4.

3.1. *Measurement Set-Up*

To define a baseline for subsequent design improvements, the tribological maximum speed of the unmodified wobble-plate pump was experimentally determined. The investigated unit was a Sunfab SLPD 40/20 wobble-plate pump with a total displacement of 61.1 cm³/rev. The manufacturer-rated maximum speed is 2200 rpm, and the maximum working pressure is 350 bar. The wobble-plate angle was chosen to be fixed 12° for the presented tests. A maximum angle of 18.2° is available on the same platform, which increases the displacement to 94.6 cm³/rev while maintaining the same rotating kit.

The pump mass is 22 kg, resulting in a power density of 3.6 kW/kg at 12° and 5.5 kW/kg at 18.2°. These values exceed typical swash-plate axial piston units of comparable displacement, and they indicate that the pump platform is well suited for high power density investigations.

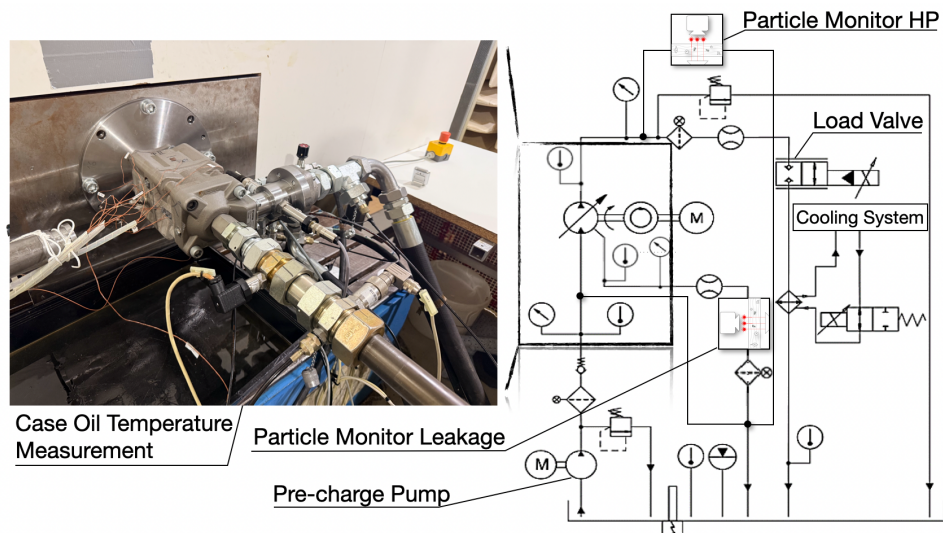


Figure 3.6. Measurement Set-up of the 61.1 cc wobble plate pump with thermal and particle monitoring.

Figure 3.6 shows the overall test rig configuration including photo and hydraulic schematic. A combined thermal and particle monitoring approach was selected to enable a controlled approach to the tribological limit, where imminent damage could be detected before catastrophic failure occurred. The test rig includes a pre-charge pump capable of up to 20 bar boost pressure, a load valve for pressure control, and a 160 kW cooling system. The cooling system is capable of controlling the inlet temperature within ± 0.1 °C. Unless stated otherwise, all measurements were recorded at thermal steady state with an inlet temperature of 40 °C.

Particles were monitored in both the high-pressure line and the leakage line to provide early warning of increasing wear activity. Temperature was measured at the ports (high-pressure, low-pressure, leakage) and at multiple internal locations inside the pump, as shown in **Figure 3.7**. Thermocouples were installed near both radial bearings and the axial bearing, at multiple internal case oil locations, and in the piston/bushing region.

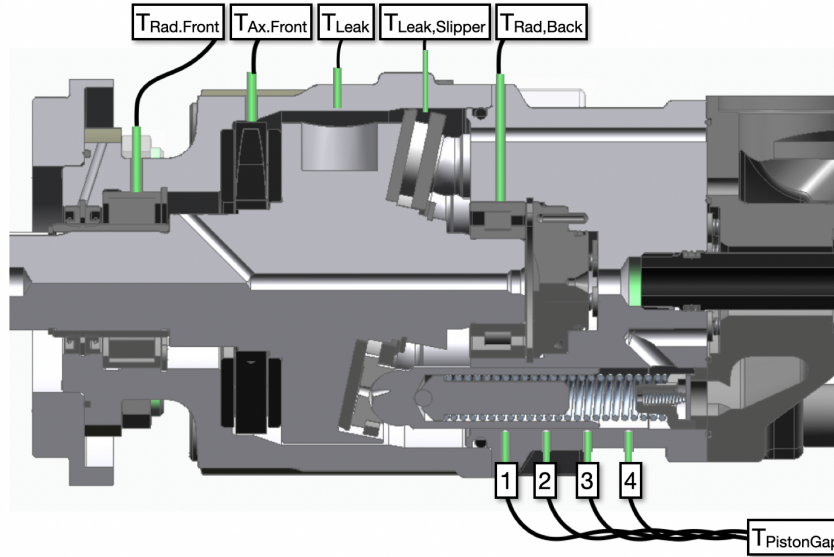


Figure 3.7. Thermal monitoring of the wobble plate pump at key areas.

A key advantage of the wobble-plate architecture is that the cylinder block is stationary, which simplifies the instrumentation of the piston–bushing gap. Four temperature probes were placed along the piston/bushing gap as illustrated in Figure 3.7, enabling a local temperature gradient to be captured from the displacement chamber region toward the outer bore section.

3.2. *Measurement of the tribological speed limit*

All operating points were considered to be at thermal steady state when the rate of change of all monitored temperatures was below 0.1 K/min. An example steady-state operating point is shown in **Figure 3.8**. The pump was operated at 1500 rpm and 300 bar, with an inlet temperature of 40.6 °C and an outlet temperature of 42.4 °C. The leakage temperature measured at the external leakage port was 67.3 °C. However, the internal measurements showed substantially higher local temperatures: Leakage temperatures inside the housing approached 120 °C near the slippers and approximately 108 °C near the wobble plate. The axial bearing exhibited the highest temperature, exceeding 125 °C. The piston gap temperatures ranged from approximately 51 °C to 91 °C, with lower values toward the displacement chamber side.

This operating point illustrates two critical findings:

1. **The axial bearing and slipper region represent the dominant thermal hot spots** in the investigated pump configuration.
2. **External leakage temperature measurements do not represent internal gap temperatures**, even when sensors are placed close to the pump (5 cm in this case). This deviation becomes particularly pronounced at low leakage flow. For the shown operating point, the leakage flow was 0.6 l/min, such that the external leakage line did not provide sufficient thermal coupling to reflect internal film temperatures.

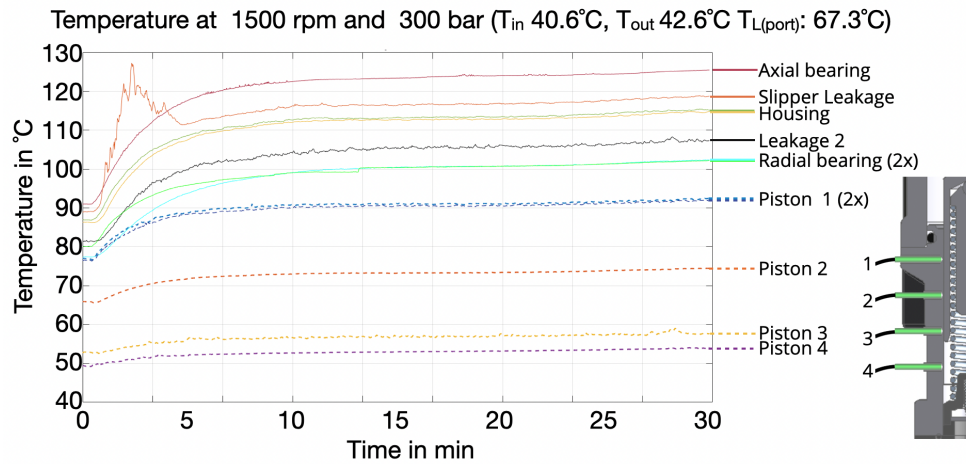


Figure 3.8. Internal temperatures at 1500 rpm and 300 bar.

The presence of steady-state internal gap temperatures above 100 °C at 1500 rpm indicates a limited thermal margin already at moderate speed. This condition differs from typical swash-plate axial piston pumps, where internal leakage flow and oil distribution often provide higher convective cooling of loaded interfaces.

The unmodified pump was tested up to 4000 rpm, exceeding the rated speed by approximately 80%. The tribological speed limit for the unmodified configuration was empirically identified at **300 bar and 3700 rpm**. At this operating point, the slipper-related temperature continued to rise beyond 150 °C without stabilizing, and a sustained increase in particle counts ($\geq 6 \mu\text{m}$) was detected in both leakage and high-pressure monitoring. The test was terminated based on these stop criteria to preserve component integrity for failure mode evaluation.

After disassembly, all parts remained functional; however, pronounced wear marks were observed on the slippers and on the wobble plate running surface, as shown in **Figure 3.9**. The pistons showed wear, but it remained within acceptable limits. All bearings were still operational after the approximately 40 h test campaign. However, the axial bearing exhibited the beginning of pitting.

In addition to the qualitative wear images, topographical measurements of the slipper profile were performed at for two slippers in three conditions: (i) new condition, (ii) after 16 h of

operation (up to approximately 2000 rpm), and the second slipper (iii) after 40 h at the end of the full overspeed campaign. The profile comparison in Figure 3.9 shows a continuous and substantial loss of material across a large contact region. This trend indicates that the original slipper was not designed for continuous operation under the combined conditions of high speed and high pressure, and it supports the temperature and particle signals that defined the tribological limit.

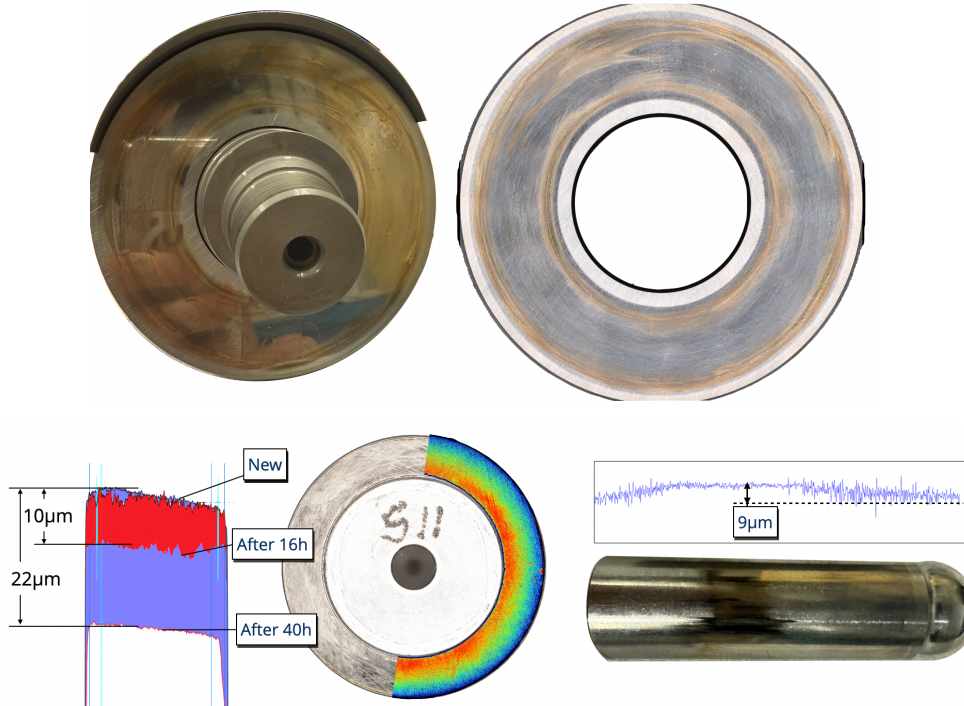


Figure 3.9. Worn components after the test campaign, including topological measurements.

Measured volumetric efficiency and hydromechanical efficiency at three pressure levels are shown in Figure 3.10. The rated maximum speed (2200 rpm) is marked by the dashed line. Volumetric efficiency remained high up to the rated speed and decreased beyond that point. Above 2500 rpm, boost pressure was applied and increased gradually, reaching 4.5 bar inlet pressure at 3700 rpm at 300 bar Δp . With boost pressure applied, volumetric efficiency remained above 90% for most operating points. Without boost pressure, volumetric efficiency dropped below 90% at 200 bar shortly after approximately 2600 rpm. The remaining drop at high speed despite boost pressure is likely caused by limitations in inlet check valve opening dynamics and available effective inlet flow area.

Hydromechanical efficiency is the more relevant metric for identifying tribological limitations because it represents the fraction of mechanical power converted into hydraulic power and therefore captures frictional losses. In the present measurements, the lowest hydromechanical efficiencies were observed at 100 bar, where efficiency decreased approximately linearly with speed. At 300 bar, hydromechanical efficiency remained above

90% over a large portion of the operating map. Importantly, the tribological limit at 300 bar and 3700 rpm could not be identified from efficiency trends alone. The limit was detected only by local temperature escalation and continuous particle generation.

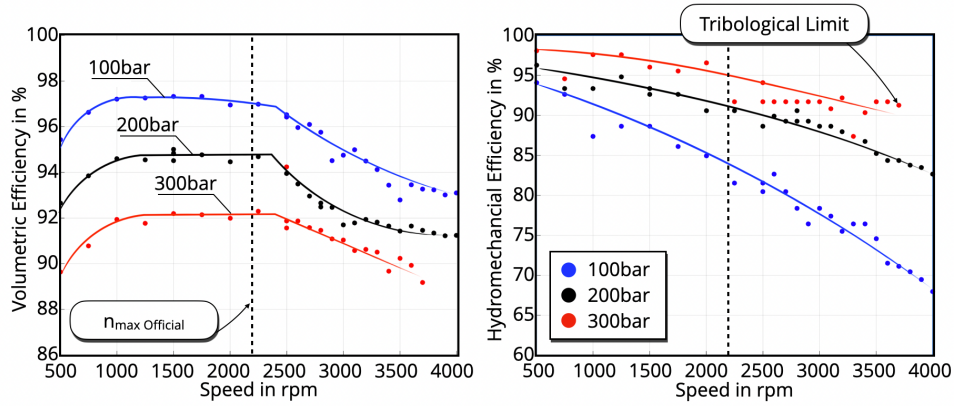


Figure 3.10. Volumetric and hydromechanical efficiency for three pressure levels

Figure 3.11 summarizes the temperature fields over speed and pressure for the slipper leakage, piston gap, and axial bearing. Across the investigated operating regime, the axial bearing consistently exhibited the highest temperatures. The dashed line indicates the region where the temperature is above 100°C. An improved design should stay below this temperature to ensure a long component and oil life-time. The bearing is a roller-type design specific to this pump platform. Due to the low leakage flow of the pump, insufficient oil flow appears to be available for convective cooling of this bearing at elevated pressure and speed. The resulting overheating increases the global case oil temperature and raises the temperature level in other lubricated interfaces.

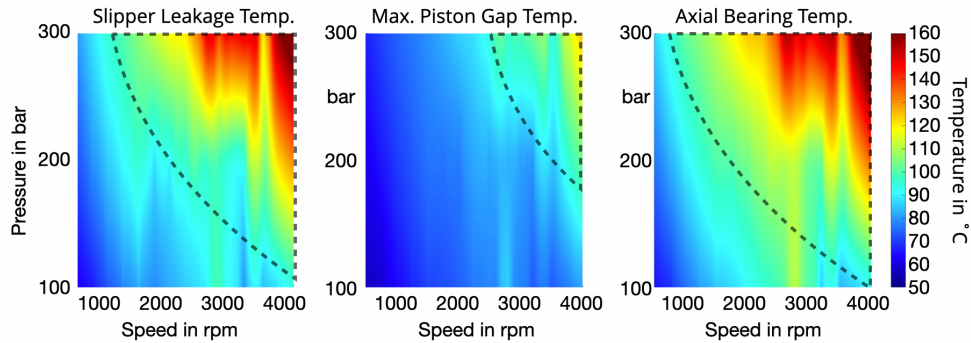


Figure 3.11. Captured slipper, piston and axial bearing temperature until 4000 rpm

The leakage temperature measured in the vicinity of the slipper gap increased strongly with speed and pressure, reaching approximately 150 °C at 3700 rpm and 300 bar. The maximum piston gap temperature (measured at the top bore position P1) exceeded 100 °C at high speed. This increase is attributed primarily to elevated case temperatures and thermal conduction through the stationary cylinder block. In contrast, at 3700 rpm and 300 bar the piston gap temperature near the displacement chamber (P4) remained comparatively low at

58.9 °C, while the external leakage port temperature was only 50.6 °C at a leakage flow of 0.7 l/min.

Overall, the pump survived the overspeed campaign, and a baseline tribological speed limit of 3700 rpm was established for the unmodified configuration. The central outcome of the baseline measurements is that **(i) the axial bearing is thermally under-supplied with oil**, which elevates the entire thermal level of the unit, and **(ii) the slipper interface operates at high temperature and exhibits continuous wear** (see Figure 3.9). To establish a more generally applicable tribological speed limit for wobble-plate type pumps, both improved axial-bearing cooling and improved slipper performance are required.

4. THERMAL IMPROVEMENT OF SLIPPER

The pump was modelled using Caspar FSTI, a numerical simulation environment used to predict power losses and operating conditions in the lubricating gaps of axial piston machines. The tool was developed specifically for piston pump tribology, enabling efficient gap-level simulation without requiring extensive user effort in numerical discretization, meshing strategies, or sensitivity studies. The relevant micro-motion of components and elastic deformation due to both pressure and temperature are captured, which has been shown to strongly influence the operating conditions in tribological interfaces [21]. The tool was validated for the individual lubricating gaps by comparison with measured temperatures and gap heights, as reported in [22], [23], [24], [25], [26], [27], [28], [29].

Using this validated toolchain, both the piston–bushing gap and the slipper–swashplate gap were predicted to approach wear-critical conditions at elevated pressure and at speeds exceeding 3000 rpm. However, a critical thermal behavior was predicted for the slipper already at approximately 1500 rpm. In this regime, the slipper leakage was predicted to be too low, causing the oil to remain trapped in the film region. As a result, heat generated by viscous shear was not removed sufficiently by convective transport, leading to a rapid increase of local film temperature. The temperature rise was predicted to deform the swashplate, thereby worsening the gap geometry and accelerating wear progression.

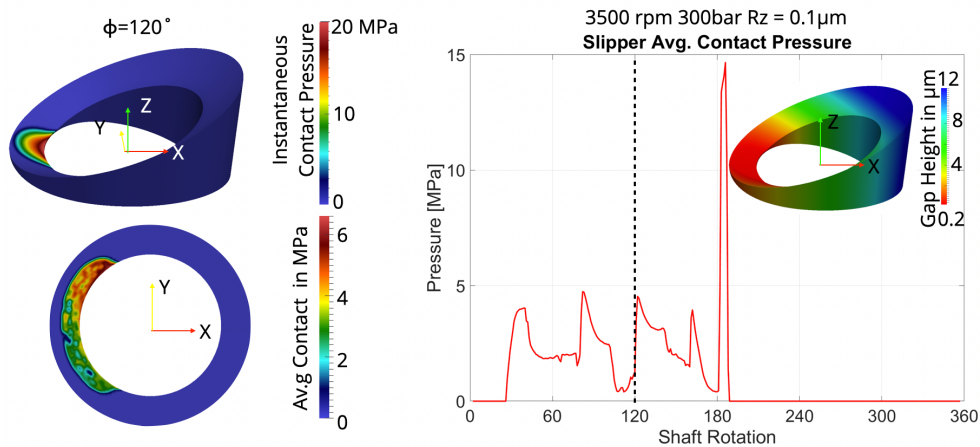


Figure 4.12. Predicted slipper contact at high speed and high pressure

Figure 4.12 shows the predicted contact behavior of the slipper film at 3500 rpm and 300 bar. The instantaneous contact pressure distribution is shown at a rotational angle of $\phi = 120^\circ$, corresponding to the middle of the high-pressure stroke. Local contact pressures up to 20 MPa were predicted during this instance. The right-hand plot shows the averaged contact pressure per rotational degree, obtained by averaging the local contact pressure across all computational cells of the gap and evaluating this average for each shaft angle. In addition, the top-right inset shows the predicted film thickness distribution at $\phi = 120^\circ$, where the positive X-direction corresponds to the direction of motion. The film is predicted to form a positive wedge, building hydrodynamic pressure toward the trailing end. At this low gap height this inclination causes contact at local regions of the slipper.

The accumulated contact pressure over one shaft revolution is shown in the bottom-left of Figure 4.12, where the average instantaneous contact pressure is integrated (or accumulated) across the revolution and visualized as the spatial distribution of cumulative contact exposure. This representation indicates that contact initiates at the inner edge of the sealing land and then progresses outward. This predicted wear initiation location is consistent with the measured wear pattern presented in Figure 3.9, where the inner sealing-land edge exhibits the largest material loss.

To reduce slipper wear and to decrease excessive leakage oil temperatures, the slipper dimensions were varied using the validated simulation model. The study was conducted across four operating conditions (OC1–OC4), shown in **Figure 4.13**, spanning the regular operating regime of the pump. The purpose of this first slipper iteration was to shift the thermal behavior toward a more typical piston-pump baseline and to evaluate whether a conventional improvement of hydrostatic balance can increase speed capability before surface structures are introduced.

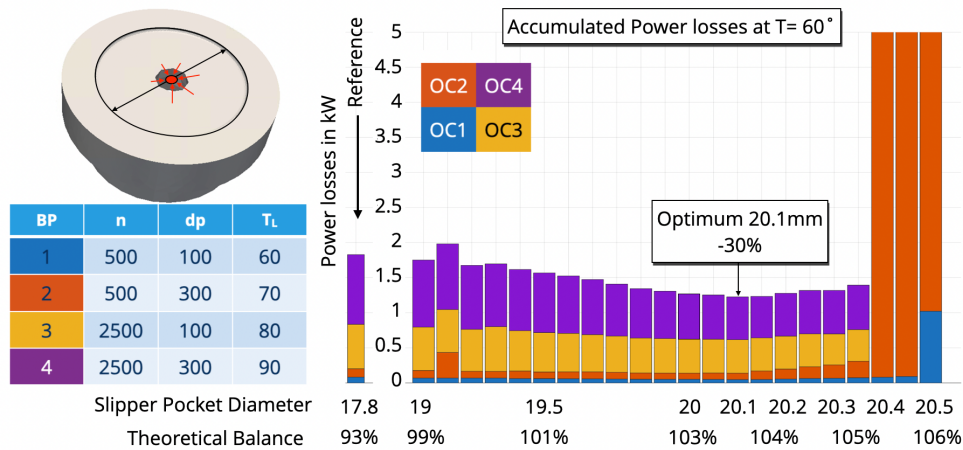


Figure 4.13. Slipper pocket diameter study vs accumulated power losses across four operating conditions.

When varying the inner pocket diameter of a single-land slipper (i.e., a slipper with one active sealing surface), the hydrostatic balance is altered. The balance is defined as the hydrostatic force generated in the pocket and across the sealing land divided by the piston force acting on the slipper. The ratio is normally designed to be lower than unity because

hydrodynamic pressure generation adds to the hydrostatic support. A balance higher than 1 would imply more lifting force than clamping force, which may cause excessive lift-off and loss of sealing ability. However, these simplified balance estimates rely on strong assumptions, in particular that the slipper remains parallel to the running surface and that elastic deformation due to pressure and temperature does not alter the film geometry. In practice, slipper motion is dynamic and the film is almost never parallel. Furthermore, pressure and thermal loads deform the components on the micrometer scale, which is comparable to the film thickness itself. Consequently, balance estimates are not fully reliable as absolute design rules, but they remain useful as comparative metrics between designs and platforms. The reference slipper has an inner pocket diameter $\text{dinG} = 17.8$ mm, corresponding to approximately 93% balance. This is within typical industrial design practice, where balances between 90–95% are common. As shown in Figure 4.13, an increase of pocket size to a balance well above 100% was predicted to be beneficial for the investigated wobble-plate pump, with the minimum accumulated power loss occurring at a pocket diameter of 20.1 mm ($\approx 103.8\%$ balance). At this point, the simulation predicts approximately 30% lower slipper losses compared to the reference.

From a traditional design perspective, a balance above 100% is typically avoided because high leakage would be expected. The simulation results show that this concern becomes dominant only beyond approximately 20.4 mm pocket diameter ($\approx 105.5\%$ balance), where leakage is predicted to increase sharply with pressure and the pump would effectively cease to operate as intended. This risk is the reason why conservative designs are often selected. In the present case, the simulation predicts that the $>100\%$ balance region can still be feasible because the slipper deforms under pressure and temperature, altering the sealing land shape and effectively reducing active hydrostatic area. This reduces the net hydrostatic lifting force and leaves margin for hydrodynamic effects without leading to catastrophic lift-off. It is therefore emphasized that this is **not** a universal recommendation to design above 100% balance. The central point is that without a power-loss simulation tool that captures elastic and thermal effects, it is difficult to determine the optimum dimensions reliably, and design “recipes” cannot be transferred directly from one pump to another because material, thickness, and thermal boundary conditions change the deformation and therefore the tribological outcome.

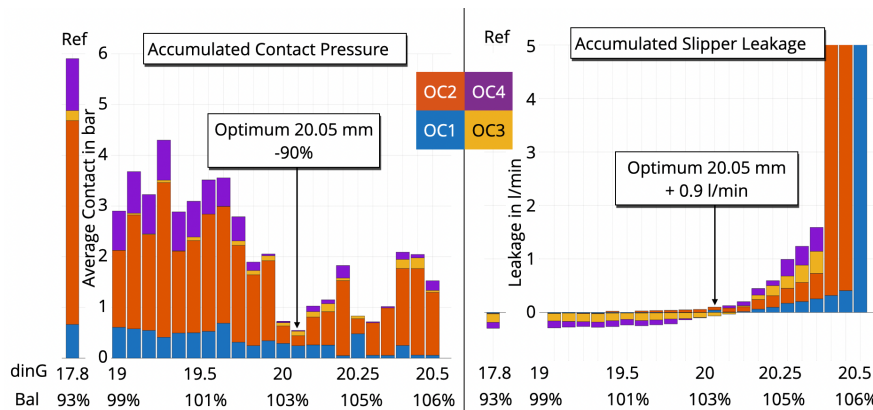


Figure 4.14. Accumulated average contact over 4 operating conditions (left) and accumulated leakage vs slipper pocket diameter (dinG) and balance.

As shown in **Figure 4.14**, increasing the inner pocket diameter for this wobble-plate slipper reduces accumulated contact pressure (particularly at OC2: 500 rpm / 300 bar) while keeping accumulated leakage nearly constant over most of the tested range. Only beyond approximately 20 mm does leakage start to increase noticeably. Some leakage is beneficial because it cools the film and reduces the risk of thermal runaway. In practice, the target is not minimum leakage, but minimum total power loss, which implies a compromise between frictional losses and leakage losses.

Based on the combined objective of minimal total power loss, minimal contact, and controlled leakage, an inner pocket diameter of **20.05 mm** was selected. While some low-speed contact was still predicted, it was substantially reduced relative to the reference design. The remaining contact was predicted to primarily smooth the relatively high initial surface roughness and then diminish as the running surfaces adapt.

The improved slipper design was predicted to reduce viscous shear losses in the slipper film and thereby reduce the thermal loading of the wobble plate. As shown in **Figure 4.15**, the peak swash/wobble plate temperature at corner power was predicted to decrease from 155°C to 82°C. This 73 K reduction of the local hot spot was predicted to reduce leakage temperature by approximately 40 K. It is noted that the axial bearing was not included in the simulation model; therefore, thermal boundary conditions had to be estimated, which may influence the absolute values. Nevertheless, the modified slipper is expected to improve the thermal behavior of the pump while only marginally increasing leakage.

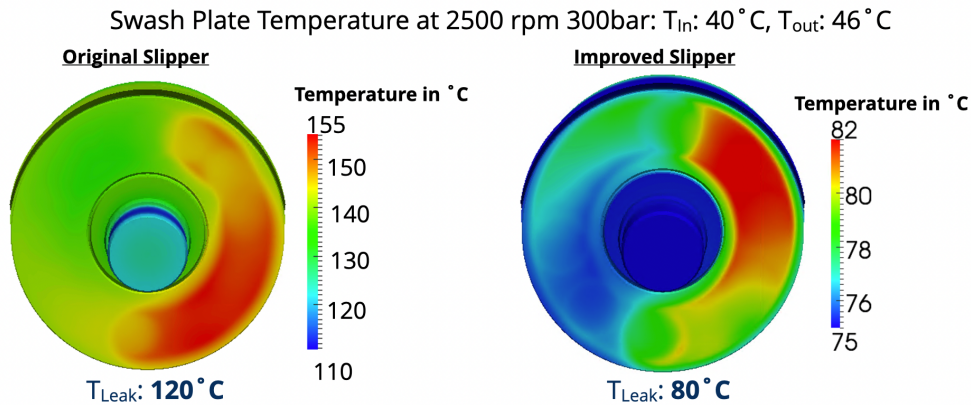


Figure 4.15. Simulated swash plate temperature at 2500 rpm and 300 bar for two different slipper designs: Left: reference slipper; Right: new slipper.

4.1. Measurement results of the improved slipper

The improved slippers were manufactured and implemented into a new rotating kit, and the test campaign was continued. Although measurements are still ongoing, most of the regular pump operating envelope has already been covered. The observed improvements are summarized in **Figure 4.16**. The leakage temperature near the slipper gap decreased substantially with the new slipper design, particularly at high pressure. At 300 bar and 2000 rpm, a temperature reduction of 28 K was measured. While the corner power condition has not yet been measured, the current trend indicates a temperature reduction exceeding 30 K at that operating point.

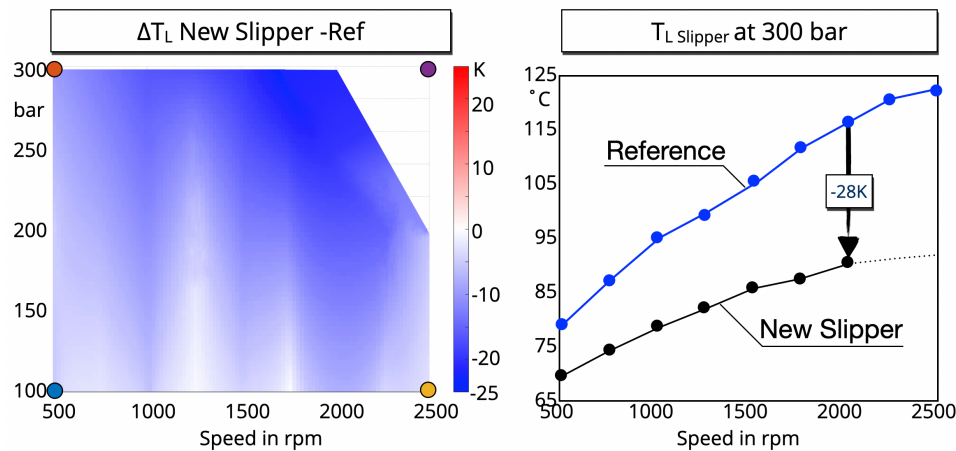


Figure 4.16. Reduced slipper leakage temperature (left) and slipper leakage temperature trend with speed at 300 bar (right)

The primary objective—reduction of slipper temperature—was achieved with the modified slipper. While the measurement campaign is still ongoing, the observed trend indicates that the design represents a successful improvement. The achieved temperature reduction at 2500 rpm and 300 bar is not expected to match the simulated magnitude exactly, but the trend is captured well. The reduction is attributed primarily to reduced viscous losses in the slipper film rather than a large increase in leakage.

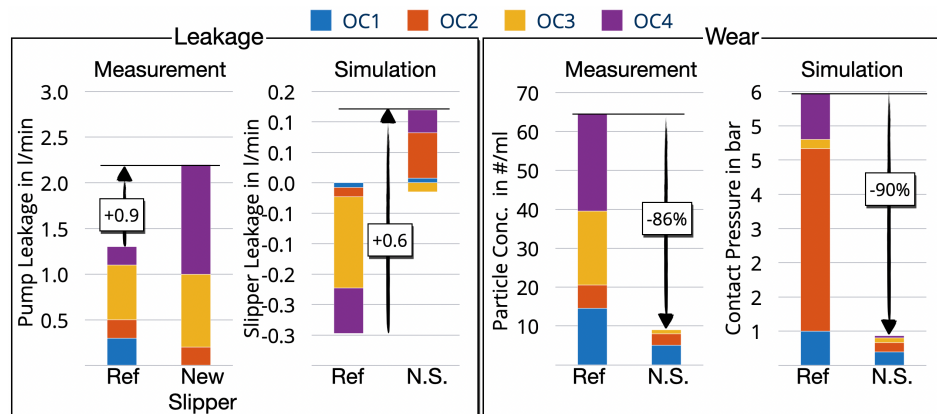


Figure 4.17. Predicted and measured accumulated leakage (left) and wear (right) for the reference and the new slipper (N.S.).

This is supported by the leakage evaluation shown in **Figure 4.17** (left), where accumulated pump leakage is compared across the four operating conditions. Although the OC4 leakage value is currently a prediction, the data indicate that the additional leakage is below 1 l/min when accumulated across all four operating conditions. The measured trend matches the simulation, although not in absolute magnitude. Such deviations are expected because measured total pump leakage includes contributions from multiple leakage paths, whereas the simulation focuses on an individual gap. Nevertheless, no significant increase in pump

leakage was observed, confirming that the selected ~104% balance did not introduce excessive leakage.

The right-hand side of Figure 4.17 shows that the predicted reduction in wear tendency was confirmed by measurement. The simulation predicted approximately 90% lower accumulated contact pressure across the four operating conditions, and the measured particle concentration of particles $\geq 6 \mu\text{m}$ decreased in a similar manner. This result may change slightly once OC4 is fully measured; however, the current trend does not indicate significant wear at corner power for the new slipper.

Overall, while the measurement campaign is still ongoing, it can be concluded that the new slipper design reduces operating temperature at high speed and high pressure and provides a more representative baseline for the next research step: the addition of surface structures on both slipper and piston and subsequent testing toward higher rotational speeds.

5. CONCLUSION

This paper reported the first step toward an ambitious goal: operating a ~60 cm³/rev hydrostatic unit at rotational speeds up to 10,000 rpm for electrified mobile machinery. This speed increase would push the power density of this already highly compact pump from 3.7 kW/kg to 16.2 kW/kg. Within this paper the pump was already operated at 6.4 kW/kg, a 60% increase. It was shown that the wobble-plate principle removes several dominant speed-limiting mechanisms of conventional swash-plate axial piston pumps. In particular, cylinder-block rotation is eliminated, such that centrifugal-force-driven effects and the valve-plate sealing interface no longer define the operating boundary. Although self-priming and inlet filling remain relevant, the tribological speed limit is not constrained by an inherent physical barrier in the same manner as in rotating-block architectures, and it can be shifted by targeted design measures.

A validated Caspar FSTI-based simulation workflow was used to predict gap behavior, thermal loading, and contact risk, and it was applied to redesign the slipper for high-speed/high-pressure operation. By increasing the hydrostatic balance of the slipper from 93% to approximately 104% (i.e., +11% absolute), the excessive heat generation of the slipper interface was reduced in simulation and subsequently confirmed experimentally.

Baseline testing of the unmodified pump demonstrated that, with inlet boosting, the operating speed could be increased substantially beyond the rated value. A tribological speed limit was empirically identified at 300 bar and 3700 rpm, triggered by continuously rising slipper-related temperatures and increased particle activity, while the hardware remained intact for post-test inspection. The measurements showed that internal gap temperatures were significantly higher than external leakage temperatures and that the axial bearing and slipper interface dominated the thermal loading. This finding established a clear improvement path: thermal management of the axial bearing and reduction of slipper viscous losses were identified as primary levers to extend speed capability.

The redesigned slipper was manufactured and tested, and a clear improvement was measured. The leakage temperature in the slipper region was reduced by more than 30 K at high pressure and speed, while the total pump leakage increased only marginally (below

1 l/min accumulated across the evaluated operating conditions). In parallel, particle generation was reduced by more than 80%, matching the predicted reduction of accumulated contact and indicating substantially improved tribological stability.

Overall, the presented methodology—instrumented overspeed testing with early-warning criteria combined with simulation-guided redesign—was shown to be effective for identifying and shifting the tribological speed limit of a wobble-plate pump platform. In the next step, thermally adaptive and surface-structured concepts will be engineered on both slipper and piston interfaces to further stabilize the fluid film at ultra-high speed. These designs will be validated on a dedicated high-speed test rig capable of testing operation up to 10,000 rpm.

6. NOMENCLATURE

Latin symbols

- A_i – inlet flow area of intake manifold / suction port [m^2]
- A_k – kidney area (effective port area per displacement chamber) [m^2]
- A_p – piston cross-sectional area [m^2]
- F_{SP} – assembled spring force (cylinder-block / hold-down spring force) [N]
- M_p – mass of one piston [kg]
- M_s – mass of one slipper [kg]
- N – number of pistons [-]
- N.S. – New slipper design
- P_i – inlet pressure at intake manifold / suction port [bar]
- P_{cav} – cavitation pressure (threshold pressure for vapor formation) [bar]
- R_{Block} – outer radius of the cylinder block [m]
- R_s – maximum lever arm of swash reaction on slipper [m]
- r_p – piston pitch radius [m]
- X_s – distance from piston–slipper ball joint to slipper mass center [m]

Greek symbols

- β – displacement angle (swash/wobble plate inclination angle) [$^\circ$]
- ω – angular speed of shaft [rad/s]
- ρ – fluid density [kg/m^3]
- ϕ – shaft rotational angle [$^\circ$]

7. REFERENCES

- [1] Tesla, “Battery Day,” 2020. [Online]. Available: [tesla.cpm](https://www.tesla.com/battery-day)
- [2] BYD, “BYD Super DM (Dual Mode) Technology.” [Online]. Available: <https://www.byd.com/en-az/super-dm-technology>
- [3] LucidMotors, “Lucid Unveils State-of-the-Art Motorsports Electric Drive Unit, Taking Lucid Air’s Advanced Electric Motor and Drivetrain Technology to the Racetrack.” [Online]. Available: <https://ir.lucidmotors.com/news->

releases/news-release-details/lucid-unveils-state-art-motorsports-electric-drive-unit-taking/

- [4] YASA, "YASA smashes own unofficial power density world record pushing state-of-the-art electric motor to staggering new 59kW/kg benchmark." [Online]. Available: <https://yasa.com/news/yasa-smashes-own-unofficial-power-density-world-record-pushing-state-of-the-art-electric-motor-to-staggering-new-59kw-kg-benchmark/>
- [5] J. Schröter, G. Jacobs, S. Zhitkova, M. Felden, and K. Hameyer, "Development of High Speed Electrical Drives for Mobile Machinery – Challenges and Potential Solutions," *9th International Fluid Power Conference (IFK)*, pp. 417–427, 2014.
- [6] Rexroth, "Rexroth eLION solution," HannoverMesse. [Online]. Available: https://www.hannovermesse.de/apollo/hannover_messe_2025/obs/Binary/A1408136/eLION_Product_Portfolio.pdf
- [7] A. S. Danfoss, "H1P 060 Data Sheet," no. June, pp. 1–2, 2025, [Online]. Available: <https://assets.danfoss.com/documents/latest/498340/AI152886482421en-000901.pdf>
- [8] O. Company, "PFBK-043 – Fixed Displacement Axial Piston Pumps." [Online]. Available: <https://www.oilgear.com/products/piston-pumps/pfbk/pfbk-043/>
- [9] Bieri, "Axial piston pumps AKP110." [Online]. Available: <https://www.bierihydraulics.com/en/products/pumps/axial-piston-pumps/axial-piston-pumps-akp110/>
- [10] N. D. Manring, V. S. Mehta, B. E. Nelson, K. J. Graf, and J. L. Kuehn, "Scaling the speed limitations for axial-piston swash-plate type hydrostatic machines," *Journal of Dynamic Systems, Measurement and Control, Transactions of the ASME*, vol. 136, no. 3, 2014, doi: 10.1115/1.4026129.
- [11] M. Kunkis and P. J. Weber, "Experimental and numerical assessment of an axial piston pumps's speed limit," in *FPMC2016*, 2017, pp. 1–9.
- [12] N. D. Manring, "Tipping the cylinder block of an axial-piston swash-plate type hydrostatic machine," *Journal of Dynamic Systems, Measurement and Control, Transactions of the ASME*, vol. 122, no. 1, pp. 216–221, 2000, doi: 10.1115/1.482445.
- [13] P. Achten and S. Eggenkamp, "Barrel tipping in axial piston pumps and motors," *Proceedings of 15:th Scandinavian International Conference on Fluid Power, 15th Scandinavian International Conference on Fluid Power, Fluid Power in the Digital Age, SICFP'17, June 7-9 2017 - Linköping, Sweden*, vol. 144, pp. 381–391, 2017, doi: 10.3384/ecp17144381.
- [14] J. Zhang *et al.*, "Effect of piston-slipper assembly mass difference on the cylinder block tilt in a high-speed electro-hydrostatic actuator pump of

- aircraft," *International Journal of Precision Engineering and Manufacturing*, vol. 18, no. 7, pp. 995–1003, 2017, doi: 10.1007/s12541-017-0117-1.
- [15] C. Zhang, C. Zhu, B. Meng, and S. Li, "Challenges and solutions for high-speed aviation piston pumps: A review," *Aerospace*, vol. 8, no. 12, pp. 1–25, 2021, doi: 10.3390/aerospace8120392.
 - [16] A. M. Wondergem, "Piston/Cylinder Interface of Axial Piston Machines - Effect of Piston-Micro Surface Shaping," 2014. doi: 1585442.
 - [17] A. M. Wondergem and M. Ivantysynova, "The Impact of the Surface Shape of the on Power Losses," in *FPNI*, 2014, pp. 1–12.
 - [18] L. Shang and M. Ivantysynova, "A temperature adaptive piston design for swash plate type axial piston machines," *International Journal of Fluid Power*, vol. 18, no. 1, pp. 38–48, 2017, doi: 10.1080/14399776.2016.1213115.
 - [19] R. Ivantysyn, S. Horn, and J. Weber, "Design of a Lead-Free Slipper Bearing for Low Speed Axial Piston Pump Applications," *International Journal of Fluid Power*, vol. 25, no. 2, pp. 183–202, 2024, doi: 10.13052/ijfp1439-9776.2524.
 - [20] A. A. Darbani, L. Shang, J. R. Beale, and M. Ivantysynova, "Slipper surface geometry optimization of the slipper/swashplate interface of swashplate-type axial piston machines," *International Journal of Fluid Power*, vol. 20, no. 2, pp. 245–270, 2019, doi: 10.13052/ijfp1439-9776.2025.
 - [21] S. Horn, R. Ivantysyn, and T. Schmidt, "Tribological properties of different slipper designs of an axial piston pump," in *IFK2024*, 2024, pp. 1–18.
 - [22] U. Wiecek and M. Ivantysynova, "Computer aided optimization of bearing and sealing gaps in hydrostatic machines—the simulation tool caspar," *International Journal of Fluid Power*, vol. 3, no. 1, pp. 7–20, 2002, doi: 10.1080/14399776.2002.10781124.
 - [23] A. Schenk, "Predicting Lubrication Performance Between the Slipper and Swashplate in Axial Piston Hydraulic Machines," Purdue University, 2014.
 - [24] M. Pelosi, "An Investigation on the Fluid-Structure Interaction of Piston/Cylinder Interface," Purdue University, 2012.
 - [25] M. Zecchi, "A novel fluid structure interaction and thermal model to predict the cylinder block/valve plate interface performance in swash plate type axial piston machines," Purdue University, West Lafayette, IN, 2013. Accessed: Aug. 20, 2014. [Online]. Available: <http://gradworks.umi.com/36/13/3613547.html>
 - [26] S. Horn, R. Ivantysyn, and J. Weber, "Validated efficiency improvements by implementation of structures on the slipper surface of an axial piston pump," in *IEEE GFPS 2022*, Naples, 2022.

- [27] R. Ivantysyn and J. Weber, "Investigation of the Thermal Behaviour in the Lubricating Gap of an Axial Piston Pump with Respect to Lifetime," in *11. IFK 2018*, 2018.
- [28] R. Ivantysyn and J. Weber, "'Transparent Pump' – An approach to visualize lifetime limiting factors in axial piston pumps," in *ASME 2016 9th FPNI Ph.D Symposium on Fluid Power*, Florianapolis, Brazil, 2016.
- [29] R. Ivantysyn, J. Weber, A. Kunze, and W. Uffrecht, "Thermal Analysis of the Cylinder Block of an Axial Piston Pump – The Key to Monitoring Efficiency," in *IFK2024*, 2024, pp. 1–21.

When Orifice Flow Must Consider Extensional Effects

Austin Zapata

Andrea Vacca

Purdue University, zapatal@purdue.edu

Abstract.

Orifice flows are a fundamental building block of hydraulic pumps and systems, but the complex physics within them is often simplified in frequently-used relationships for ease-of-use. When polymer melts which exhibit strong non-Newtonian behaviors are used, though, these phenomena become dominant and have significant effects on the flow behavior, leading to inaccuracies if the orifice equation is used. Typical orifice relationships often consider only the effect of flow regime, which is captured by the Reynolds number. The ratio of extensional to shearing flow, often called the Trouton ratio, must be considered in some specialized cases. This study will develop an orifice model which accounts for extensional effects of the fluid by a relationship between dimensionless parameters: the Euler number, the Reynolds number, and the Trouton ratio. The use cases of this expression are then detailed. The model is developed using experimental data of pressure drop and flow for six different fluids across which there are a total of 282 data points. The model's coefficient of correlation is shown to be 0.9990, demonstrating the high accuracy of this model across a wide range of Reynolds numbers.

Keywords. Orifice flow, orifice equation, complex fluids, viscosity.

1. INTRODUCTION

Orifice flows frequently appear in the analysis of hydraulic systems and components. These flows are characterized by their strong dependence on a variety of fluid properties and flow characteristics. In fluid power applications, these flows influence the pressurization of volumes within pumps and systems, such as displacement chambers, hydraulic cylinder chambers, and the flow through valves. It is crucial, then, to have a known relationship between the flow and pressure drop for an orifice. In academia and industry, it is nearly ubiquitous to utilize the orifice equation for these flows, shown below, which is derived from the Bernoulli's equation and conservation of energy.

$$Q = C_d A \sqrt{\frac{\Delta p}{\rho}} \quad (1.1)$$

The orifice equation is often used to model the internal flows of positive displacement machines [1, 2, 3, 4] where constricted flow between volumes is often characterized as orifice flow. The discharge coefficient, given by C_d in Eq. (1.1), accounts for losses in the orifice due to contraction, vortex development, friction, and other factors.

The problem faced by this study, though, cannot be accurately described by Eq. (1.1). The fluids being studied follow the Williamson model [5], which is a viscosity function characterized by a Newtonian plateau at low shear rates and a power law region at high shear rates. These polymer melts also exhibit other non-Newtonian behavior in their elasticity, which greatly effects flow in turbulent conditions and at the entrance of an orifice.

The fundamental work of Smith et. al. [6] was one of the first to parameterize this coefficient. Over the proceeding century, a vast quantity of work has been performed to determine the effect of the orifice diameter ratio on the boundary layer and vortex development in the orifice [7, 8], quantifying how laminar flow leads to lower discharge coefficients [9], what parameters affect regime change in orifice flow [10], and the effect of flow disturbances caused by cavitation on frictional loss [11, 12]. All of the work previously mentioned, though, assumed the flow of a Newtonian fluid, but in this case non-Newtonian operating fluids are used and thus these correlations are not applicable.

Previous approaches to modeling non-Newtonian orifice flow have been quite similar to those for Newtonian fluids. Some of these works, such as those by Dziubiński et. al. [13] and Ntamba et. al. [14], capture non-Newtonian effects by altering the discharge coefficient within the orifice equation. Others, such as Bohra et. al. [15] and Rituraj et. al. [16] form relationships between dimensionless parameters based on experimental results to determine the flow. These are the orifice diameter ratio β , which is the ratio of the orifice diameter to the surrounding pipe diameter, and the aspect ratio, which is defined as the ratio of the orifice length to its diameter.

Relevant to this study are the flows within hydraulic pumps and motors, which are characterized by small values of β and aspect ratios approaching zero, meaning the orifices are sharp. This is because the pressurization of displacement chambers typically occurs when small connections exist between the displacement chambers and the ports. Both of these geometrical considerations must be modeled correctly to maintain the accuracy of flow predictions. The only study which studied flows with non-Newtonian fluids through sharp orifices with small diameter ratios, though, was that from Rituraj et. al. [16]

All of these studies, though, neglect the effect of extensional behavior on the flow. Extension rates are the diagonal components of the extra-stress tensor, and they greatly influence boundary layer formation and vortex development at the entrance of the orifice as discussed in [17]. A simple example of extension is seen below in Figure 1.1, which shows the fluid velocity profile at the entrance of a pipe.

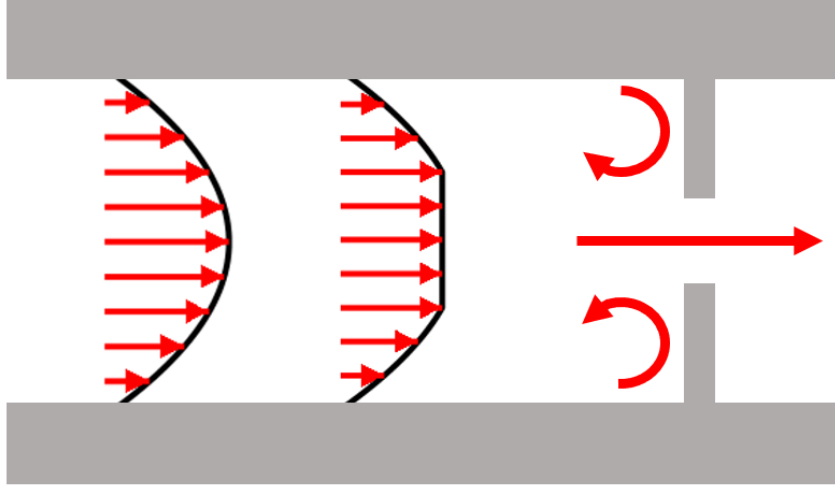


Figure 1.1. Fluid velocity at pipe entrance

As seen in the figure, the fluid flow leading up to the orifice entrance is parabolic as expected in laminar flow. As the fluid approaches the orifice, though, the profile takes on a straighter shape and eventually breaks down as vortices form at the corners between the orifice and the walls, and the resulting streamline through the orifice has a higher velocity than the initial pipe flow. A flow profile like Figure 1.1 is shown in [18]. A consequence of this is that the fluid velocity in the x direction, as denoted in the figure, is changing along the x direction. This is defined as an extension, of the fluid, shown in its mathematical form in Eq. (1.1), below. In this case, the extension would be negative, which is also classified as a contraction of the fluid.

$$\dot{\epsilon} = \left[\frac{\partial u_x}{\partial x} \quad \frac{\partial u_y}{\partial y} \quad \frac{\partial u_z}{\partial z} \right]^T \quad (2.4)$$

It is apparent from Figure 1.1 that this geometric strongly resembles that of an orifice, and thus extensional effects may heavily influence the loss across the orifice. Existing literature on orifice models, though, often neglects these effects. Jones et. al. [19] demonstrated how two distinct fluids with the same shear-viscosity, in this case a Newtonian fluid and a Boger fluid, can have distinct extensional characteristics, and this is a factor which current orifice models for non-Newtonian fluids neglect. Papaevangelou [20] found that the Trouton ratio [21], which is the ratio of extensional-to-shear viscosity, can be upwards of 70 in orifice flow of shear-thinning fluids, whereas the Trouton ratio for a Newtonian fluid is always three. Ascanio et. al. [22] found a correlation between the Euler number, Reynolds number, and Trouton ratio for flow through a conical die. All fluids they tested exhibited significant shear-thinning behavior, but their elastic properties were quite distinct which led to differing relationships between pressure and flow for each fluid tested.

The state-of-the-art demonstrates the significance of elastic and extensional effects in the orifice flow of non-Newtonian fluids, but there is no existing literature for sharp orifices with small orifice diameter ratios for non-Newtonian fluids which considers these extensional effects. This study aims to derive and verify an orifice expression for Williamson

fluids in sharp orifices with small orifice diameter ratios which can capture the elasticity of the fluid by incorporating the Trouton ratio. This expression will be limited to applications using Williamson fluids, but its general form can be utilized for other polymer melts, such as the Carreau-Yasuda fluids from [16], by training similar experimental data from those fluids to find the constants.

With the objectives of the study laid out, the organization of this paper is now detailed. In Section 2, a closed-form expression for the Trouton ratio is developed in terms of measurable parameters and a corresponding orifice relationship is proposed. In Section 3, the experimental setup is discussed including the fluids which are tested and the components used. In Section 4, experimental results are shown and used to identify the unknowns from the model proposed in Section 2, thereby completing the study. Finally, conclusions are drawn based on these results.

2. METHODOLOGY

Two primary methods commonplace in the creation of an orifice model are, as discussed in the introduction, fitting an expression to the discharge coefficient or by fitting a relationship between dimensionless parameters. For this study, the latter method has been selected due to the prior research done on sharp orifices with non-Newtonian fluids. The prior work done on these flows, such as that by Rituraj and Vacca [16], demonstrates the advantages of relating dimensionless parameters via the Buckingham π theorem. Their existing model, shown in Eq. (2.1), considers four dimensionless parameters: the Euler number Eu , which captures energy losses in the flow, the Reynolds number Re , which quantifies the flow regime, the orifice diameter ratio β , which affects vortex and boundary layer development, and the Weissenberg number Wi , which captures the viscoelasticity of the fluid.

$$Eu = \frac{k_1}{Re^{k_2}} \beta^{k_3} Wi^{k_4} \quad (2.1)$$

$$Eu = \frac{\Delta p}{\frac{1}{2} \rho v^2} \quad (2.2)$$

$$Re = \frac{\rho v d}{\eta} \quad (2.3)$$

$$\beta = \frac{d}{D} \quad (2.4)$$

$$Wi = \lambda \dot{\gamma} \quad (2.5)$$

To consider extensional effects, though, the model must use an additional parameter. This parameter which quantifies the effect of extension rate in the flow is the Trouton ratio [21], defined by Eq. (2.6).

$$Tr = \frac{\eta_{Ex}}{\eta} \quad (2.6)$$

Cogswell [23] proposed a model, later dubbed the Cogswell model, for purely-extensional polymer flow which parameterized the extension rate, extensional stress, and extensional viscosity in a capillary rheometer as shown in Eq. (2.7-2.9) respectively below. Because the

capillary rheometer has a similar geometry to an orifice, with a flow constriction between two pipes, the authors assume these expressions correlate well to orifice flows.

$$\dot{\epsilon} = \frac{4}{3(n+1)} \frac{\eta \dot{\gamma}}{\eta_{Entrance}} \quad (2.7)$$

$$\sigma_E = \frac{3(n+1)}{8} \eta_{Entrance} \dot{\gamma} \quad (2.8)$$

$$\eta_{Ex} = \frac{\sigma_E}{\dot{\epsilon}} \quad (2.9)$$

Where n is the power law index of the fluid, $\eta_{Entrance}$ is the shear viscosity at the capillary entrance, $\dot{\gamma}$ is the apparent shear rate, and η is the shear viscosity at the end of the rheometer. An unknown here is the shear viscosity at the capillary entrance, which is difficult to obtain experimentally, but Ye et. al. [24] offered a correlation given by Eq. (2.10).

$$\eta_{Entrance} = \frac{\Delta p}{\dot{\gamma}} \quad (2.10)$$

Substitution of Eq. (2.7-9) into Eq. (2.6) yields the expression for the Trouton ratio, shown in Eq. (2.11)

$$Tr = \frac{9}{32} \left((n+1) \frac{\Delta p}{\eta \dot{\gamma}} \right)^2 \quad (2.11)$$

The shear rate in this equation is given by Eq. (2.12), which was initially derived by Metzner and Reed [25] and later used by Dziubinski et. al. [13] and Rituraj et. al. [16] in works related to non-Newtonian orifice modeling.

$$\dot{\gamma} = \frac{3n+1}{4n} \frac{32Q}{\pi d^3} \quad (2.12)$$

The shear viscosity, naturally, is also a function of the shear rate. Now that the Trouton ratio has been expressed in terms of fluid parameters and flow properties, an orifice expression can be derived of the form in Eq. (2.13).

$$Eu = \frac{k_1}{Re^{k_2}} \beta^{k_3} Wi^{k_4} Tr^{k_5} \quad (2.13)$$

This expression takes the same form as that in the state-of-the-art [16], but has been adjusted to include the Trouton ratio, which will account for the extensional effects of the working fluids. This expression does, though contain five different constants which must be determined. This is achieved through a set of experiments varying pressure, orifice diameter, and fluid as discussed in the next section.

3. EXPERIMENTAL SETUP

The hydraulic circuit for the experimental setup used to obtain the fluid data is given in Figure 3.1. An electric motor drives an external gear machine, which for this study was a MAAG CHEM 25 unit, which then delivers this flow through an orifice. Three orifice diameters were tested, which resulted in orifice diameter ratios of 0.16, 0.08, and 0.04. The hydraulic line leading to the orifice has a diameter of 29 mm. Two pressure transducers, P1

and P2, were placed upstream and downstream, respectively, of the orifice to measure the pressure drop. The flow rate is then measured via a Coriolis type flow meter. A pressure relief valve is placed after the flow meter to allow control over the pressure downstream from the orifice.

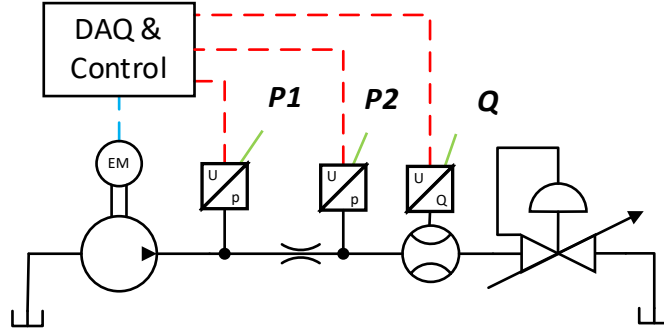


Figure 3.1. ISO Schematic of the Experimental Hydraulic Circuit

For the fluids in question, the viscosity-shear rate relationship follows the Williamson model [5], shown by Eq. (3.1). This relationship is characterized by a Newtonian plateau at low shear rates, where the viscosity is close to η_0 , dubbed the zero-shear rate viscosity, and a power law region at high shear rates, the slope of which is determined by the power law index n . The transition between these two regions is controlled via the characteristic time λ .

$$\eta = \frac{\eta_0}{1 + (\lambda\dot{\gamma})^{1-n}} \quad (3.1)$$

Three distinct shear-thinning fluids were tested for this study, all of which closely adhere to the Williamson model, as shown via Figure 3.2. These fluids have zero shear rate viscosities ranging from 5 - 100 $Pa \cdot s$, power law indices between 0.4 - 0.6, and characteristic times between 0.001 - 0.1 s. Further information on these fluids cannot be disclosed due to confidentiality.

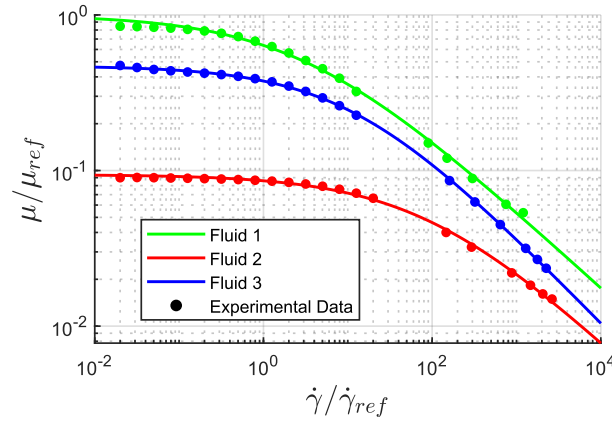


Figure 3.2. ISO Schematic of the Experimental Hydraulic Circuit

Experiments are conducted over a wide range of conditions by varying the pump speed, orifice diameter, and back pressure P2. At each data point, the pressure on both sides of the orifice and flow are recorded. Overall, 282 data points were collected with which the model was trained.

4. MODEL DEVELOPMENT

The experimental data for pressure, flow, and fluid properties was used to obtain the dimensionless parameters in Eq. (2.13) at each data point. This data is collected in Figure 4.1 below. The trend of Re versus Eu aligns with expectations [16], that being a negative linear relationship on a log-log scale. Variation from this line is in large-part due to the factors discussed in Section 2, those being the vortex and boundary layer development, fluid viscoelasticity, and extensional effects of the fluid.

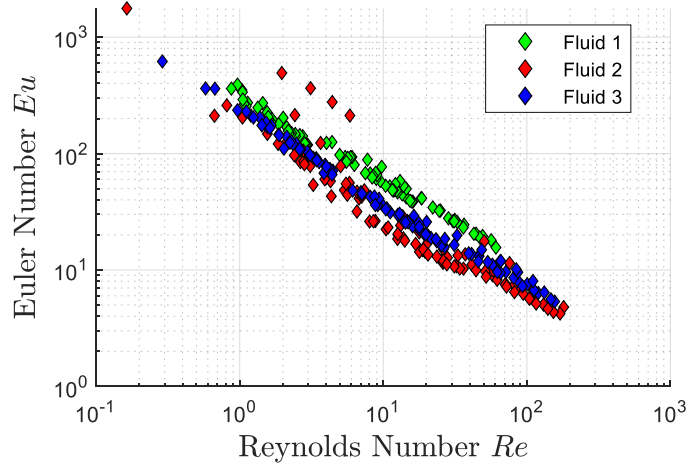


Figure 4.1. Experimental Data of all three fluids for the Reynolds number versus Euler number

The correlation proposed by Eq. (2.13) is then trained to determine the values of the constants k_1, k_2, k_3, k_4 and k_5 via a least-squares multiple linear regression analysis performed on a logarithmic scale to minimize errors across the data collected, which spanned multiple orders of magnitude, and to adhere to the linear trend observed in Figure 4.1 on a log-log scale. The constants obtained for this analysis are shown in Table 4.1.

Constant	Value
k_1	30.1402
k_2	0.9865
k_3	0.0116
k_4	0.0059

k_5	0.4684
-------	--------

Table 4.1. Constants for the proposed relationship

This makes the final correlation:

$$Eu = \frac{30.1402}{Re^{0.9865}} \beta^{0.0116} Wi^{0.0059} Tr^{0.4684} \quad (4.1)$$

The coefficient of correlation for this relationship is $R^2 = 0.9990$. The range of applicability for this correlation is $0.1 < Re < 200$ and $0.04 \leq \beta \leq 0.16$. Figure 4.2 shows the experiments and predictions overlayed separated by fluid, which demonstrates the high correlation between this model and the experimental data.

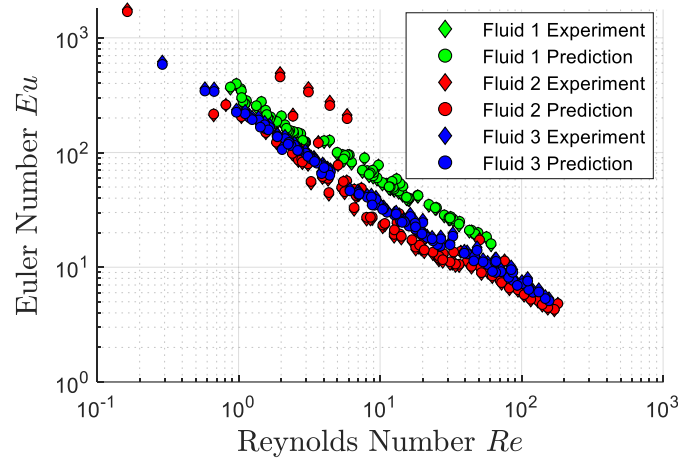


Figure 4.2. Reynolds number versus Euler number for the experiment and the proposed orifice model

Figure 4.3 also serves to validate the model by comparing the measured Euler number versus the predicted Euler number. The plot shows how closely the data points follow the ideal curve, that being the $y = x$ line.

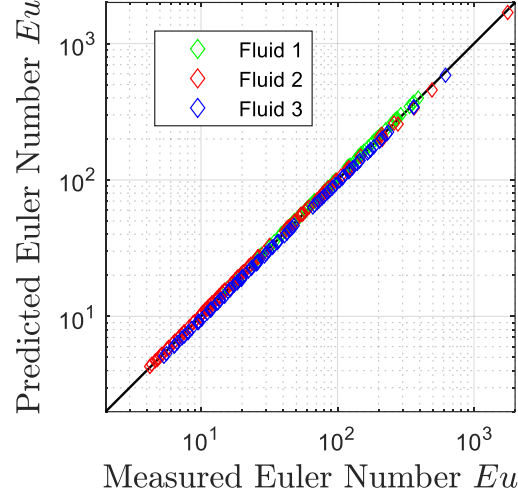


Figure 4.3. Measured versus predicted Euler number for the proposed orifice model

To validate the proposed orifice model, the expression developed is tested on a separate set of experimental data from those points used to train it. The data from Rituraj and Vacca [16] is utilized to test its efficacy to other fluids of similar non-Newtonian characteristics on which the model was not developed. The results of this can be seen in Figure 4.4

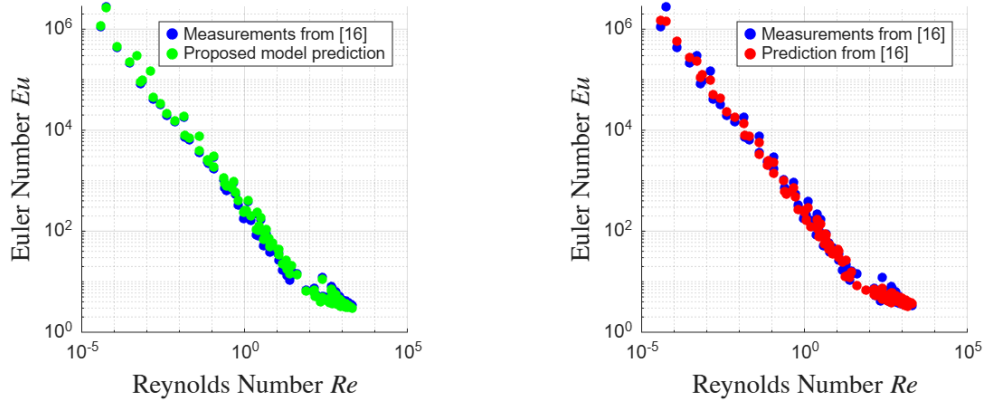


Figure 4.4. Validation results for the proposed orifice model compared to that from [16]

Figure 4.4 visually shows the clear improvement of the consideration of the Trouton ratio on the accuracy of the orifice model, even using data on which the model was not trained. Numerical analysis further supports this claim. While the standard error of the orifice model from [16] is 0.0020, the model proposed by this study has a standard error of 0.0013, showing a marked improvement. This serves to validate both the generalizability of this orifice model and the importance of the consideration of extensional effects on orifice flows for non-Newtonian fluids.

5. CONCLUSION

In this study, an orifice model was proposed for sharp orifice flows with non-Newtonian fluids to consider the extensional behavior of the fluid and its effect on the effective viscosity. An expression for the Trouton ratio was developed in terms of parameters from the Cogswell model. Experiments ranging across orifice diameter ratios of 0.04, 0.08, and 0.16 were tested with pressure differences between 2-50 bar and flows ranging from 0.1-1.5 L/min. The experimental data was then used to develop a model relating the Euler number to the Reynolds number, the orifice diameter ratio, the Weissenberg number, and the Trouton ratio. The derived relationship is applicable to polymer melts which exhibit prominent shear-thinning behavior and obey the Williamson fluid model. The relationship derived successfully parameterizes the flow across the orifice in terms of fluid properties, orifice geometry, and the pressure drop, as shown by its correlation coefficient of 0.9990 with the experimental data. This model was then validated using experimental data from a previous study, showing that the proposed model gives improved predictions from the state-of-the-art for data on which it was not trained.

6. REFERENCES

- [1] D. Pan and A. Vacca, "Modeling of Crescent-Type Internal Gear Pumps Considering Gear Radial Micromotion," *Chemical Engineering & Technology*, vol. 46, no. 1, pp. 128-136, 2022.
- [2] A. Vacca, S. Dhar and T. Opperwall, "A Coupled Lumped Parameter and CFD Approach for Modeling External Gear Machines," in *The 12th Scandanavian International Conference on Fluid Power*, Tampere, 2011.
- [3] A. Vacca and M. Guidetti, "Modelling and experimental validation of external spur gear machines," *Simulation Modelling Practice and Theory*, vol. 19, pp. 2007-2031, 2011.
- [4] M. Eaton, P. S. Keogh and K. A. Edge, "The modelling, prediction, and experimental evaluation of gear pump meshing pressures with particular reference to aero-engine fuel pumps," *Proceedings of the Institution of Mechanical Engineers, Part I: Journal of Systems and Control Engineering*, vol. 220, no. 5, pp. 365-379, 2006.
- [5] R. V. Williamson, "The Flow of Pseudoplastic Materials," *Industrial & Engineering Chemistry*, vol. 21, no. 11, pp. 1108-1111, 1929.
- [6] D. Smith and W. Walker, "Orifice Flow," *Proceedings of the Institution of Mechanical Engineers*, vol. 104, no. 1, pp. 23-36, 1923.
- [7] F. Shan, Z. Liu, W. Liu and Y. Tsuji, "Effects of the orifice to pipe diameter ratio on orifice flows," *Chemical Engineering Science*, vol. 152, pp. 497-506, 2016.
- [8] B. Gulsacan, N. Tokgoz, E. Karakas, M. Aureli and C. Evrensel, "Effect of orifice thickness-to-diameter ratio on turbulent orifice flow: An experimental and numerical

- investigation,” *International Communications in Heat and Mass Transfer*, vol. 151, p. 107213, 2024.
- [9] W. Borutzky, B. Barnard and J. Thoma, “An orifice flow model for laminar and turbulent conditions,” *Simulation Modelling Practice and Theory*, vol. 10, no. 3-4, pp. 141-152, 2002.
 - [10] Y. Yan and R. B. Thorpe, “Flow regime transitions due to cavitation in the flow through an orifice,” *International Journal of Multiphase Flow*, vol. 16, no. 6, pp. 1023-1045, 1990.
 - [11] K. Sato and Y. Saito, “Unstable Cavitation Behavior in a Circular-Cylindrical Orifice Flow,” *JSME International Journal Series B Fluids and Thermal Engineering*, vol. 45, no. 3, pp. 638-645, 2002.
 - [12] S. Dabiri, A. Sirignano and D. Joseph, “A numerical study on the effects of cavitation on orifice flow,” *Physics of Fluids*, vol. 22, no. 4, p. 042102, 2010.
 - [13] A. Marcinkowski and M. Dziubiński, “Discharge of Newtonian and non-Newtonian Liquids from Tanks,” *Chemical Engineering Research and Design*, vol. 84, no. 12, pp. 1194-1198, 2006.
 - [14] B. N. Ntamba and V. Fester, “Pressure Losses and Limiting Reynolds Numbers for Non-Newtonian Fluids in Short Square-Edged Orifice Plates,” *Journal of Fluids Engineering*, vol. 134, no. 9, p. 091204, 2012.
 - [15] L. K. Bohra, L. Mincks and S. Garimella, “Experimental Investigation of Pressure Drop Characteristics of Viscous Fluid Flow Through Small Diameter Orifices,” *Journal of Fluids Engineering*, vol. 143, no. 2, p. 021306, 2021.
 - [16] R. and A. Vacca, “Modeling the Flow of Non-Newtonian Fluids Through Sharp Orifices,” *Journal of Fluids Engineering*, vol. 140, 2018.
 - [17] M. D. J., “Extensional flow of non-newtonian fluids—A review,” *Polymer Engineering and Science*, vol. 11, no. 6, pp. 433-445, 1971.
 - [18] T. C. H., “A revisit of generalized newtonian fluids,” *Journal of Rheology*, vol. 64, no. 3, pp. 493-504, 2020.
 - [19] D. M. Jones, K. Walters and P. R. Williams, “On the extensional viscosity of mobile polymer solutions,” *Rheologica Acta*, vol. 26, pp. 20-30, 1987.
 - [20] G. Papaevangelou, “The flow through an orifice of semi-rigid-polymer solutions,” in *International Conference on Continuum Mechanics*, Chalkida, Greece, 2006.
 - [21] F. Trouton, “On the coefficient of viscous traction and its relation to that of Viscosity,” *Proceedings of the Royal Society of London. Series A, Containing Papers of a Mathematical and Physical Character*, vol. 77, no. 519, pp. 426-440, 1906.

- [22] G. Ascanio, P. Carreau, E. Brito-De La Fuente and P. Tanguy, "Orifice Flowmeter for Measuring Extensional Rheological Properties," *The Canadian Journal of Chemical Engineering*, vol. 80, pp. 1189-1196, 2002.
- [23] F. N. Cogswell, *Polymer Melt Rheology: A Guide for Industrial Practice*, Woodhead Publishing, 1981.
- [24] Q. Ye, W. Niemeier, O. Tiedje and B. Shen, "Determining extensional viscosity from the measured pressure drop in a capillary rheometer for paint liquids based on fluid dynamic simulations," *Journal of Non-Newtonian Fluid Mechanics*, vol. 343, p. 105444, 2025.
- [25] A. B. Metzner and J. C. Reed, "Flow of non-newtonian fluids—correlation of the laminar, transition, and turbulent-flow regions," *American Institute of Chemical Engineers Journal*, vol. 1, no. 4, pp. 434-440, 1955.

Design and Optimization of Fluid Supply Flow Paths for a Balanced Twin-Lip Vane Pump

Jinglun Qu^{a,*}, Andrea Vacca^{a,b}, Sri Krishna Uppaluri^c

^a*School of Mechanical Engineering, Purdue University, West Lafayette, 47907, IN, USA*

^b*School of Agricultural and Biological Engineering, Purdue University, West Lafayette, 47907, IN, USA*

^c*Veljan Denison Limited, A18 and 19, APIE, Balanagar, Hyderabad, 500037, Telangana, India*

**Corresponding author: qu134@purdue.edu*

Abstract

This paper presents a methodology for designing and optimizing the fluid supply flow path of a balanced twin-lip vane pump, supported by experimental validation. Cavitation in the suction port, caused by insufficient fluid supply to the displacement chamber, is a leading cause of pump failure due to the severe damage it inflicts on critical components. This paper includes the further development of a previously established numerical model to consider the complex supply path of the casing. The methodology utilizes preliminary Computational Fluid Dynamics (CFD) simulations to characterize the hydraulic resistance of the upstream orifices and chambers. The baseline model was validated against existing experimental results to confirm its fidelity. The updated extended model successfully captures the localized cavitation effects occurring within the casing, leading to proposed geometry updates. By characterizing the new design using CFD, the model demonstrates significantly improved supply flow capabilities. Further updates of the cam-ring orifice were conducted, and simulation results predict the substantially improved performance of the pump at maximum operating speeds, which is experimentally validated in this study.

Keywords: Vane pump, Fluid supply flow paths, Cavitation, Lumped parameter model, Computational Fluid Dynamics.

Nomenclature

Symbols

A	Cross-sectional area [m ²]
C_d	Discharge coefficient [-]
c_e	Coefficient of restitution [-]
E	Young's modulus [Pa]
η	Efficiency [-]
K	Fluid bulk modulus [Pa]
$\dot{m}$	Mass flow rate [kg/s]
p	Pressure [Pa]
Δp	Differential pressure drop [Pa]
Q	Flow rate [L/min]
Re_{crit}	Critical Reynolds number [-]
T	Temperature [°C]
t	Time [s]
V	Volume [m ³]
α_{air}	Dissolved air volume fraction [-]
α_{gas}	Gas volume fraction [-]
ν	Kinematic viscosity [m ² /s]
ρ	Fluid density [kg/m ³]
Ω	Normalized operating speed [RPM]

Subscripts

atm	Atmospheric conditions
CFD	Computed via Computational Fluid Dynamics
i, j	Control volume or chamber indices
in	Inlet
max	Maximum
nom	Nominal
out	Outlet
vol	Volumetric

Abbreviations

CFD	Computational Fluid Dynamics
EHL	Elasto-Hydrodynamic Lubrication
FSMI	Fluid-Structure Motion Interaction
LC	Lip Chamber
LP	Lumped-Parameter
RPM	Revolutions Per Minute
VC	Vane Chamber

1 Introduction

Vane pumps are rotary positive displacement machines widely utilized in automotive applications and hydraulic control actuation systems due to their compactness, low noise, and cost efficiency. Among the various design configurations, the balanced twin-lip vane pump is highly suitable for high-pressure applications because its symmetrical porting eliminates net radial loads on the rotor [1]. Furthermore, the twin-lip design allows for pressure compensation mechanisms that handle the critical vane-cam-ring interface, preventing excessive wear [1].

Accurately modeling these units requires capturing the complex interactions of various physical domains. A comprehensive multi-physics baseline simulation model was previously developed by Mistry et al. [1], which coupled fluid dynamics, contact dynamics, lubricating interfaces, and mixed elasto-hydrodynamic lubrication (EHL) into one unified framework. While this baseline model successfully evaluated internal pressures using a lumped-parameter formulation and solved the planar motions of the vanes using Newton’s laws of motion, its primary limitation was that the fluid supply flow path through the pump casing was never modeled. Insufficient fluid supply through these unmodeled casing paths can lead to cavitation in the suction port, a phenomenon heavily dependent on fluid properties [2], which could then limit the pump’s operational envelope and damage components.

This paper expands upon the previous numerical model [1] to explicitly consider the supply path through the casing. The cartridge unit and the overall fluid flow path are illustrated in Figure 1. Utilizing preliminary CFD simulations, the characteristics of the supply orifices and chambers are defined and integrated into the model. Following geometric analysis, further improvements to the casing channels and the cam-ring orifice are proposed to expand the volumetric efficiency and capacity of the unit at maximum operating speeds (Ω_{max}).

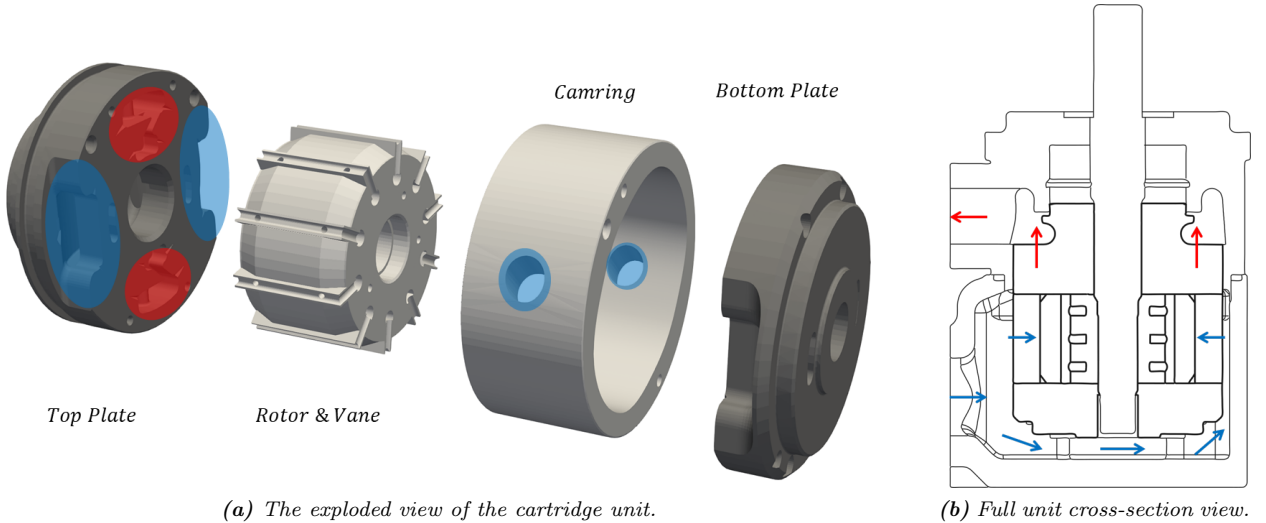


Figure 1: In figure (a), the blue area represents the inlet flow port, and the red area represents the outlet flow port. In figure (b), the outline includes the casing (thinner lines) and the cartridge unit (bold lines), with arrows indicating the flow path and colors corresponding to low and high pressure.

Several recent studies have focused specifically on the filling capability and cavitation phenomena within vane pumps. Yan et al. [3] utilized 3D Computational Fluid Dynamics (CFD) to capture the cavitation performance of a sliding vane pump; however, their study was restricted to a non-balanced single-lip design and only investigated lower rotational speeds. For balanced vane pumps, Lobsinger et al. [4] performed multi-phase CFD simulations to evaluate the effects of air content on the outlet flow and filling characteristics. Similarly, Stuppioni et al. [5] and Suzuki et al. [6] demonstrated the capability of 3D CFD approaches to model dissolved gas and predict pressure ripples by comparing simulations to measured internal unit pressures. Furthermore, Rundo et al. [7] utilized CFD to analyze how various geometrical features affect the

filling behavior of vane pumps operating at high shaft speeds.

While these past works demonstrate that 3D CFD approaches allow for an in-depth analysis of internal flow features—specifically the aeration and vaporization effects intimately related to a unit’s filling capability—they possess significant limitations. Because full 3D CFD models are highly computationally expensive, they typically require initial assumptions of fixed gap heights at the critical vane-cam-ring interface. Consequently, they cannot dynamically evaluate the planar motions and tilting of the vanes during operation, which are essential for accurately predicting the torque losses and overall hydromechanical efficiency of the unit.

To overcome these limitations, the methodology proposed in this paper bridges the gap between high-fidelity spatial flow mapping and computationally efficient dynamic modeling. Instead of relying solely on a computationally heavy CFD model or a 1D analytical model, this work utilizes preliminary steady-state CFD simulations to extract the hydraulic restrictions and orifice discharge coefficients of the unmodeled casing paths. These characterized parameters are then directly integrated into a fast, dynamically coupled lumped-parameter framework. This approach successfully captures the high-speed cavitation limitations of the entire unit while preserving the dynamic evaluation of mechanical losses, allowing for rapid geometry optimization that extends the pump’s operational speed.

The remainder of this paper is organized as follows. Section 2 introduces the numerical framework, detailing the geometric evaluation and the extended fluid dynamics module. Section 3 describes the CFD characterization used to define the upstream casing parameters. Section 4 presents the simulation results, identifying baseline cavitation limitations. Section 5 introduces the optimized casing geometries with numerical simulation results, and Section 6 outlines the experimental validation. Finally, Section 7 draws the main conclusions of this work.

2 Numerical Framework

The foundation of the numerical framework utilized in this study is a strongly coupled Fluid-Structure Motion Interaction (FSMI) module, which was comprehensively developed, detailed, and experimentally validated in the reference work by Mistry et al. [1]. This baseline architecture is designed to evaluate the highly complex internal physics of a twin-lip balanced vane pump by coupling fluid dynamics, rigid body kinematics, contact mechanics, and mixed Elasto-Hydrodynamic Lubrication (EHL) into a single unified environment. To achieve this, the FSMI framework is structured around fundamental sub-modules operating in tandem, as shown in Figure 2.

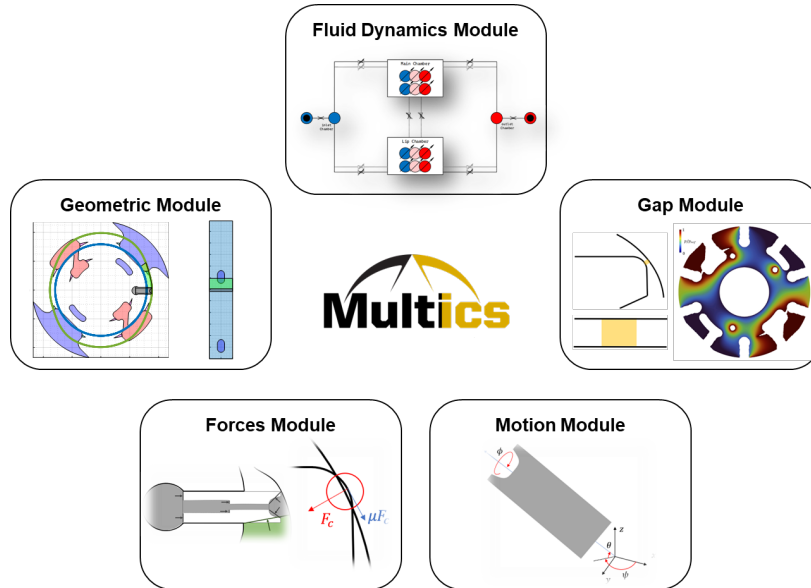


Figure 2: Overview of the coupled multi-physics simulation architecture.

This paper focuses on extending the baseline multi-physics model, specifically within the geometric and fluid dynamics aspects, to investigate the cavitation phenomena occurring during high-speed unit operation. While the original framework effectively captured the core cartridge dynamics, it did not account for the upstream casing supply restrictions. The sections below detail the procedural advancements made to capture these effects, highlighting the integration of characterized casing supply pathways and the expansion of the fluid circuit to dynamically track localized density drops and premature cavitation.

2.1 Geometric Model

The geometric module functions as a preprocessor that extracts precise geometrical features based directly on the physical geometry of the pump's internal components. By evaluating this geometry, representative polygons are generated without relying on simplified analytical assumptions [1]. This module calculates the theoretical radial coordinates of the vanes for every degree of shaft rotation by moving the vane profile iteratively until the distance between the vane-lip and cam ring profiles is near zero [1]. By stitching bounding points into closed polygons, the module accurately defines the instantaneous volumes of the two primary fluid control volumes: the Vane Chamber (VC) and the Lip Chamber (LC). Finally, it performs polygon intersections to evaluate the overlapping flow areas between the displacement chambers and the inlet/outlet ports. These geometrical features are stored as a lookup table and supplied to the fluid dynamics module.

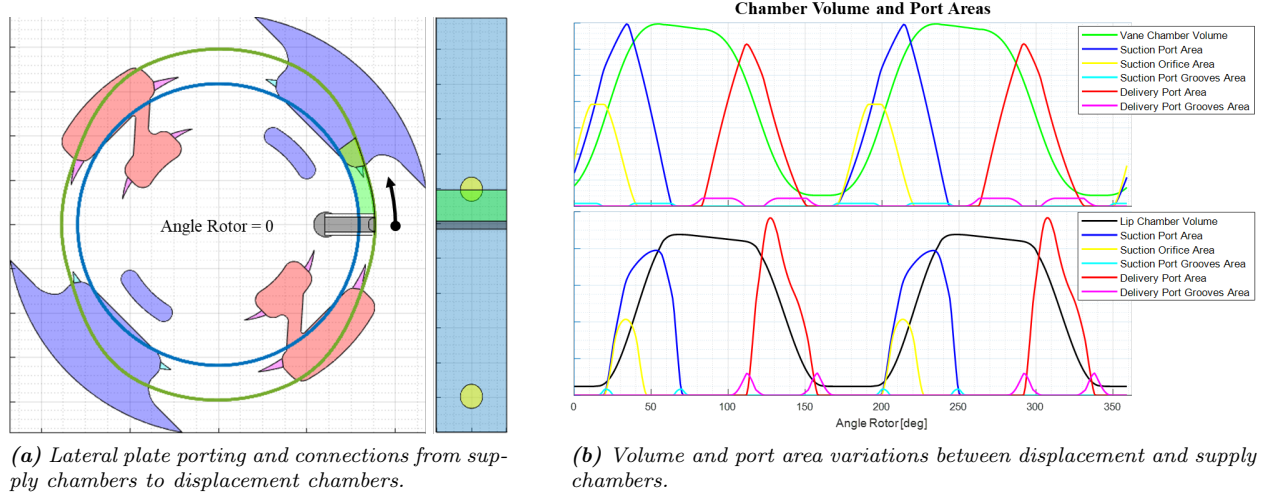


Figure 3: Plot (a) illustrates the geometric shape of the lateral plate port and chamber volume connections at the initial state. Plot (b) illustrates the corresponding areas (y-axis, left) and volumes (y-axis, right) at each angular step (x-axis, from 0° to 360°).

The geometry of the Vane Chamber (VC) and Lip Chamber (LC) volumes at different angles is continuously evaluated as the rotor spins. Zero degrees is defined when rotor vane 1 is perfectly horizontal as shown in Figure 3a. At each angular step, the module solves for the instantaneous volume of the displacement chambers and the corresponding connection area between the ports and the chamber volumes. Figure 3b illustrates these chamber volumes and connection port areas at each angular step, which directly dictate the fluid mass transfer in the subsequent fluid dynamics module. To visualize this kinematic progression, Figure 4 demonstrates a displacement chamber undergoing half of a full rotation cycle, and captures the timing of the chamber enters and exits the low-pressure inlet port, as well as its transition into and out of the high-pressure outlet port at various key angular locations.

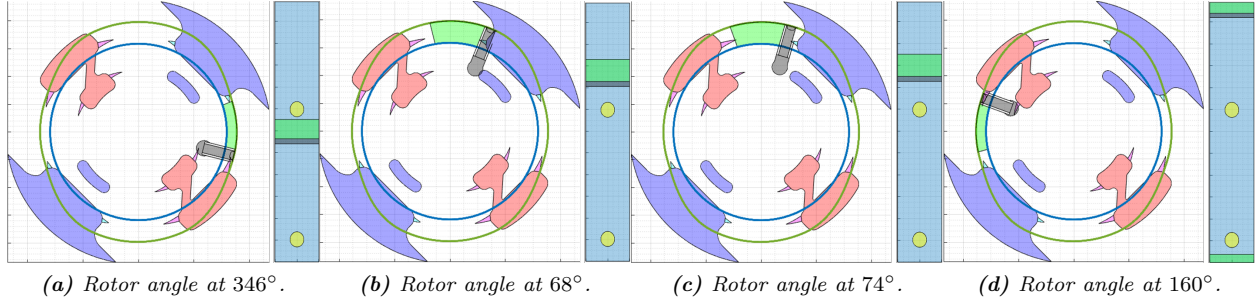


Figure 4: Demonstration of a displacement chamber undergoing half of a full rotation cycle (blue: inlet port, red: outlet port). The figure illustrates the displacement chamber entering the inlet port at 346° (a), exiting the inlet port at 68° (b), entering the high-pressure outlet port at 74° (c), and exiting the outlet port at 160° (d). Subsequent plots tracking performance across rotor angles directly correspond to these critical port connection points.

2.2 Fluid Dynamics Module

The fluid dynamics module evaluates the fluid state inside the control volumes using a lumped-parameter approach centered around the fundamental pressure build-up equation [2]. This equation is derived directly from the principle of mass conservation for a variable control volume V_i :

$$\sum \dot{m}_{in,i} - \sum \dot{m}_{out,i} = \frac{d(\rho_i V_i)}{dt} = V_i \frac{d\rho_i}{dt} + \rho_i \frac{dV_i}{dt} \quad (1)$$

To relate the transient changes in localized fluid density (ρ_i) to the internal chamber pressure (p_i), the fluid bulk modulus (K_i) is utilized. Substituting this relationship into Equation 1 yields the pressure build-up equation:

$$\frac{dp_i}{dt} = \frac{K_i}{\rho_i V_i} \left(\sum \dot{m}_{in,i} - \sum \dot{m}_{out,i} - \rho_i \frac{dV_i}{dt} \right) \quad (2)$$

Within this framework, cavitation effects are captured through the pressure-dependent state formulations of density (ρ_i) and bulk modulus (K_i) [1]. The fluid is modeled as a homogeneous mixture of liquid, dissolved air, and free gas/vapor. When the local pressure in the suction chamber drops, the fluid density decreases, and the bulk modulus drops drastically [2]. This dynamic variation signifies the onset of cavitation, allowing the model to accurately track the percentage of gas and vapor bubbles dynamically.

While the baseline model applied this formulation successfully, it previously considered only the core cartridge unit of the pump. Consequently, it assumed an unrestrictive, ideal fluid supply directly to the cartridge inlet ports, neglecting the complex upstream geometries of the pump casing. To accurately capture the high-speed filling limitations of the machine, the fluid dynamic model was extended to explicitly include the complete flow path from the casing inlet to the cartridge inlet. Physically, the hydraulic fluid traverses a tortuous internal pathway, routing from the main inlet chamber to a base chamber via a rectangular-shaped orifice, and continuing from the base chamber through a thin rectangular slot to reach the back chamber of the cartridge.

To maintain high computational efficiency while capturing these spatial restrictions, preliminary steady-state 3D CFD simulations are utilized purely for physical characterization to evaluate the pressure drops across these simplified casing volumes. These extracted parameters are then used to define orifice discharge coefficients (C_d) for each casing segment.

Once characterized, these intricate casing supply pathways are incorporated directly into the LP fluid circuit as upstream variable control volumes connected by variable orifices, as depicted in Figure 5. The fluid dynamics module actively resolves the mass flux through these characterized casing channels, which are situated upstream of the fluid entering the primary VC and LC displacement chambers. By evaluating the pressure restrictions through these upstream casing channels, the extended numerical framework is fully equipped to dynamically track severe fluid density drops and the subsequent onset of cavitation occurring at the cartridge inlet.

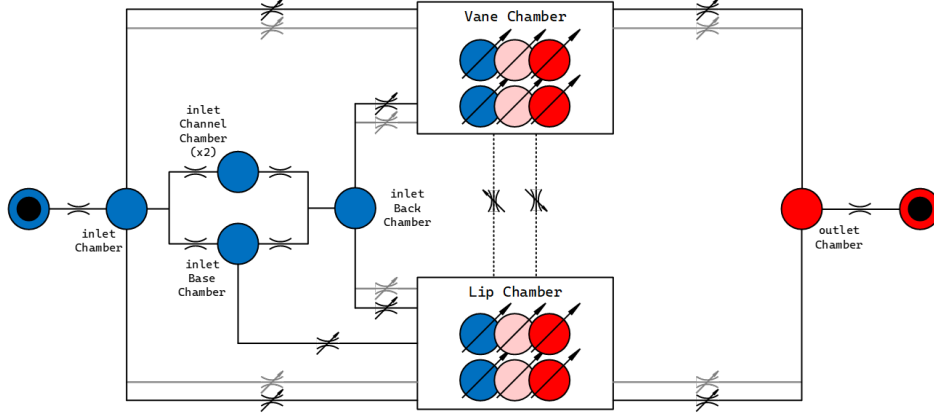


Figure 5: Lumped-parameter method of the updated fluid path, illustrating how the CFD-characterized casing supply paths are integrated as upstream control volumes into the overall multi-physics framework shown in Figure 2.

3 CFD Characterization

To accurately capture the hydraulic restrictions prior to the primary suction port, the casing supply path was thoroughly analyzed using CFD. To reduce computational overhead while maintaining strict volumetric fidelity, the 3D CFD model utilized a simplified geometry that preserved the exact fluid volumes and critical cross-sectional areas of the physical pump. Specifically, the modeled fluid domain was discretized into three primary supply pathways. The first pathway routes, in Figure 6, fluid from the inlet chamber to the base chamber via a rectangular-shaped orifice. The second pathway channels fluid from the base chamber through a thin rectangular slot to reach the back chamber. Finally, a third supplementary pathway directs fluid from the inlet through a small channel directly to the back chamber.

The primary objective of these CFD simulations is to characterize the hydraulic resistance of these specific pathways so they can be accurately represented in the 1D lumped-parameter model [1]. This is achieved by determining the orifice discharge coefficient (C_d) for each geometric restriction. The CFD simulations were performed using SOLIDWORKS Flow Simulation software. A steady-state, incompressible pressure-based solver was utilized with a standard $k - \epsilon$ turbulence model to accurately capture the turbulent flow structures. Convergence was monitored by ensuring all residuals dropped below 10^{-4} and mass imbalances were negligible. The simulation methodology prescribes a steady-state nominal mass flow rate ($\dot{m}_{nom}$) under a nominal operating pressure boundary condition. By solving the fundamental Navier-Stokes equations across the simplified spatial domain, the CFD model evaluates the corresponding pressure drop (Δp_{CFD}) that occurs across each volumetric segment.

This methodology leverages the standard orifice loss equation, derived from Bernoulli's principle for flow across a sudden restriction. Considering the fluid as incompressible across the localized restriction, the mass flow rate ($\dot{m}$) is mathematically defined as:

$$\dot{m} = \text{sign}(p_i - p_j) C_d A \sqrt{2\rho |p_i - p_j|} \quad (3)$$

By rearranging this governing equation and substituting the nominal parameters evaluated during the CFD process, the discharge coefficient is explicitly calculated as:

$$C_d = \frac{\dot{m}_{nom}}{A \sqrt{2\rho \Delta p_{CFD}}} \quad (4)$$

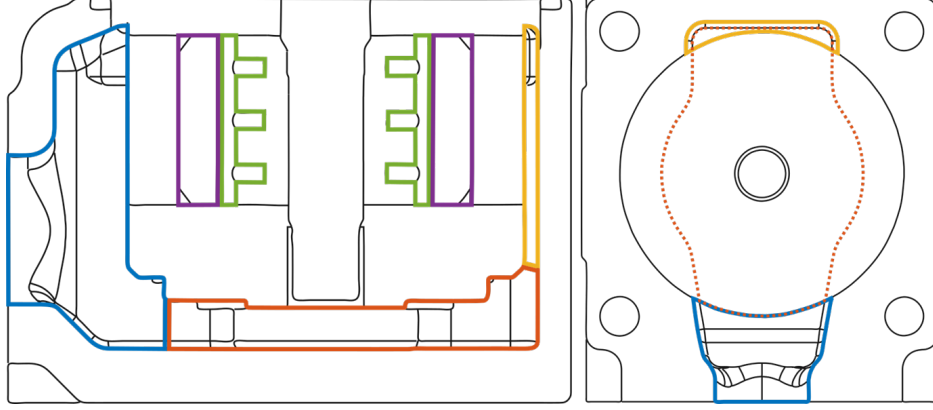


Figure 6: Each chamber defined in the fluid circuit is highlighted in the pump frame in the left cross-section view; the right figure shows the top view of the lower section flow path with dotted lines indicating routing below the cartridge body.

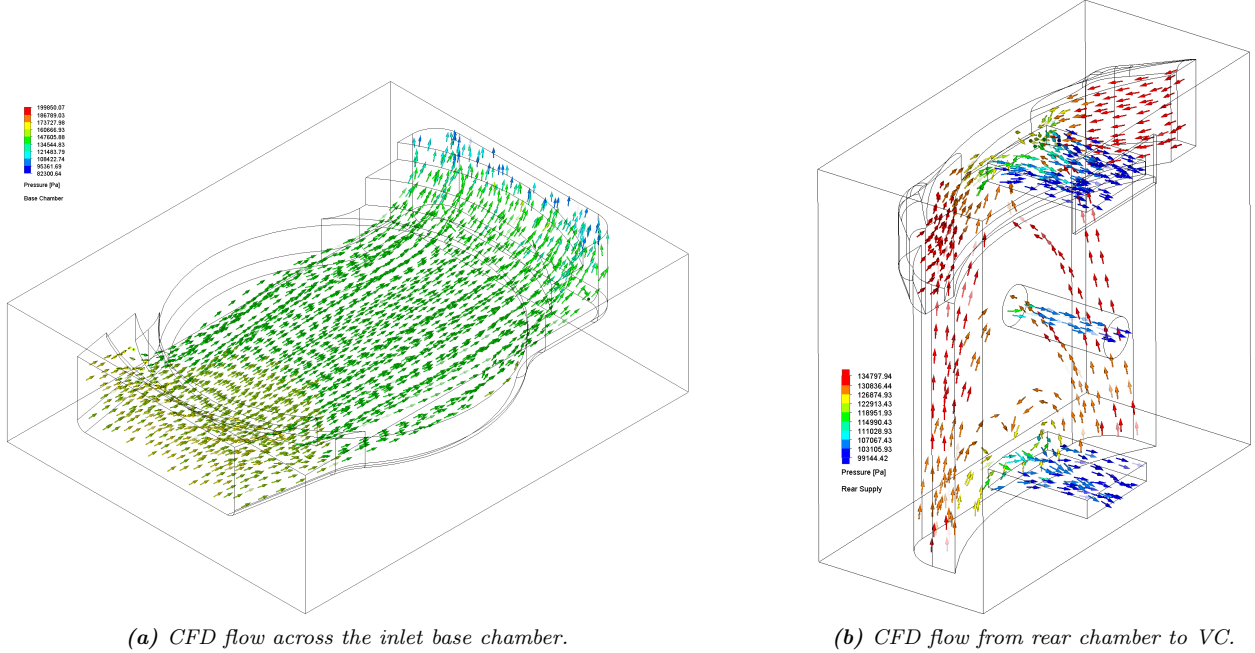
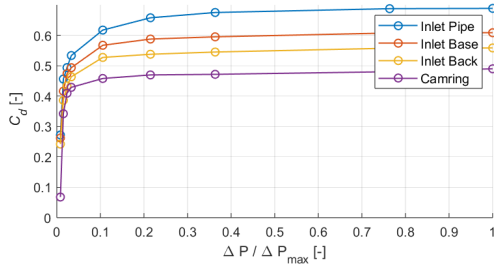
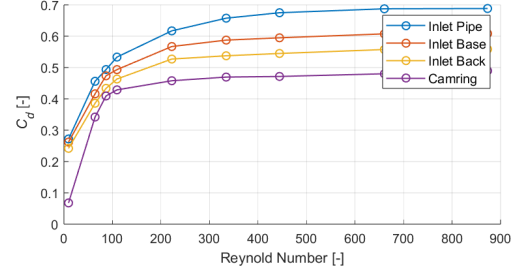


Figure 7: CFD simulation for each port connection; note the pressure drop (colored) pressure drop across the simplified fluid volumes used to evaluate the discharge coefficients.

Once these specific discharge coefficients are characterized for their corresponding orifices, they are integrated into the lumped-parameter fluid circuit of the whole unit. As highlighted by the CFD characterization process shown in Figure 7, the characteristic of the orifice varies significantly across the path from the inlet to the cartridge unit. This insight ultimately allowed the numerical model to accurately capture the severe cavitation effect previously observed at the second suction port.



(a) Orifice discharge coefficient at different Δp .



(b) Orifice discharge coefficient at corresponding Reynolds number.

Figure 8: CFD simulation results evaluating the discharge coefficients (a) and critical Reynolds transition points (b) for each geometric restriction.

Figure 8 presents the results from the CFD characterization, highlighting the distinct discharge coefficients (C_d) and critical Reynolds numbers (Re_{crit}) for the different port connections. The precise evaluation of C_d is of paramount importance, as it actively dictates the magnitude of the pressure drop across each specific geometric restriction for a given mass flow rate, effectively capturing the localized flow contraction (vena contracta) and viscous losses. Furthermore, identifying the critical Reynolds number (Re_{crit}) is essential for the lumped-parameter fluid circuit. Re_{crit} defines the specific flow condition at which the fluid transitions from a laminar regime to a turbulent regime. Capturing this transition point allows the 1D model to accurately switch between linear and quadratic flow-pressure relations, ensuring a high-fidelity evaluation of the hydraulic resistance across all operating speeds.

4 Simulation Results

Prior to evaluating the performance improvements of the newly proposed casing and cam ring designs, it is crucial to establish the baseline parameters and fundamental assumptions of the multi-physics simulation. The model evaluates a reference balanced twin-lip vane pump operating with an ISO VG 46 mineral hydraulic fluid. The operating conditions and outcomes are normalized relative to the unit's selected operating speed (Ω_{max}) to protect proprietary data, while the nominal operating temperature is fixed at 25°C. The primary fluid properties, structural material constants, and surface features utilized in the simulation are defined in Table 2.

Table 1: Pump specifications.

Manufacturer	Veljan Denison Limited
Model Number	VT6C025
Displacement	79.3 cm ³ /rev
Maximum Pressure	280 bar
Maximum Speed	2800 rev/min
Number of Vanes	10

Several key assumptions are made to maintain computational efficiency while preserving physical fidelity throughout the simulation. First, the fluid dynamics module assumes isothermal flow conditions within the control volumes. Second, while the pressure plates undergo significant deformation under high-pressure loads, structural analysis indicates that the inward deformation of the over-compensated bottom plate and the outward deformation of the top plate effectively reduce the net load to have constant gap height. Additionally, the dynamic contact and friction forces at the critical vane-cam-ring interface are evaluated using a mixed Elasto-Hydrodynamic Lubrication (EHL) formulation [1], which is a well-proven physical phenomenon dictating these interactions. This formulation relies directly on the measured surface roughness and radius of curvature of the contacting steel components.

Table 2: Simulation Parameters

Fluid Properties	
Temperature (T)	40 °C
Atmospheric Pressure (p_{atm})	1.01325×10^5 Pa
Dissolved Air Volume Fraction (α_{air})	0.08 -
Atmospheric Density (ρ_{atm})	860 kg/m ³
Atmospheric Kinematic Viscosity (ν_{atm})	46×10^{-6} m ² /s
Structural & Material Properties	
Young's Modulus (E)	287×10^9 Pa
Coefficient of Restitution (c_e)	0.78 -
Surface Roughness	0.2 μ m
Radius of Curvature	Undisclosed

4.1 Flow Behavior

Before evaluating the newly proposed optimized geometries, the extended whole-unit fluid dynamic model was tested to ensure it maintained the accuracy established by the baseline cartridge-only model, which has been experimentally validated in the past. The simulation of the updated fluid circuit was run across standard operational ranges and validated against existing experimental results.

As illustrated in Figure 9(a), the integration of the casing supply pathways demonstrates excellent agreement with the original cartridge-only model [1]. Specifically, the modified simulation accurately matches the original model predictions for volumetric efficiency prior to the onset of severe cavitation limitations. Furthermore, Figure 9(b) illustrates the baseline pressurization of the Vane Chamber (VC) and Lip Chamber (LC) in one complete cycle, behaving as expected. The model perfectly captures the inherent transient delivery features and outlet flow behavior, aligning with the periodic shape and amplitude of the experimentally measured pressure ripples. This baseline alignment confirms that integrating the upstream casing restrictions via characterized orifice coefficients does not disrupt the fundamental physics of the internal displacement cycle. While the original model assumed ideal filling and thus overpredicted volumetric efficiency at high speeds, the current extended model successfully captures the efficiency drop-off due to casing restrictions, aligning more closely with the observed physical limits, as detailed in the following sections.

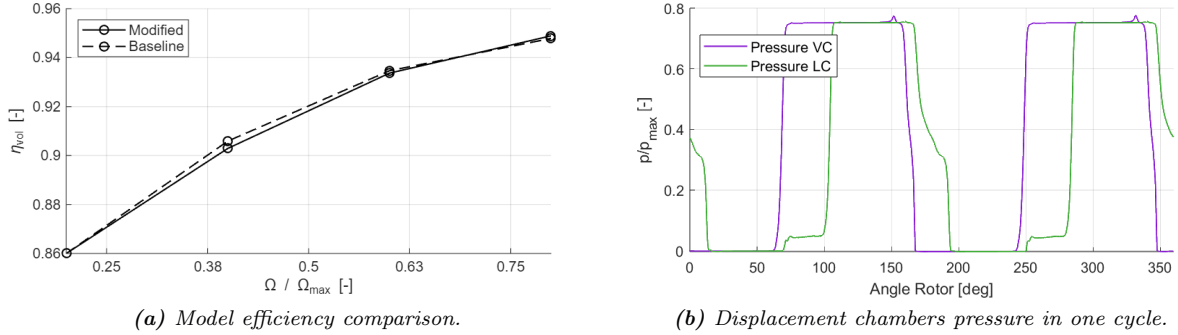


Figure 9: Left figure shows that the modified simulation model matches the original baseline model predictions. Right figure shows the displacement chamber VC and LC pressurization for one cycle.

4.2 Chamber Cavitation

While the baseline model successfully predicted performance at standard speeds, it was designed to capture the cartridge unit performance without considering the supply path in the casing. By updating the fluid circuit to consider the true geometry of the casing flow paths, the extended simulation successfully captures the severe fluid density drops and pressure restrictions occurring prior to the main displacement chambers.

Leakage at the vane tip in contact with the cam-ring varies as the chamber enters and exits the high-pressure region. Increasing the operating RPM alters the pressure distribution in the EHL contact, varying the gap height. These discrepancies closely correlate with the inter-chamber leakages that dictate the unit's final volumetric efficiency.

As the rotational speed of the pump increases, the displacement chambers demand a significantly higher mass flow rate during the suction phase. In the physical unit, fluid entering the primary casing inlet must navigate a tortuous path to reach the rear of the cartridge, traversing from the inlet chamber to the base chamber, and finally squeezing through a thin rectangular slot to reach the back chamber. The lumped-parameter model dynamically calculates the pressure drop across these geometric bottlenecks. At elevated speeds, this severe hydraulic restriction causes the localized pressure within the casing channels to plummet.

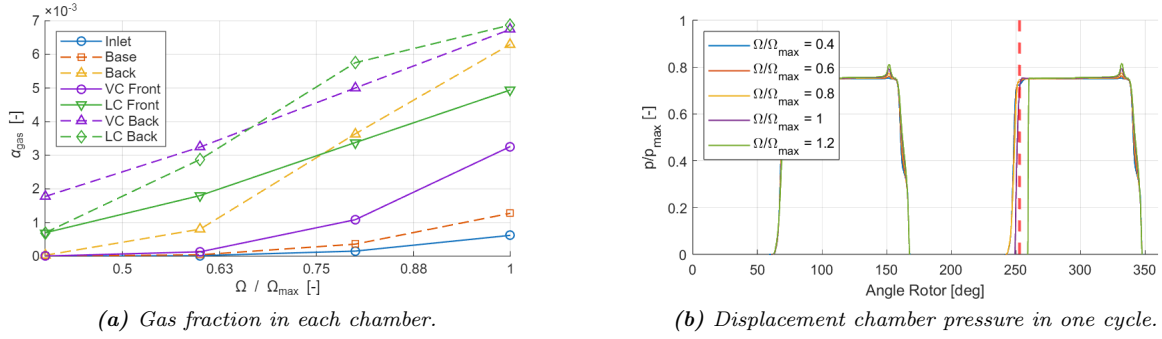


Figure 10: Lumped-parameter simulation of the internal casing path illustrating that the chambers are cavitating (a), and the delayed pressurization (b) confirming this event.

The simulation reveals a critical phenomenon, visualized in Figure 10(a): as the pressure drops below the fluid's saturation and vapor pressures, the localized fluid density and bulk modulus decrease drastically. This triggers premature aeration and vaporization within the casing itself—effectively a "pre-cavitation" event occurring well before the fluid even crosses the suction port boundary into the Vane Chamber (VC) and Lip Chamber (LC). Figure 10(a) presents the simulation results showing the percentage of free gas and vapor across the different chambers. The back displacement chamber and its corresponding upstream supply chambers indicate significant cavitation occurring at much earlier phases of operation compared to the adequately supplied front chambers.

By explicitly tracking these localized fluid properties throughout the entire unit, the model directly links the upstream casing restrictions to the onset of severe suction port cavitation. The simulation demonstrates that the displacing chambers are unevenly filled. Because the fluid must travel through the highly restrictive thin port to reach the back chamber, the second suction port experiences high cavitation effect.

This physical limitation is further illustrated by evaluating the pressure within the displacement chambers at various normalized operating speeds ($\Omega/\Omega_{\text{max}}$). As seen in Figure 10(b), the VC experiences a significantly delayed pressurization when operating at higher rotational speeds. This delay at the second inlet is a direct consequence of the upstream pre-cavitation choking the supply flow, coupled with the chamber's own localized cavitation. As the pump operating RPM increases, the restrictive upstream casing pathways choke the flow, forcing the displacement chambers drawing from this rear port to ingest a vapor-heavy mixture, leaving them severely under-filled compared to the front chambers.

5 Geometry Modification

To mitigate the cavitation effects identified in the baseline unit, an updated geometry was proposed and analyzed. The geometries were iteratively modified to expand the cross-sectional areas within the allowable manufacturing constraints, rather than using a formal optimization algorithm. The original restrictive casing channels were primarily a result of conservative manufacturing tolerances and spatial constraints within the pump housing. Expanding these channels does not compromise the structural integrity of the casing, provided the minimum wall thickness requirements are maintained.

The fundamental mechanism driving the pre-cavitation is the excessive localized pressure drop caused by flow constriction. Based on the preliminary CFD investigations, the primary casing modifications included geometrically expanding the internal casing channels—specifically increasing their respective radii and widening the gap between the cartridge and the casing wall. These physical modifications effectively increase the cross-sectional area of the supply pathways.

According to the principles of fluid dynamics, expanding the cross-sectional area proportionally reduces the localized fluid velocity. Consequently, the dynamic pressure losses are minimized, allowing the absolute pressure within the casing channels to remain above the fluid’s saturation and vapor pressures. By characterizing these newly expanded geometries with CFD, updated orifice discharge coefficients (C_d) were derived and integrated into the lumped-parameter fluid circuit. The significance of these updated characteristics is most accurately captured by the model’s ability to simulate cavitation dynamically.

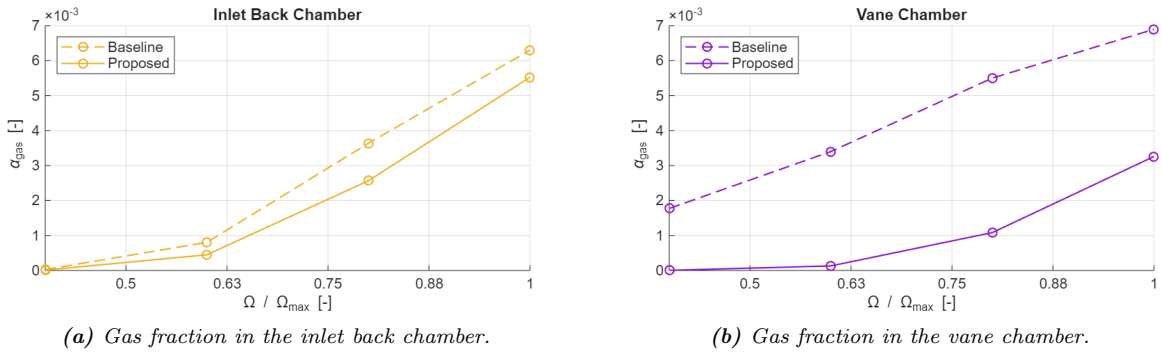


Figure 11: Simulation results comparing the gas fraction in the displacement chambers, demonstrating significantly reduced aeration and cavitation with the optimized casing design.

As shown in Figure 11, simulating the expanded geometry with the new orifice characteristics demonstrates a drastic reduction in the free gas percentage within the back chamber compared to the baseline model. By minimizing the upstream restriction, the fluid flow reaching the second suction port maintains a density equivalent to that of the front chamber, successfully mitigating the severe pre-cavitation effects.

These geometric modifications successfully suppress the premature release of dissolved air and the vaporization of the hydraulic fluid. Consequently, the simulation predicts a significant increase in volumetric efficiency at elevated operational speeds. In the baseline experimental data, the pump’s operation was strictly limited, with distinct signs of efficiency decreases beginning at mid-range speeds due to incomplete filling. However, with the newly proposed casing design, the simulation indicates that the updated design will allow the unit to reduce cavitation, enabling operation at higher speeds. These numerical predictions are scheduled for experimental validation.

6 Experimental Validation

6.1 Test Rig Setup

To validate the simulation predictions and quantify the performance improvements of the optimized fluid supply paths, an experimental test rig built at the authors' research lab was utilized. The hydraulic circuit is designed to precisely control the operating conditions and measure the steady-state performance of the prototype. The layout of the test rig is illustrated in Figure 12. A needle valve was used as a loading element to set the desired pressure at each operating shaft speed. Additionally, a sharp edge fixed orifice and a calibrated pipe (with a constant diameter of 16 mm and length of 200 mm) were placed at the pump outlet; this setup allowed for the establishment of standing waves for pressure ripple tests. The prime mover drives the pump while absolute pressure sensors monitor the inlet and outlet pressures, which is critical for identifying the onset of cavitation vacuum. A flow meter measures the delivery flow rate, and temperature sensors track the thermal state of the fluid. Prior to testing, an optical profilometer (Bruker NPFLEX-1000) was used to measure the critical surface features of the components, such as the roughness and radius of curvature of the vane-lips. The details and accuracies of the sensors used are listed in Table 3.

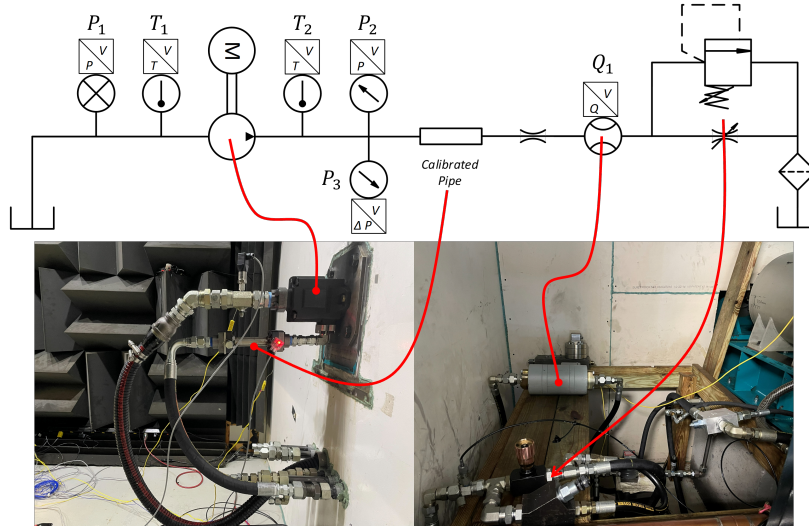


Figure 12: Schematic drawing of the experimental test rig used to evaluate the pump's performance and photograph of the physical experimental test rig installation.

Table 3: Sensor specifications.

Sensor	Type	Description
T_1, T_2	Thermocouple	OMEGA, scale: 0 °C to 200 °C, 0.4% FS accuracy
P_1	Strain gauge	WIKA, scale: 0–1.6 barA, 0.125% FS accuracy
P_2	Piezoelectric (quartz)	KISTLER, scale: -1–10 bar, 13 kHz natural frequency, 0.8% FS accuracy
P_3	Piezoresistive	WIKA, scale: 0–345 bar, 0.5% FS accuracy
Q_1	Gear Flow meter	VSE RS400, scale 1–400 l/min, 0.25% measured value accuracy

6.2 Results

The steady-state flow performance of the newly proposed geometry was experimentally evaluated across its operational envelope and directly compared against the physical baseline design.

As illustrated in Figure 13a, the flow rate of the optimized design scales proportionally with the shaft speed, demonstrating improved delivery characteristics at higher operating conditions compared to the base-

line unit. When evaluated in terms of volumetric efficiency (Figure 13b), the experimental validation indicates that the proposed design maintains higher performance levels across the elevated speed range. Whereas the baseline unit experienced a reduction in efficiency attributed to incomplete chamber filling, the optimized casing flow paths effectively mitigate this suction restriction. Consequently, the modified design sustains high volumetric efficiency up to the maximum operating speed, verifying the success of the geometric improvements.

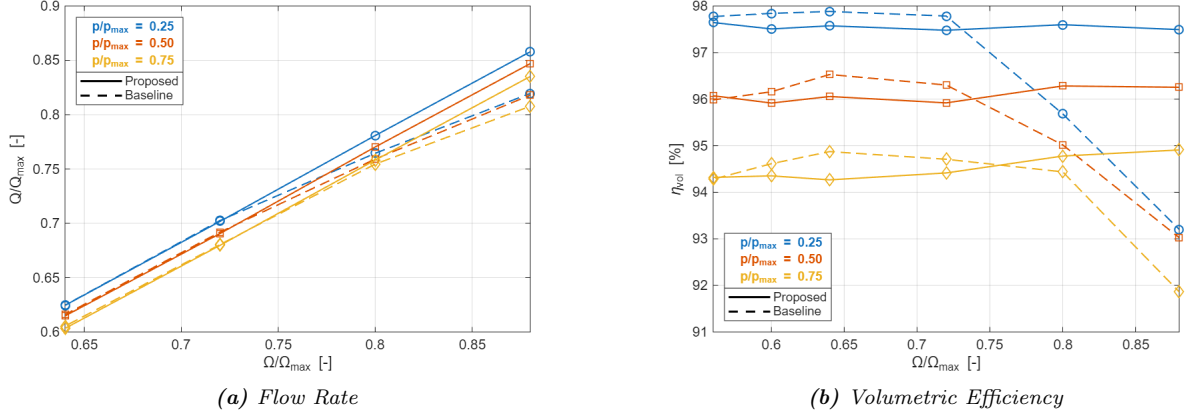


Figure 13: Experimental validation comparing the steady-state flow rate (a) and volumetric efficiency (b) between the baseline and optimized pump designs.

7 Conclusion

This paper details the further development of a multi-physics numerical model for balanced twin-lip vane pumps to include the previously unmodeled casing fluid supply paths. Preliminary CFD simulations were used to define the characteristics of the orifices and chambers, allowing the model to capture destructive cavitation effects in the casing. The baseline numerical model was successfully validated against existing experimental results to confirm its fidelity. Subsequently, geometry updates were proposed and characterized via CFD, resulting in a newly designed parameter set that demonstrates improved supply flow capability. By physically expanding the casing channels and further updating the cam-ring orifice, flow velocities and dynamic pressure drops were minimized. Simulation results predict drastically improved performance of the pump at high speeds, resolving the premature cavitation limitations observed in the baseline unit. These numerical predictions and geometric modifications are confirmed through experimental validation, which proves the optimized design's ability to maintain high volumetric efficiency and linear flow delivery at elevated operational speeds.

References

- [1] Z. Mistry, A. Vacca, S. K. Uppaluri, “Modeling and experimental validation of twin lip balanced vane pump considering micromotions, contact mechanics, and lubricating interfaces,” *Simulation Modelling Practice and Theory*, vol. 133, p. 102914, 2024.
- [2] A. Vacca, G. Franzoni, *Hydraulic Fluid Power*, Wiley, 2021.
- [3] Z. Yan, et al., “Structural design and cavitation performance research of single-acting sliding vane pump under high rotational speed and petroleum medium,” *Proc. IMechE Part C: J. Mechanical Engineering Science* (Sage Journals), vol. 239, no. 1, 2023.
- [4] T. Lobsinger, T. Hieronymus, and G. Brenner, “A CFD investigation of a 2D balanced vane pump focusing on leakage flows and multiphase flow characteristics,” *Energies*, vol. 13, no. 13, 2020.
- [5] U. Stuppioni, A. Suman, M. Pinelli, and A. Blum, “Computational fluid dynamics modeling of gaseous cavitation in lubricating vane pumps: An approach based on dimensional analysis,” *Journal of Fluids Engineering*, vol. 142, no. 7, p. 071206, 2020.
- [6] K. Suzuki, Y. Nakamura, S. Yakabe, H. Watanabe, and K. Nakamura, “Characteristics prediction of vane pump by CFD analysis,” *KYB Technical Review*, vol. 53, pp. 8–15, 2016.
- [7] M. Rundo, G. Altare, and P. Casoli, “Simulation of the filling capability in vane pumps,” *Energies*, vol. 12, no. 2, 2019.

Cylinder Block Deformation as Indicator for Cylinder Pressure in Axial Piston Units: A Novel Method Using Strain Gauges

**Karl Stübig, Luca Löding, Frank Geuer, Lars Leonhard,
Dr.-Ing. Matthias Liermann, Sumeru Srinivasamurthy, Carsten Fiebing,
Dr.-Ing. Robert Rahmfeld**

*Danfoss Power Solutions GmbH & Co. OHG, Krokamp 35, 24539 Neumünster
mliermann@danfoss.com*

Abstract

Cylinder bore pressure prediction is the foundation for designing timing notches in valve plates of axial piston units. The timing notches in a valve plate affect not only efficiency of the axial piston unit, but also its controllability and NVH characteristics. It is state of the art to use cylinder bore pressure measurement for verification of simulation results. With a telemetry system, the bore pressure data can be retrieved from the rotating unit to a stationary data acquisition system. This data can be used for comparison with simulation results, to support the optimization of different timing designs of the valve plate and operating conditions.

The feasibility of integrating a pressure sensor into the rotating kit of a production unit depends on space constraints between housing and kit and is further complicated by routing of wires through the shaft to the telemetry unit. As the units become smaller, it is increasingly difficult, if not impossible, to fit the pressure sensor and cables securely.

This paper investigates a novel approach to use a strain gauge applied to the outside of the cylinder block as an indication of the pressure inside the cylinder block. This approach addresses a major obstacle to the experimental investigation of smaller size displacement units, since the strain gauge is mounted flat on the surface of the block in an area, where most deformation is to be expected.

Measurement results show the correlation of the strain gauge signal in comparison with a conventional pressure measurement. The signal is affected by additional parameters, such as shaft speed and angle, line delta pressure, and temperature.

Keywords. Telemetry, simulation validation, data acquisition

1. Introduction

Hydrostatic pumps and motors convert mechanical energy into hydraulic energy and vice versa. In a swash plate type axial piston pump, see schematic in Figure 1, the cylinder block is driven by the drive shaft. The pistons located in the cylinder bores perform a translational movement when sliding with their slippers on the inclined swash plate. By changing the displacement angle β , the unit displacement volume can be adjusted. In pumping mode, during half of one revolution, the pistons move from inner dead point (IDP) to outer dead point (ODP), while fluid is drawn into the displacement chamber from the low-pressure port. Moving in the opposite direction from ODP to IDP during the second half of the revolution, the pistons push out the fluid from the displacement chamber into the high-pressure port. The unit used during this investigation has an odd number of pistons ($k = 9$), resulting in a periodic change of either four- or five-cylinder bores connected to high pressure.

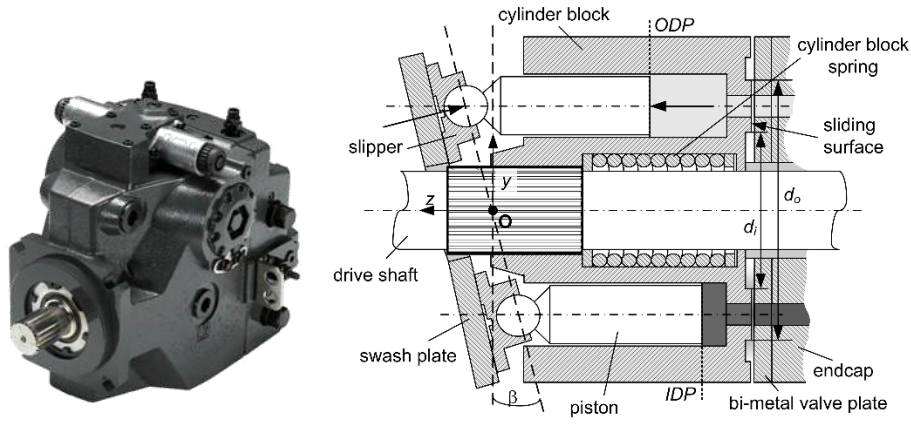


Figure 1 Principle of a swash plate type axial piston hydrostatic pump

The transient pressure in the cylinder bores of axial piston pumps is the result of several flows, see Figure 2. They are defined by the kinematics of the moving mechanical parts which drive the compression and relaxation of the fluid and cause the opening of flow passages between different fluid cavities within the pump and the connected hoses. The most important flow passages defined by the moving mechanical parts are the overlap between cylinder block kidneys and the valve plate [1]. Minor flow passages that influence the cylinder pressure exist in form of leakage paths across valve plate sealing lands, piston and bore bushing, and through the hydrostatically balanced slippers.

The valve plate is the crucial component connecting the rotating cylinder block with the stationary pump endcap. It determines the timing and shape of the pressure transition. A steep transition without cross-port leakage is efficient. However, with some degree of cross-port leakage permitted, the transition is smoother and leads to a quieter pump. The swash plate moment is directly related to the cylinder pressures. The swash moment $M_{S,x}$ around the x-axis can be written as

$$M_{S,x} = \frac{R}{\cos^2 \beta} \sum_{i=1}^9 F_{piston,i} \cos \varphi_i \quad (2.1)$$

where R is the pitch radius, β the swash plate angle, $F_{piston,i}$ the axial force exerted by the i -th piston, and φ the shaft rotational angle [2]. The axial piston force $F_{piston,i}$ is primarily dependent on the cylinder pressure. Secondly it is composed of inertia and friction forces.

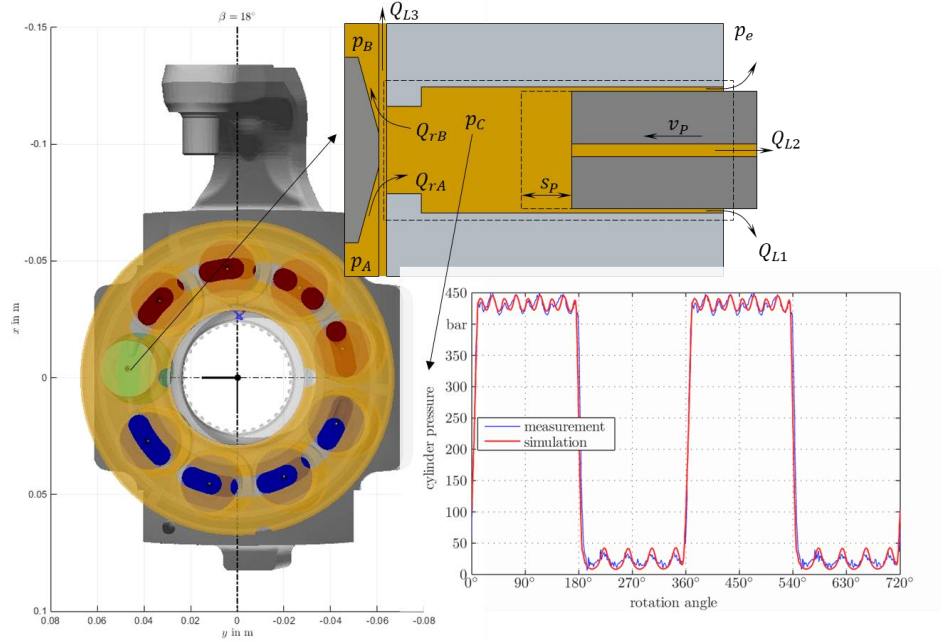


Figure 2 Transient cylinder pressure and control volume with in- and outbound flows

Lumped-parameter system simulation can be used to predict the transient pressure signal in axial piston units. It is an important tool for the design engineer to optimize the valve plate area characteristic [3] [4]. The simulation gives insight not only into the transition of cylinder pressures, but also into flow ripples, volumetric efficiency and swash plate moments. This type of simulation is state-of-the-art and basic equations are common knowledge. However, there are frequently cases in practice where model predictions do not align with measured performance. To find the root cause for this mismatch, experiments are needed that give insight into the transient nature of the cylinder pressure. This paper presents a novel experimental method of using a strain gauge to measure the strain caused by deformation of the cylinder block to get a high-frequency signal that correlates with cylinder pressure. The main advantages of this method are the reduced cost of the sensor, speed and ease of sensor application and the reduced required installation space. With this method, cylinder pressure measurement in small displacement pumps becomes feasible where it was not possible before.

1.1. Pressure measurement of cylinder bore pressure in axial piston units

The study of hydraulic-mechanical performance and the transient nature of the cylinder pressure of axial piston units have a long history [5]. The cylinder pressure is of special interest because it determines directly or indirectly several performance characteristics.

Kleist [6] and Ivantysynova [7] used specially designed single-piston test setups to investigate temperature and pressure distributions in the gap between piston and cylinder. These setups used inverted kinematics so that sensors did not have to be part of a rotating unit. Inverted kinematics means that the kinematic chain of the axial piston unit is reversed compared to conventional operation. In this configuration, the rotating group is held stationary while the surrounding components reproduce the relative motion. This allows all sensors to remain in a stationary reference frame, eliminating the need for integration into rotating components and simplifying measurement setup and signal acquisition.

But cylinder pressures in rotating cylinder blocks have also been measured, for example in Olems [8] and Helgestad et al. [9], who modified a commercial pump by fitting a miniature piezoelectric pressure sensor into the cylinder block. Ivantysynova et al. [10] designed a special tribo-pump with the ability to measure cylinder pressure and piston friction force in the rotating cylinder block. Challenges of fitting miniature pressure sensors into the cylinder block are the limited space in the cylinder block to fit sensor and cables. Another challenge is that the bore hole that connects the pressure sensor with the cylinder creates additional dead volume of the piston or changes the flow pattern and therefore influences the measurement. Bergemann [11] addressed this challenge with a custom-applied manganin wire inside the cylinder bore. Manganin is a trademarked name for an alloy that changes its electrical resistance with temperature as well as with hydrostatic pressure. With a second measurement coil it is possible to compensate for the temperature dependency and with appropriate amplification to produce a high-responsive pressure signal.

Experimental work is still important today alongside simulation efforts. More recently, Ivantysyn et al. [12] [13] developed a complex measurement setup which included 24 temperature sensors at 20 distinct positions and a pressure sensor in one of the chambers of a rotating cylinder block to confirm a temperature/efficiency relationship predicted by simulation. They concluded that while simulation results were largely confirmed in target operating conditions, new insights were gained in border-line conditions, which are hard to predict. Wieczorek et al. [14] similarly conclude that simulation tools cannot replace experiments but are another approach to support development. The literature survey shows that the experimental effort to measure physical quantities in the rotating cylinder block is considerable, to the extent that only in two published studies [8] [9] sensors could be placed into series hardware; however, even in this case, significant modifications to the hardware were required to accommodate sensors, connectors, cables, amplifiers, and the telemetry unit.

Pressure sensors have limitations regarding installation space, cost, and durability in harsh operating conditions. This study explores an alternative approach to measure cylinder pressure by a surface-attached device. We propose to use a strain gauge to sense the surface tension caused by deformation of the cylinder block and to derive the cylinder pressure by correlation. The main objective of this work is to develop and validate a measurement setup capable of tracking pressure changes based on cylinder block deformation and to identify the key challenges and limitations of this method. Experimental results demonstrate that the method reliably captures pressure trends and dynamic effects, making it a promising tool for both research and industrial testing, particularly in small-displacement axial piston units.

2. Methodology

The proposition of this paper is that the deformation of the cylinder block can be used as indication of the pressure inside the cylinder bore. Figure 3 shows a FEM analysis of the deformation of a cylinder block of a Danfoss H1P 280 pump with 420 bar pressure applied in static condition. The pressure load is applied uniformly across the cylinder surface from the dead volume region near the block kidneys up to the piston in its current stroking position. The number of high-pressure bores corresponds to the instantaneous porting condition of the machine. In the current condition 4 cylinders are connected to high pressure while 5 cylinders are connected to low pressure. The deformation leads to surface strain which can be measured with strain gauges applied close to the bottom of a particular cylinder. But other parameters may have an influence on the measurement as well, such as rotational speed and angle, line delta pressure, and temperature. For example, the lateral force of the piston that is supported in the cylinder bore adds to the block deformation. It is therefore expected that the strain gauge signal also depends on the block rotational angle. The influences of these parameters is shown in chapter 3 and discussed in chapter 4.

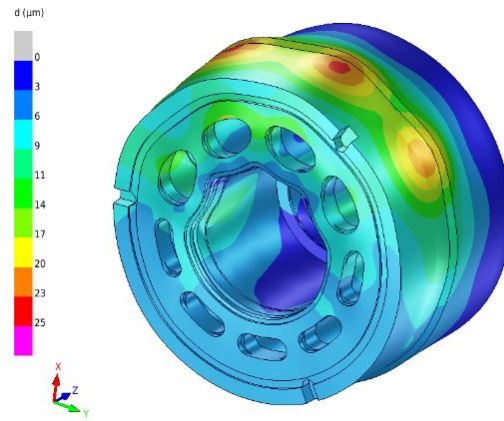


Figure 3 H1P280 cylinder block deformation at $p_{HP} = 420$ bar and $p_{LP} = 20$ bar

2.1. Measurement program

Goal of the measurement program is to verify the proposition of this paper, that a strain gauge applied to the lateral surface of the cylinder block can deliver a signal that corresponds to the cylinder pressure. To do this, it is necessary to apply a pressure sensor to the cylinder chamber for verification and test several operating conditions. Table 1 lists the conducted experiments to investigate the influence of speed, pressure, displacement and temperature on the strain gauge signal.

Table 1: Test plan

Test run	Purpose	Temp [°C]	Speed [RPM]	Δp [bar]	swash plate angle [%]	Figure
1	Proof of function, pressure steps	34-54	<0.1	0-400, steps	0%	6
2	Speed ramp I	53-58	200-1500	100	39%	7
3	Speed ramp II	49-54	200-1500	0	0%	8
4	Pressure ramp	59-60	1500	0-450, ramp	42%	9
5	Influence of swash plate angle I	36-42	1	200	0%	10
6	Influence of swash plate angle II	31-35	1	200	100%	10
7	Influence of temperature	25-90	1000	250	50%	11

2.2. Test setup

A pump with large displacement (H1P 280) is selected as test unit to give sufficient space for sensors and cables, see Figure 4. Pressure, strain gauge and temperature signals are transferred from the rotating cylinder block to a stationary data acquisition system via a telemetry unit. An IMC MTP-NT telemetry unit is used for this purpose. Several components of the pump had to be reworked, and some new parts had to be designed and added to create this measurement setup. A type J thermocouple and Mesurex PM6 Series pressure sensor were mounted into threaded bore holes in such a way as to get as close to the bottom of the cylinder as possible to avoid influence on the chamber dead volume. Two HBK VA71K1.6/350_E strain gauges were applied onto two adjacent cylinder chambers. Two strain gauges were applied to have signal redundancy in case one is damaged and to investigate whether the pressure rise in one cylinder may affect the signal of the adjacent cylinder strain gauge. All cables are glued into a groove that was manufactured into the cylinder block for protection from mechanical damage caused by slushing in the oil-filled case as the cylinder block rotates. The cables are then guided towards the shaft and routed through a hole in the drive shaft, across the shaft coupling which connects the charge pumps and further on through the custom-made telemetry into the telemetry unit.

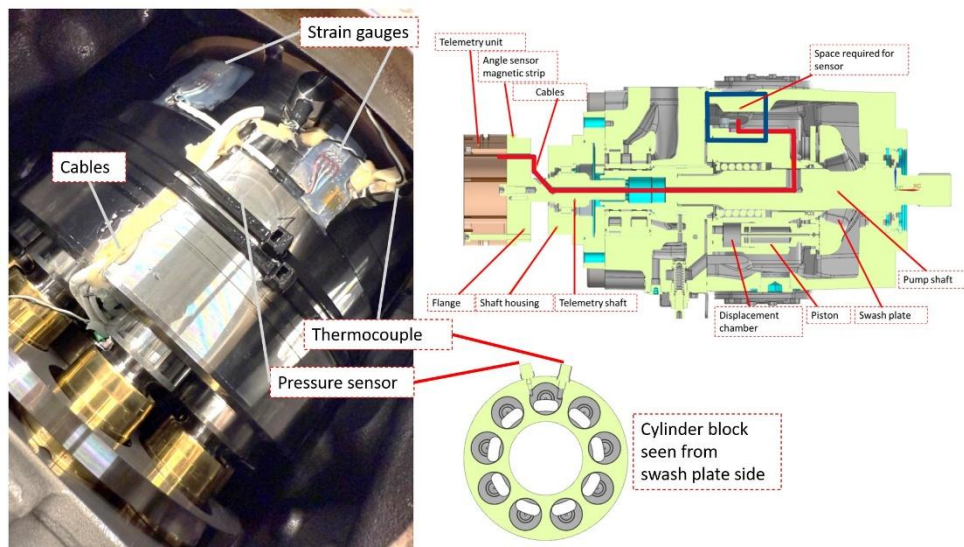


Figure 4 Sensor integration in H1P 280. Two strain gauges, a pressure sensor, and a thermocouple are fitted onto the cylinder block with cables that are guided through a customized hollow shaft to a telemetry unit.

Figure 5 shows the assembled pump with telemetry unit on the test bench. The telemetry unit houses four integrated signal amplifiers and A-D converters for the two strain gauges, the pressure sensor and thermocouple. Their digital signals are transmitted as pulse code modulated bit stream signals via inductive transmitter to the pick-up head. Power to the sensor modules is provided via induction from the power head. A rotational angle read-head that reads the pump rotational angle from a magnetic scale is mounted to the top of the telemetry shaft housing. Pressure and temperature sensors are connected to hydraulic return and feed line ports.

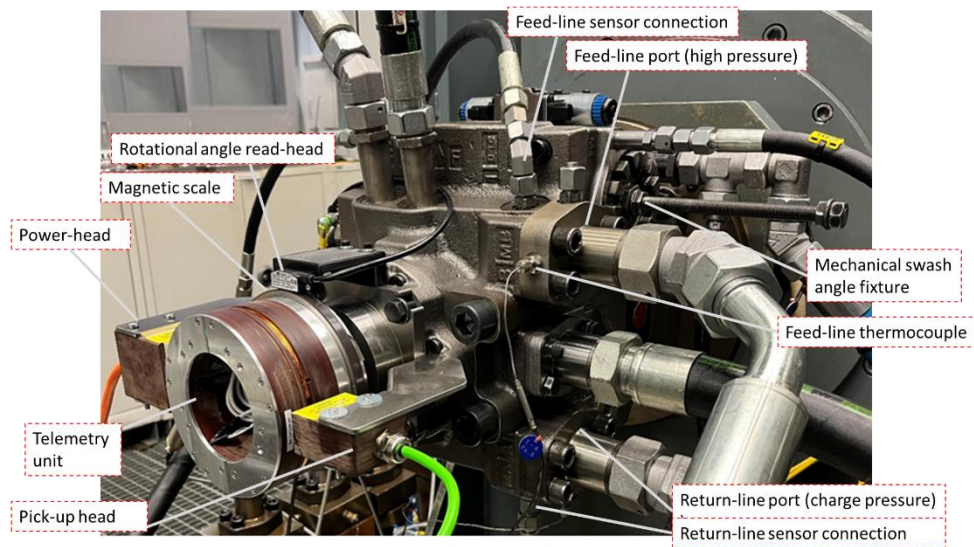


Figure 5 Pump under test with telemetry unit, rotational angle sensor and further sensors connected to the pump port connections

3. Results

The following test results show to which extent the strain gauge signal corresponds to the cylinder pressure. The main take-away is that the strain gauge is primarily sensitive to cylinder pressure but picks up other influences as well, which will be hard to compensate for in practice. Some potential causes and what this sensitivity to secondary effects means for the applicability of this measurement principle will be discussed in chapter 4.

3.1. Proof of function

Figure 6 shows how cylinder pressure measurement and strain gauge signal correlate after an initial static calibration procedure. In the test, the shaft angle is held fixed, and the pump is pressurized externally in steps from 50 to 400 bar. The tank oil temperature is controlled but the cylinder temperature is not kept at a constant level because of a lack of fluid exchange in the cylinder. It varies between 34 and 54 °C.

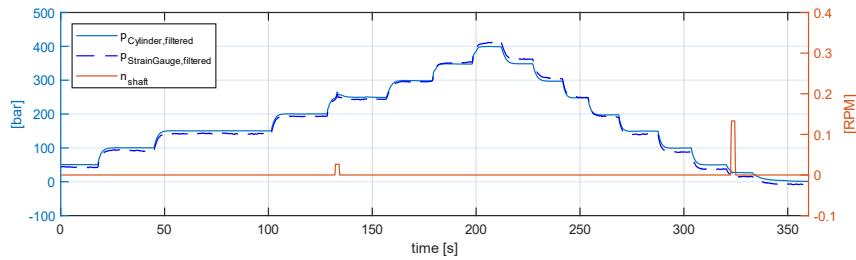


Figure 6 Comparison of cylinder and strain gauge pressure signals. Pump shaft is held fixed; pressure is applied externally in steps; temperature varies between 34-54 °C.

The correlation between pressure and strain gauge signal is qualitatively given. There is a quantitative difference in the strain gauge signal compared to the pressure sensor signal as the pressure steps up and down. For example, between $t = 190$ s and $t = 220$ s the pressure sensor signal differs only 1 bar, whereas the strain gauge signal differs more than 12 bar.

An unexpected artefact in the pressure signal is observed at $t = 133$ s. Both, pressure sensor and strain gauge signal spike briefly and then abate to their previous level. The explanation for this can be given as follows: The applied pressure impacts the load on the servo system. Due to swash plate moments, the swash plate angle experiences a sudden change. The pump displacement causes torque to be applied by the pump and a slight motion of the shaft against the fixture. The motion also impacts the cylinder pressure. This is picked up by the pressure sensor and the strain gauge alike.

3.2. Influence of shaft speed, load, and displacement angle

Figure 7 and Figure 8 show the cylinder pressure and strain gauge signals, each over one rotation at different speeds. In Figure 7 the pump is set to 40% displacement and against 100 bar line delta pressure. Figure 8 shows results for zero displacement and no-load condition.

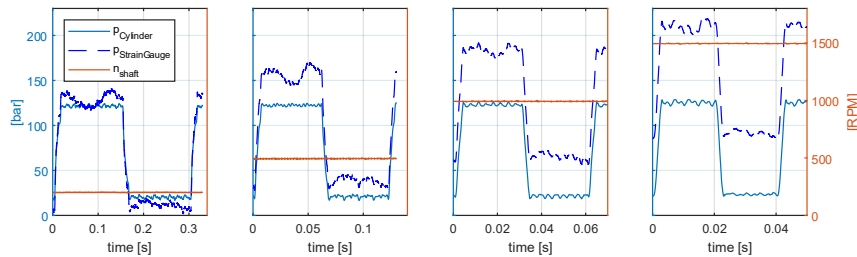


Figure 7 Comparison of cylinder pressure and strain gauge signal at $n = [200, 500, 1000, 1500]$ RPM; Line delta pressure 100 bar, 7° displacement, 55 °C

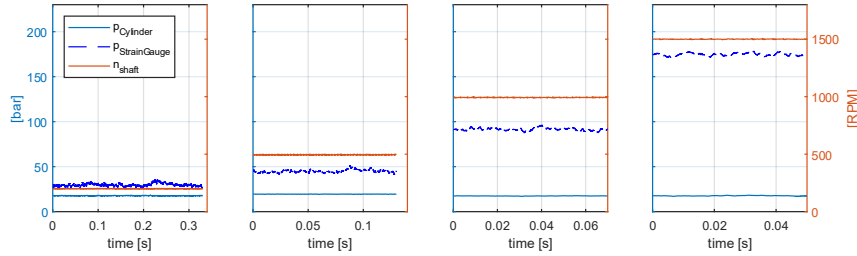


Figure 8 Comparison of cylinder pressure and strain gauge signal at $n = [200, 500, 1000, 1500]$ RPM; Line delta pressure 0 bar, 0° displacement, 52°C

The dynamic response is excellent, but quantitatively the signal is hard to interpret. Evidently, shaft speed strongly influences the strain gauge signal. To a lesser extent, line delta pressure and displacement impact the signal. Where the pressure signal reports a plateau between the high- and low-pressure transitions, the strain gauge signal bends inward. It seems that the signal is shaft angle dependent.

In Figure 9 the effect of different levels of line pressure is shown for fixed speed. The strain gauge high- and low-pressure plateaus spreads outward as the line delta pressure increases.

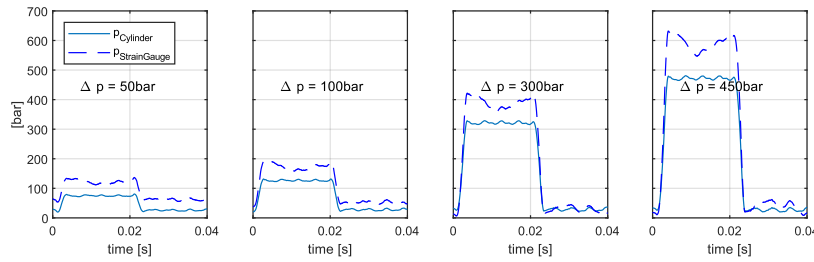


Figure 9 Comparison of cylinder pressure and strain gauge signal at fixed speed 1500 RPM and different line delta pressure $\Delta p = [50, 100, 300, 450]$ bar, 7.5° displacement

Figure 10 shows the influence of shaft angle and displacement. It shows the result of two tests in one diagram. In both tests the shaft is driven precisely with 1 RPM and the line delta pressure was held constant at 200 bar. The comparison shows the strain gauge signals over one rotation for both cases, as the displacement is changed between 0% and 100% . The influence of pump displacement angle is evident, but comparatively minor. The time for one rotation is 60 seconds, therefore the periodic patterns in the high-pressure plateau are no measurement artefacts but clearly show the effect of adjacent pistons entering and exiting the high-pressure kidney.

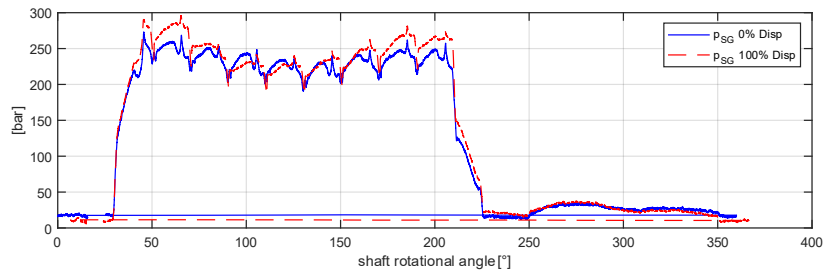


Figure 10 Influence of swash plate angle on strain gauge signal; strain gauge signal plotted over shaft rotational angle for 0 and 100% displacement at 1 RPM and 200 bar line delta pressure.

3.3. Influence of temperature

To investigate the influence of temperature alone, the pump is operated at constant speed against a fixed load. The cooling system is switched off, and the circuit allowed to heat up. At $t = 2271$ s the cooling system is activated again and the temperature decreased. The strain gauge and cylinder pressure signals are shown by their envelopes because they oscillate with high frequency between low and high pressure. Also plotted are the line pressures p_B , p_A at the inlet and outlet of the pump. They are constant throughout the test, whereas both, the cylinder sensor and strain gauge signal vary over test duration. A correlation with temperature cannot be explained easily for both, the cylinder pressure sensor and the strain gauge.

The pressure sensor signal should be compared to the inlet temperature because the cylinder is cooled by the incoming flow. The pressure signal drops at the start of the test and rises later. But it stays constant during much of the test, where the inlet temperature clearly changes. The strain gauge signal should be compared to the case temperature because it is mounted to the surface of the cylinder and in contact with case oil. However, it does not correlate in trend with the case temperature at all. It rises in the beginning, drops during the middle but starts increasing again before the cooling is switched on. At the moment of cooling down the oil, the strain gauge signal remains roughly on the same level.

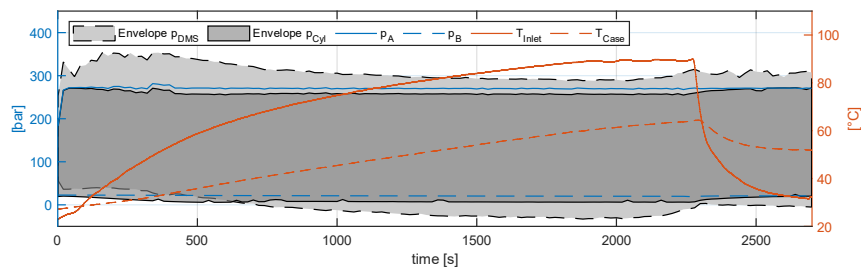


Figure 11 Influence of temperature on signals from cylinder pressure sensor and strain gauge. Speed is constant at 1000 RPM; line delta pressure is constant at 250 bar; pump displacement is 50%.

4. Discussion

The measurement results show high correlation between strain gauge and pressure signal with no noticeable delay. However, the signal is influenced strongly by secondary effects summarized in Table 2.

Table 2: Influence factors on strain gauge signal

Effect	Influence	Quantification
Shaft speed	Strongly positive	0.04 - 0.16 bar/rpm
Line delta pressure	Strong, different for high and low pressures	-0.2 - 1.5 bar/bar
Displacement	Moderate, different for high and low pressures	-0.07 - 0.2 bar/percent
Shaft angle	Pressure dips inward between transitions	10-30%
Temperature	Strong, incongruent	

No simulation was done to date that could confirm the root cause for each of the influence factors. At this point we can only hypothesize.

With regards to the influence of shaft speed, see Figure 7 and Figure 8, it is quite hard to imagine that centrifugal forces on the pistons cause deformation of the cylinder block bore that is picked up by the strain gauge. This is still a matter of investigation.

The influence of line delta pressure on the strain gauge signal is not easily explained. Figure 9 shows that the signal during the high-pressure phase is affected differently than during the low-pressure phase. By offsetting the strain gauge signal by -30 bar and multiplication with a gain of 0.8 it is possible to improve, but not to eliminate this effect, as Figure 12 shows. This gain and offset would be dependent on other parameters such as speed and temperature as can be easily shown.

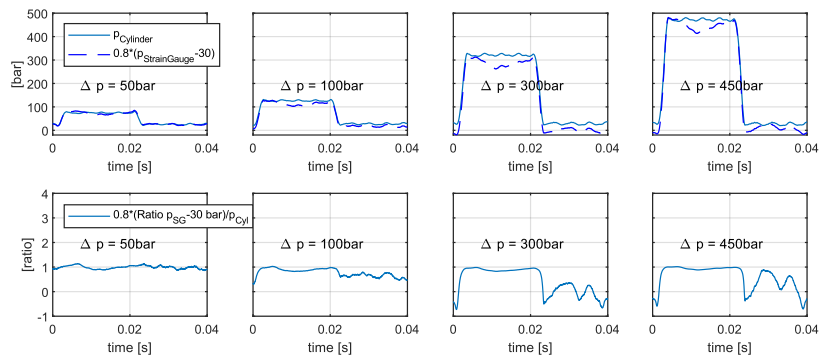


Figure 12 Results in Figure 9 with alternative scaling of strain gauge signal.

Scaling $y = 0.8(x - 30)$ shows improved matching.

Top row: absolute comparison; Bottom row: relative comparison

An interesting observation is that the strain gauge signal bends towards lower values during the high-pressure phase as the piston moves from outer dead center to inner dead center. The underlying cause of this behavior, however, is not yet fully understood.

The temperature influence is apparent, see Figure 11. However, a mapping to any of the temperature readings could not be found. This indicates that the effects of several parameters are not independent of each other.

The study shows that the strain gauge signal cannot be simply used as a pressure measurement, but certainly as an indication. It is reasonable to expect that some level of compensation for speed and pressure can be achieved through state-dependent scaling and offsets. But this scaling would not cover all influences, such as temperature. For speed and pressure, it would likely be product-specific and therefore not very useful.

Is it, after all, of any value to use the strain gauge signal as indication of the cylinder pressure? It was noted in chapter 3.2 that the high-frequency content of the strain gauge signal follows the actual pressure very closely. To illustrate this, Figure 13 shows the measurement of Figure 7 with a high-pass filter applied. It removes the low-frequency biases below 20 Hz. The bottom row plots of Figure 13 show the difference between strain gauge and pressure sensor signals. During the high to low and the low to high pressure transitions, the signal amplitude is high, while the error low. In other words, the strain gauge signal gives a good indication about what happens dynamically during the transitions. The high frequency content of the strain gauge signal by itself shows good correlation.

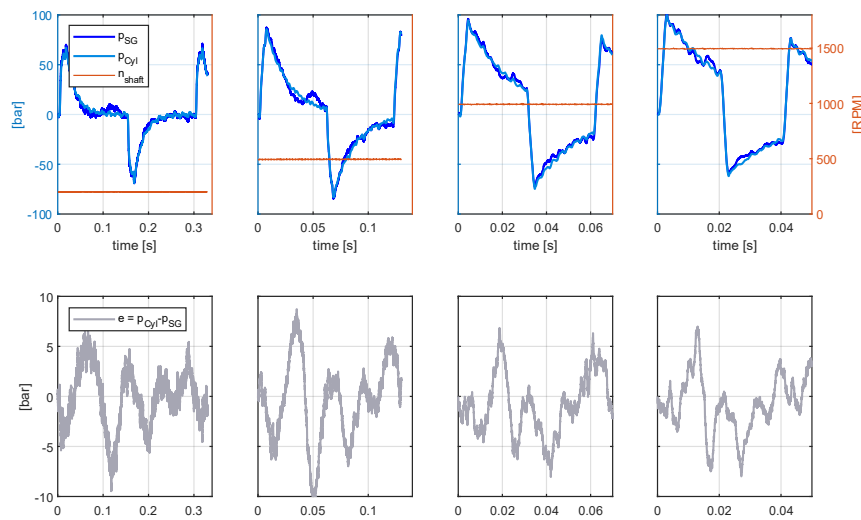


Figure 13 Comparison of cylinder pressure and strain gauge signal after high-pass filtering.
Top: High-pass filtered data from Figure 7. Bottom: Difference between both signals.
Error small compared to signal magnitude during transitions.

5. Conclusion and Outlook

The measurement of physical signals from rotating parts requires some form of telemetry unit. To measure the cylinder pressure in an axial piston pump, miniature sensors, delicate wiring, and connecting in very narrow spaces are required. This study shows that a strain gauge mounted to the outside of a cylinder block is sensitive to pressure changes, but also

to shaft speed, shaft angle, line delta pressure, displacement and temperature. We conclude that it cannot be used as a replacement for a pressure sensor in a general sense. However, the high-frequency content of the strain gauge signal corresponds very well with the cylinder pressure and may be used in cases, where the focus of study is on the fast high to low and low to high pressure transition. Also, for verification of simulation models this type of measurement is useful because it is the transient response between low and high pressure in the cylinder bore that needs validation, not the plateau phases.

6. References

- [1] R. Kornicki, R. Rahmfeld, S. Srinivasamurthy, Q. Hongbo and L. Minghui, "On-Board Cylinder Bore Pressure Measurement of Axial-Piston Hydrostatic Machines," in *International Conference on Fluid Power Transmission and Control (ICFP2017)*, Hanzhou, China, 2017.
- [2] J. Ivantysyn and M. Ivantysynova, *Hydrostatic pumps and motors : principles, design, performance, modelling, analysis, control and testing*, New Delhi: Tech Books International, 2003, 2003.
- [3] J. Tvarůžek, R. Rahmfeld, C. Fiebing, A. Krahn and W. Göllner, "Dynamic Valve Plate Design for an Axial Piston Pump (Servo-Less Pump)," in *14th International Fluid Power Conference*, Dresden, 2024.
- [4] N. J. Fulbright, S. Hall, M. Creswick, S. Smolka and R. Rahmfeld, "Efficiency Improvements in a Servoless Swash Plate Pump: Experimental and Simulation Results," in *2024 International Maha Fluid Power Conference*, Purdue, 2024.
- [5] K.-H. Titscher, "Untersuchung von Einflüssen auf den Druckverlauf in einem Zylinder einer Hochdruck-Hydraulik-Axialkolbenpumpe," RWTH Aachen University Disseration, Aachen, 1970.
- [6] A. Kleist, "Berechnung von hydrostatischen Dichtstellen in hydraulischen Maschinen," *Ölhydraulik und Pneumatik*, vol. 39, pp. 767-771, 1995.
- [7] M. Ivantysynova, "Temperaturfeld im Schmierpalt zwischen Kolben und Zylinder einer Axialkolbenmaschine," *Maschinenbautechnik*, vol. 34, pp. 532-535, 1985.
- [8] L. Olems, "Investigations of the Temperature Behavior of the Piston Cylinder Assembly in axial Piston Pumps," *International Journal of Fluid Power*, vol. 1, no. 1, pp. 27-38, March 2000.
- [9] B. O. Helgestad, K. Fostesr and F. Bannister, "Pressure Transients in an Axial Piston Hydraulic Pump," *Proceedings of the Institution of Mechanical Engineers*, vol. 188, no. 17/74, pp. 189-200, 1974.
- [10] M. Ivantysynova and R. Lasaar, "An Investigation into Micro- and Macrogeometric Design of Piston/Cylinder Assembly of Swash Plate Machines," *International Journal of Fluid Power*, vol. 5, no. 1, pp. 23-36, 2004.

- [11] M. Bergemann, Systematische Untersuchung des Geräuschverhaltens von Kolbenpumpen mit ungerader und mit gerader Kolbenzahl, Aachen: Mainz, 1994.
- [12] R. Ivantysyn and J. Weber, "Advancing Thermal Monitoring in Axial Piston Pumps: Simulation, Measurement, and Boundary Condition Analysis for Efficiency Enhancement," *International Journal of Fluid Power*, vol. 25, no. 4, pp. 547-590, 2024.
- [13] R. Ivantysyn and J. Weber, "Efficiency Forecasting in Axial Piston Pumps: Leveraging High Speed Fluid Temperature Correlations for Condition Monitoring," in *The 12th International Symposium on Fluid Power*, Hiroshima, Japan, 2024.
- [14] U. Wiecezorek and M. Ivantysynova, "Computer Aided Optimization of Bearing and Sealing Gaps in Hydrostatic Machines - the Simulation Tool Caspar," *International Journal of Fluid Power (IJFP)*, vol. 3, no. 1, pp. 7-20, 2002.

Full 3D CFD and FEM Validation of Surrogate-Optimized Hydraulic Manifolds Designed via the MOThEA Framework

Samuele Nizzardo¹, Andrea Vacca², Alberto Benato¹

¹*Department of Industrial Engineering, University of Padova, Via Venezia 1, Padova, 35131, Italy, Europe, samuele.nizzardo@phd.unipd.it, alberto.benato@unipd.it*

²*Maha Fluid Power Research Center, Purdue University, 1500 Kepner Dr, Lafayette, 47905-6578, IN, USA, avacca@purdue.edu*

Abstract

Despite their high power density and robust performance in aerospace and mobile applications, traditional hydraulic actuation systems are increasingly scrutinized for their energy consumption and operational efficiency. In this context, optimization of the hydraulic manifold architecture plays a key role. The internal routing of these components strongly affects both steady-state flow losses and overall structural mass, making manifold layout a critical factor in improving fluid-dynamic performance and the efficiency of next-generation hydraulic systems.

This paper presents the CFD and structural validation of a fully automated design algorithm for the generation of optimized internal flow topologies in additively manufactured hydraulic manifolds (MOThEA, Manifold design Optimization through Theta* path finder and Evolutionary Algorithms). The proposed framework implements a self-executing sizing methodology that estimates pressure drops using an adapted Ito correlation and determines the minimum safe wall thickness through Lamé equations. Compared with conventional design workflows, which typically require time-consuming manual iterations, the proposed approach significantly reduces design time (less than 25 minutes for a complex real-world application) while enabling a systematic exploration of highly efficient manifold configurations. To validate the algorithm, a reference optimized manifold geometry was analyzed. Assuming the same material, the CFD results show that the optimized topology reduces pressure losses by more than 84% compared to a conventional cross-drilled baseline. At the same time, finite element structural analysis shows that despite a 93.1% reduction in weight, the optimized manifold satisfies structural integrity requirements under operating pressure. Although the present mass evaluation does not include valve-thread connections or support structures, the results demonstrate the reliability of the proposed algorithm in generating feasible ultra-lightweight manifolds with markedly reduced parasitic losses.

Keywords: Computational Fluid Dynamics, Finite Element Method, Manifold Optimization, MOThEA, 3D Routing Planning, Mass Reduction, Pressure Losses Reduction.

1. INTRODUCTION

Hydraulic actuation systems are a fundamental component of modern aerospace vehicles, where efficiency, reliability, weight, and integration constraints play a critical role in overall system performance. Within these systems, hydraulic manifolds significantly influence both energy efficiency and structural weight, as internal pressure losses and material distribution directly affect power off-take requirements and integration margins. In this context, there is significant ongoing research focused on enhancing the efficiency of hydraulic systems (HSs), and the following are just a few examples of the entire landscape. Beltrami et al. [1] investigated the electrification of compact off-highway vehicles, Mahato and Ghoshal [2] reviewed energy-saving strategies in hydraulic power systems, Quan et al. [3] surveyed powertrain technologies for energy-efficient heavy-duty machinery, Li et al. [4] reviewed flow-matching technologies for hydraulic systems, Du et al. [5] presented a comprehensive review of high-efficiency electro-hydrostatic actuator technologies with energy recovery, and Padovani et al. [6] discussed the challenges and possible solutions for designing energy-efficient and low-pollutant machines in off-road hydraulics.

Fundamentally, the main aim of a manifold is to direct the hydraulic fluid in the desired direction with the right pressure, increasing the compactness of the system, supporting valves and probes. From an industrial perspective, pressure losses (PLs) within manifolds are primarily attributed to valves because they reduce the available flow area and increase localized pressure losses (LPLs). However, even though this mechanism is partly unavoidable, the design of the manifold itself strongly affects hydraulic performance, as well as its weight.

In recent years, the scientific community has started considering manifold optimization (MO) as an important research direction. In particular, MO is focusing on the design optimization of already existing manifolds, exploiting computational fluid dynamics (CFD) and additive manufacturing (AM) machining. For example, Abe et al. [7] studied the pressure drops associated with two different manifold block machining strategies by combining CFD analysis and experimental tests, while Zhang et al. [8] used a genetic algorithm to define the connection order of a network. Alshare et al. [9] investigated the feasibility of AM for the production of efficient valve manifolds for hydraulic actuators by redesigning conventionally manufactured valve blocks. Hong et al. [10] developed a method for flow-channel optimization based on 3D printing.

A first attempt to move toward a more generalized design methodology was proposed by Zhu et al. [11], who developed a design approach of hydraulic manifolds using Selective Laser Melting with a particular focus on fluid flow to reduce both pressure losses and manifold weight. Similarly, Rolinck et al. [12] introduced a development methodology for hydraulic manifolds and fluid components manufactured by laser powder bed fusion. Their process systematically leads to parts with improved performance, economic efficiency, and manufacturability. Again, Zhang et al. [13] developed a multi-objective layout design approach for lightweight and energy-saving hydraulic manifolds, and lastly, Zhang et al. [14] refined their design approach moving to an A*-based algorithm to increase the flexibility of the previous proposed method, moving to the automation of the process. The last two studies represent important steps toward more automated design procedures and, while the first is closely tied to specific manifold configurations, the second provides a more general approach.

To further push the boundaries of automated design, the authors recently developed MOThEA (Manifold design optimization through Theta* path finder and Evolutionary Algorithms), a fully automated and general design framework for the synthesis of hydraulic manifolds. MOThEA establishes a computer-aided design (CAD) methodology for 3D geometry synthesis, generating internal paths aimed at reducing both global pressure losses and structural mass. To drastically reduce computational costs during the evolutionary loop, MOThEA relies on fast analytical surrogate models: the analytical integration of Ito's equation along the 3D splines for fluid pressure drops, and Lamé's theory of thick-walled cylinders for structural thickness sizing. While these 1D/2D analytical surrogates enable the evaluation of thousands of individuals in an acceptable time-frame, they inherently simplify complex 3D phenomena, such as fully developed Dean vortices, flow

separations at junctions, and realistic 3D von Mises stress concentrations under high-pressure loads. Therefore, the present work proposes a rigorous computational validation via full 3D CFD and FEM analyses of the optimized manifolds generated by the MOThEA framework. The primary goal of this study is to assess the accuracy of the surrogate models and validate the proposed optimized designs from both a fluid dynamic and structural integrity perspective. Full-scale 3D numerical simulations are employed to verify that the analytical assumptions do not compromise the actual physics of the manifold under real operating conditions.

To highly emphasize the improvements brought by the optimization, a reference baseline configuration (an unoptimized, straightforward orthogonal geometry) is also cross-evaluated and compared. At the end of the proposed design approach, the only structural details omitted from the simulations are the valve-thread connections and external customized mounting supports.

The remainder of the paper is organized as follows. Section 2 briefly introduces the analytical models implemented in MOThEA. Section 3 presents the numerical set-up of the analyses for both CFD and FEM evaluations. Section 4 discusses the fluid-structural comparison between the baseline and the optimized geometries. Finally, Section 5 summarizes the main findings and outlines future developments.

2. ANALYTICAL SURROGATE MODELS

To contextualize the subsequent 3D numerical validation, it is necessary to briefly outline both the MOThEA framework and the analytical surrogate models it employs during the optimization loop.

MOThEA couples three stages into a single automated pipeline. First, a Theta* path finder (PF) builds, within the discretized design domain, an initial collision-free routing between the prescribed ports, which is subsequently discretized into a set of guide-points. Theta* PF denotes the any-angle path-planning algorithm used to generate the initial routing: like the classical A* search, it expands the nodes of a weighted graph to minimize a cost function $f(n) = g(n) + h(n)$, where $g(n)$ is the accumulated path cost from the start node and $h(n)$ an admissible Euclidean heuristic to the goal. However, unlike A*, it allows a node's parent to be any node in line of sight rather than only an adjacent grid neighbour, producing shorter and smoother routes that are not constrained to the grid directions. Second, a hybrid stochastic multi-objective optimizer refines this routing in continuous space: the design variables are the guide-point coordinates and the channel internal diameters d_{in} (port locations may also be released as variables when not fixed a priori), while the two competing objectives are the manifold mass and the aggregate pressure drop, both evaluated through the analytical surrogates described below. The search combines Success-History Adaptive Differential Evolution (SHADE) and Population-Based Incremental Learning with Adaptive Learning Rate (PBIL-ALR) with a semi-dynamic elitism operator. Geometric feasibility (minimum inter-branch clearance, obstacle avoidance, near-orthogonal attachment at the external ports, and non-coincidence of successive control points) is enforced through an ε -relaxed variant of Deb's feasibility rules. Third, the converged spline centrelines and diameters are swept into a software-independent solid model ready for AM. The full formulation of the optimizer is detailed in the companion MOThEA design study.

To avoid computationally expensive 3D simulations during evolutionary generation, the algorithm synthesizes the manifold topology relying on Lamé's formulation for structural sizing and Ito's correlation for fluid dynamic losses along 3D spline paths.

2.1. Structural Mass Evaluation

Because standard thin-walled approximations are inadequate for high-pressure hydraulic components, the minimum required wall thickness (t_{min}) for each generated channel is sized according to Lamé's thick-walled

cylinder theory. The structural constraints are analytically embedded using a predefined safety factor (SF):

$$t_{\min} = \frac{d_{\text{in}}}{2} \left(\sqrt{\frac{\sigma_{\text{al}} + p_{\max}}{\sigma_{\text{al}} - p_{\max}}} - 1 \right) \quad (2.1)$$

where d_{in} is the diameter of the internal channel, $\sigma_{\text{al}} = \sigma_y/SF$ is the allowable material yield stress, and $p_{\max}$ is the maximum operating pressure. The wall material of a channel of length L forms an annulus of cross-sectional area $\frac{\pi}{4}(d_{\text{out}}^2 - d_{\text{in}}^2)$, with outer diameter $d_{\text{out}} = d_{\text{in}} + 2t_{\min}$. Substituting the thickness from the previous equation gives $d_{\text{out}} = d_{\text{in}} \sqrt{(\sigma_{\text{al}} + p_{\max})/(\sigma_{\text{al}} - p_{\max})}$, so that

$$d_{\text{out}}^2 - d_{\text{in}}^2 = d_{\text{in}}^2 \left[\frac{\sigma_{\text{al}} + p_{\max}}{\sigma_{\text{al}} - p_{\max}} - 1 \right] = d_{\text{in}}^2 \frac{2p_{\max}}{\sigma_{\text{al}} - p_{\max}}.$$

Collecting this pressure- and material-dependent factor into a global load constant K ,

$$K = \frac{2p_{\max}}{\sigma_{\text{al}} - p_{\max}} = \frac{2}{\frac{\sigma_{\text{al}}}{p_{\max}} - 1}, \quad (2.2)$$

the theoretical printable mass of the topological structure is thereby computed as:

$$m_a = \frac{\pi}{4} K \rho_s d_{\text{in}}^2 L \quad (2.3)$$

where ρ_s is the density of the AM alloy and L is the true geometric arc length of the channel. Note that the structural mass scales with the geometric length L (the physical amount of material), whereas the curvature-enhanced effective length L_{eff} introduced in Section 2.2 enters only the fluid-resistance model.

2.2. Pressure Losses Estimation

Rather than applying discrete lumped loss coefficients to macroscopic bends, MOTHEA analytically predicts the pressure drop by treating curvature-induced secondary flows (e.g. Dean vortices) as an equivalent distributed friction. This is achieved by continuously integrating Ito's relation [15] along the curvilinear abscissa s . The resulting effective length L_{eff} serves as the straight-pipe thermodynamic equivalent of the complex 3D non-planar spline:

$$L_{\text{eff}} = \int_0^L \left[1 + 0.14 \left(\frac{d_{\text{in}}}{2R(s)} \right)^{0.97} \right] ds \quad (2.4)$$

where $R(s)$ is the local analytical radius of curvature and L is the true geometric arc length.

The distributed pressure drop Δp_d is then calculated using a parameterized Darcy-Weisbach formulation for a target volumetric flow rate Q :

$$\Delta p_d = \frac{8\rho_f f_t}{\pi^2 d_{\text{in}}^5} Q^2 L_{\text{eff}} \quad (2.5)$$

where ρ_f is the density of the working fluid. The parameter f_t is the turbulent friction factor, explicitly evaluated via the Swamee-Jain equation [16] and bounded by the local Reynolds number and the typical relative AM surface roughness (Ra/d_{in}). The fully-turbulent assumption is justified by the maximum-flow boundary conditions.

Finally, purely localized losses (Δp_l) are restricted exclusively to macroscopic fluid-domain transitions—namely, sudden contractions at the inlets (C_{in}) and sudden expansions at the outlets (C_{out}):

$$\Delta p_l = \frac{8\rho_f}{\pi^2 d_{\text{in}}^4} Q^2 \sum_{i=1}^k (C_{\text{in}} + C_{\text{out}})_i \quad (2.6)$$

with k being the total number of independent manifold channels.

3. NUMERICAL SET-UP

The geometry of the manifold before optimization is depicted in Figure 3.1b. It is a conventional 5-axis drilled manifold composed of two main blocks: the pre-filter section and the post-filter section. Figure 3.1a highlights the internal flow domain. The optimized geometries generated via MOTHEA are shown in Figure 3.2, where

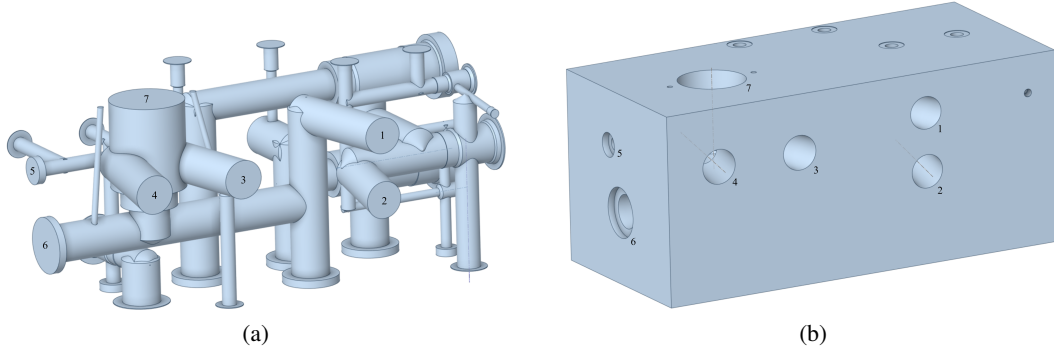


Figure 3.1. *Reference manifold from a real-world hydraulic application in a coil compactor machine. (a) Internal topology of the conventionally manufactured manifold; (b) corresponding external shape. Equal numbering denotes corresponding boundary ports.*

Figure 3.2a and Figure 3.2b represent the pre-filter and the post-filter blocks respectively, while Figure 3.2c displays the assembled solution. Note that port “6” of the baseline is a drilling-process access hole with no functional flow role; its omission from the synthesized layout therefore leaves the hydraulic function of the manifold unchanged.

3.1. CFD Validation Framework

Computational Fluid Dynamics enables the detailed evaluation of flow fields across complex spatial geometries without requiring physical prototyping. This study utilizes steady-state Reynolds-Averaged Navier-Stokes (RANS) equations to provide a reliable and validated balance between computational efficiency and physical accuracy. The commercial suite ANSYS Fluent was adopted to perform fluid dynamic validation. For a steady-state framework (excluding the initial transient filling phase), the conservative governing equations for continuity, momentum, and energy are formulated as follows:

$$\frac{\partial(\rho u_j)}{\partial x_j} = 0 \quad (3.1)$$

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \quad (3.2)$$

$$\frac{\partial(\rho e u_j + p u_j)}{\partial x_j} = -\frac{\partial q_j}{\partial x_j} + \frac{\partial(\tau_{ij} u_i)}{\partial x_j} \quad (3.3)$$

where $\rho = \rho_f$ is the fluid density (subscript “f” was avoided for clarity of the equations), x_j specifies the Cartesian coordinate direction (with $j = 1, 2, 3$), u_j is the mean velocity component, p is the static pressure, τ_{ij} represents the effective fluid stress tensor (incorporating both molecular viscosity and turbulent Reynolds stresses), e is the total specific energy, and q_j is the heat flux.

The turbulence closure was achieved using the two-equation $k - \omega$ SST model, enhanced with *Low-Reynolds* number corrections, as the maximum expected Reynolds number typically lies within the transitional to fully turbulent boundary ($Re_{\max} \leq 6.5 \times 10^4$).

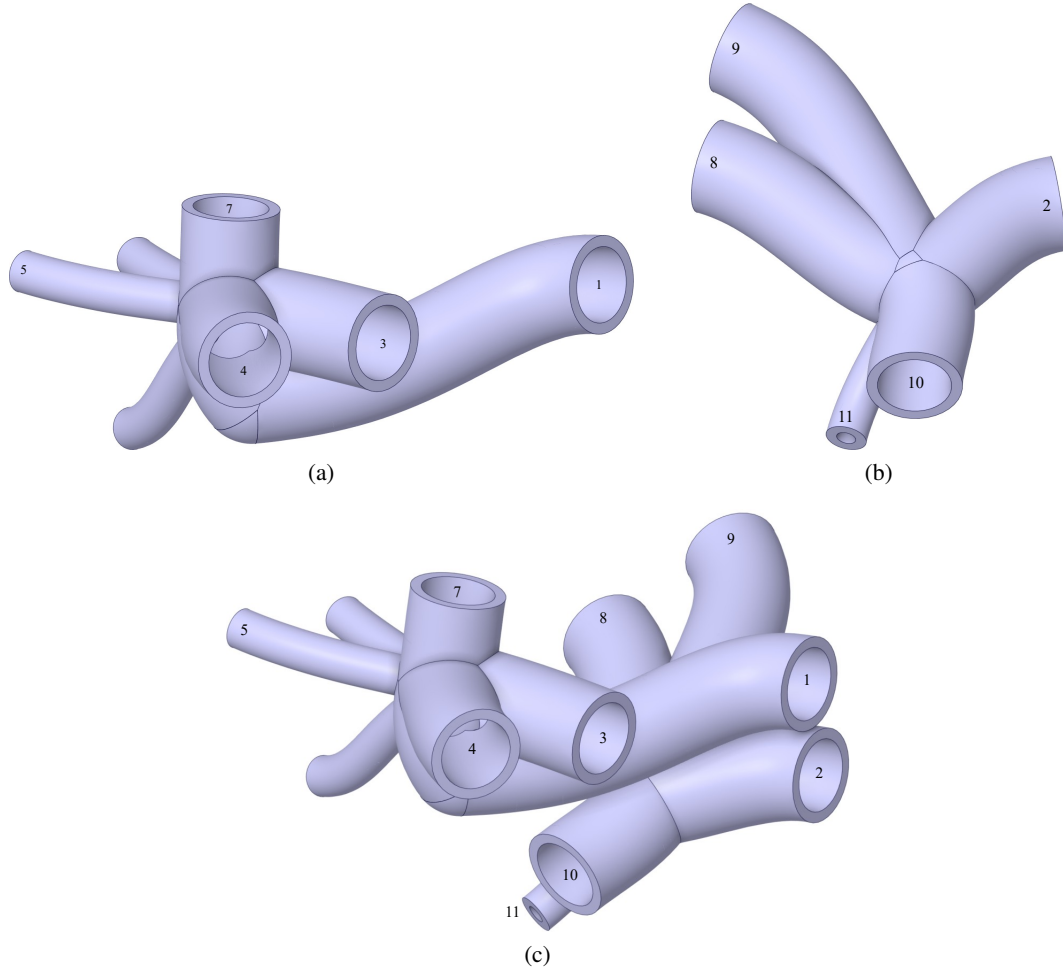


Figure 3.2. *Optimized manifold geometry. Figure (a) is the pre-filter block, while Figure (b) represents the post-filter side. Figure (c) exposes a viable embedded solution between the two blocks. Numbered ports directly map to the conventional baseline references in Figure 3.1. In particular, ports "1", "3", and "4" represent the inlet and distribution outlets of the post-filter block. Ports "2", "8", and "9" act as the outlet and inlets of the pre-filter block, whereas ports "10" and "11" identify static probing/valve lines. Ports "5" and "7" correspond to probe lines. Port "6" of the baseline is a drilling-process access hole with no functional flow role; it is therefore absent from the synthesized layout, which leaves the hydraulic function of the manifold unchanged.*

To ensure grid independence, three different unstructured tetrahedral grids were evaluated for both the pre- and post-filter fluid domains: coarse ($\sim 0.8 \times 10^6$ elements), medium ($\sim 2.3 \times 10^6$ elements), and fine ($\sim 6.2 \times 10^6$ elements). The medium density meshes were ultimately selected, as they yielded asymptotic invariants compared to the fine grid, thus optimizing the computational cost. To properly resolve the momentum and thermal boundary layers, guaranteeing a non-dimensional wall distance $y^+ \leq 1$, a wall-normal inflation layer composed of structured prismatic elements was strictly enforced on all no-slip boundaries. Figure 3.3 illustrates the medium-sized computational grid for the post-filter block (Figure 3.3a) alongside a detailed view of the inflation layers (Figure 3.3b). For the sake of conciseness, the pre-filter mesh highlights are omitted.

The imposed boundary conditions reflect the nominal operational state of the hydraulic circuit, using *Velocity*

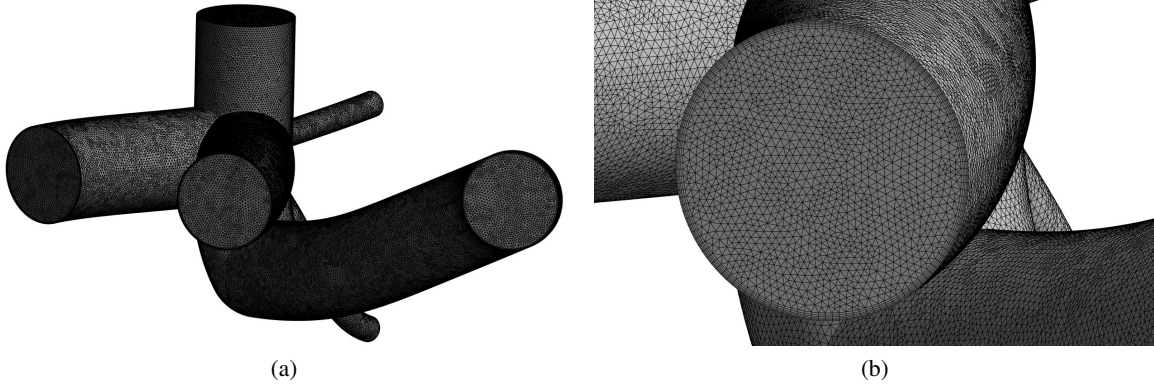


Figure 3.3. Representation of the unstructured computational mesh. (a) Post-filter fluid domain volume grid; (b) detailed cut-plane highlighting the near-wall prism layer inflation.

Inlet and *Pressure Outlet* formulations. The two blocks are analyzed as separate fluid domains, following the port mapping of Figure 3.2. In the pre-filter domain, a velocity inlet is prescribed at each of the two feed ports “8” and “9”, and a pressure outlet at port “2”; in the post-filter domain, a velocity inlet is prescribed at port “1” and pressure outlets at the two distribution ports “3” and “4”. The static probing/valve lines (ports “10” and “11”) carry no net flow and are treated as no-slip walls. The prescribed inlet velocities correspond to a nominal volumetric flow rate of $Q = 400$ L/min ($200 + 200$ L/min at ports “8” and “9”), while the static pressure at the outlets is set to 140 bar. The same inlet flow rates and the same inlet split were imposed on the baseline and the optimized geometries, so that the two configurations are compared under strictly identical boundary conditions. To simulate the friction induced by conventional and additive manufacturing, an equivalent sand-grain roughness of $H_{Ra} = 5 \times 10^{-5}$ m was assigned to the wall boundaries. The thermal gradients were modeled by imposing a uniform wall temperature $T_w = 288$ K that interacts with an active fluid inlet temperature of $T_{in} = 313$ K.

3.2. FEM Analysis Framework

To verify that the 1D structural surrogate sizing based on Lamé’s theory remains reliable across the complex 3D topology generated by MOThEA, a rigorous structural validation was performed. The commercial structural suite ANSYS Mechanical was employed to simulate the static load behavior under the maximum design pressure.

Given the ductile nature of high-strength AM steel alloys adopted in advanced manifold manufacturing, the structural integrity was evaluated globally using the von Mises yield criterion. The equivalent von Mises stress (σ_{vm}) was calculated in the solid domain and strictly monitored against the allowable material yield limit:

$$\sigma_{vm} = \sqrt{\frac{1}{2} [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]} \leq \frac{\sigma_y}{SF} \quad (3.4)$$

where $\sigma_1, \sigma_2, \sigma_3$ are the principal stresses, σ_y is the material yield strength, and $SF = 1.7$ designates the safety factor. To correctly isolate the linear-elastic response of the material from non-physical localized stress singularities, a specific grid sensitivity analysis was conducted. Three unstructured solid tetrahedral meshes (coarse, $\sim 1.1 \times 10^5$ elements; medium, $\sim 2.2 \times 10^5$ elements; fine, $\sim 5.5 \times 10^5$ elements) were generated. The medium grid was deemed sufficient to capture the stress concentrations around topological junctions without inducing artificial stress divergence.

Regarding the boundary conditions, the worst-case static loading scenario was simulated. A uniform internal

static pressure equal to the maximum circuit threshold ($p_{\max} = 240$ bar) was assigned to all internal flow channel faces. Furthermore, to accurately replicate the structural rigid constraints of the real-world installation, *Fixed Support* conditions (zero displacement) were applied to all the primary structural anchoring ports of the hydraulic circuit (refer to the topological mapping in Figure 3.2). Specifically, these rigid constraints were enforced at the functional interfaces bridging the manifold blocks to the filter assembly (port "2" acting as the main outlet of the pre-filter block, and port "1" acting as the main inlet of the post-filter block). Simultaneously, the two primary fluid inlets (ports "8" and "9" on the pre-filter) and the two primary fluid distribution outlets (ports "3" and "4" on the post-filter) were completely constrained. In contrast, the structural boundaries of the functional ports dedicated to hydraulic valves and sensing probes were left free. This constraint configuration allows these terminal connections to undergo unconstrained elastic deformation driven by the internal pressurization, closely replicating their authentic floating boundary behavior within the working machine.

4. RESULTS AND DISCUSSION

As detailed in the numerical setup (Section 3), all full-scale CFD and FEM validations were performed relying on the grid-independent medium computational meshes. The objective of this section is twofold. Firstly, the 3D flow simulations aim to elucidate complex, curvature-induced secondary flow patterns and macroscopic mean-flow separations that cannot be entirely modeled by the 1D analytical surrogate equations, thereby providing a definitive assessment of the actual distributed pressure losses. Secondly, the solid mechanics FEM study serves to verify the global structural integrity of the complex parametrically generated splines, ensuring that no localized stress singularities near the connection ports compromise the structural margins computed by Lamé’s sizing framework.

4.1. Surrogate Model Validation & Pressure Losses Reduction

Table 1 summarizes the quantitative performance metrics, comparing the conventional baseline against the MOTHEA-optimized geometry in terms of global pressure losses. The topological optimization process yielded a drastic fluid dynamic enhancement: operating under identical boundary conditions, the optimized framework achieved an 84.1% reduction in total pressure drop compared to the traditional drilled manifold.

Table 1: *Quantitative performance comparison among the CFD analysis of the conventional baseline, the CFD validation of the optimized design, and the analytical predictions formulated by the proposed algorithm.*

Configuration Setup	Pressure Losses [bar]	PLs Reduction ¹ [%]
Conventional baseline (CFD)	6.42	-
Optimized actual CAD (CFD)	1.02	84.1
Optimized routing (Analytical algorithm)	1.08	83.1

¹ Compared to the conventional baseline CFD results.

Interestingly, the pressure drop estimated by the 1D analytical surrogate (1.08 bar) slightly over-predicts the actual CFD result (1.02 bar), with a mismatch of about 6.1%. This conservative bias is the intended consequence of the fully-turbulent worst-case assumption adopted for the friction model (Section 2.2), which upper-bounds the distributed losses over the operating range. Dead-end branches (e.g. pressure probes or closed valve pilot lines, ports “10” and “11”) were assigned zero flow, consistent with their role as static terminal stubs; the small localized loss produced in the main channels by the flow passing these dead legs is resolved by the CFD but is not represented in the analytical model. Since this omission acts opposite to the

friction over-estimate and is small, the analytical prediction remains a conservative upper bound on the CFD losses. The close agreement between the two values confirms that the analytical surrogate reliably reproduces the pressure drop of the optimized geometry.

To visually elaborate on these numerical achievements, Figure 4.4 maps the velocity magnitude and total pressure streamlines within the pre-filter flow field, directly contrasting the optimized routing against the conventional design. The macroscopic differences dictating the PL drop in the pre-filter block are clearly identifiable. First, the traditional baseline geometry exhibits massive stagnation regions (dead zones) inevitably induced by orthogonal cross-drilling manufacturing. In contrast, MOTHEA-generated splines completely bypass these dissipative volumes, maintaining a highly efficient and perfectly attached flow along the entire routing. Secondly, the maximum localized fluid velocity is drastically reduced, dropping from a peak of 24.5 m/s in the baseline to just 12.3 m/s. This is partially attributed to the optimizer autonomously finding a better volumetric trade-off, expanding the internal hydraulic diameter from 31 mm to ~ 38 mm. Lastly, the elimination of sharp 90° junctions suppresses severe *vena contracta* effects, significantly mitigating localized pressure drops. The corresponding total pressure fields (Figure 4.4e-h) confirm the sudden energy dissipation across the sharp corners of the baseline, compared to the smooth pressure gradient of the AM-ready solution. Similar fluid-dynamic enhancements govern the primary transport channels of the post-filter block, as displayed in Figure 4.5. Although the smooth continuous curvature successfully prevents severe boundary layer separations along the main conduits, localized flow recirculation and large swirling structures are naturally observable within the terminal plenums (Figure 4.5a,b). These macroscopic volumetric expansions are physically required by the functional housing of the hydraulic components (e.g. valves) and cannot be bypassed. Nevertheless, compared to the conventional baseline, the optimized 3D routing ensures that the fluid momentum is delivered to these terminal expansions with minimal intermediate dissipation, confirming the efficiency of the overall structural topology.

4.2. Structural Integrity & Mass Reduction

Alongside the reduction in fluid pressure losses, the topological optimization process produced a significant containment of structural mass. Table 2 details the quantitative mass breakdown, comparing the conventional drilled baseline against the algorithmically generated splines.

Table 2: *Quantitative mass comparison between the conventional baseline manifold block and the MOTHEA-optimized free-form topology. The structural assessment assumes AM Inconel 718 ($\rho_s = 8150 \text{ kg/m}^3$).*

Configuration	Pre-Filter [kg]	Post-Filter [kg]	Total Mass [kg]
Conventional baseline	-	-	81.078
Optimized actual CAD	2.598	3.044	5.642
Mass Reduction	93.1%		

As reported in Table 2, the surrogate-driven algorithm decreased the overall system mass from 81.078 kg to 5.642 kg, corresponding to a relative reduction of 93.1%. This mass saving highlights the potential of free-form topological design for weight-sensitive applications.

To practically realize this geometry and withstand high operational pressures, the selected structural material must exhibit adequate mechanical properties. The analysis relies on Inconel 718 processed via Direct Metal Laser Sintering. This AM material is characterized by an orthotropic yield strength induced by the layer-by-layer manufacturing process: a nominal $\sigma_{y,XY} = 1240 \text{ MPa}$ in the printing plane, and a lower $\sigma_{y,Z} = 1145 \text{ MPa}$ along the vertical build direction. Consequently, the full 3D FEM structural evaluation

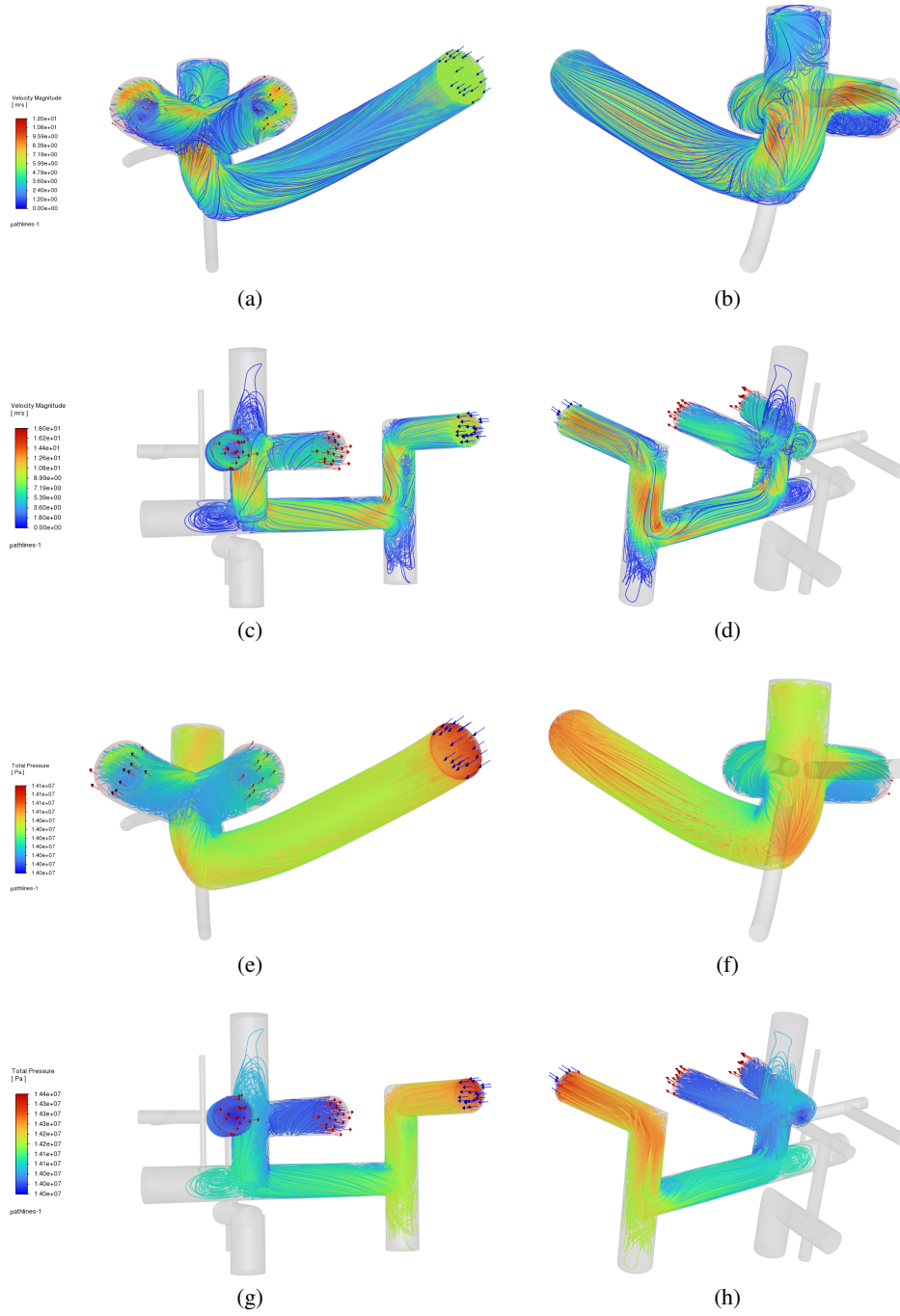


Figure 4.5. Comparative streamline analysis of the post-filter domain. Velocity magnitude for the optimized topology (a, b) versus the conventional baseline (c, d). Total pressure distribution for the optimized (e, f) versus the baseline (g, h).

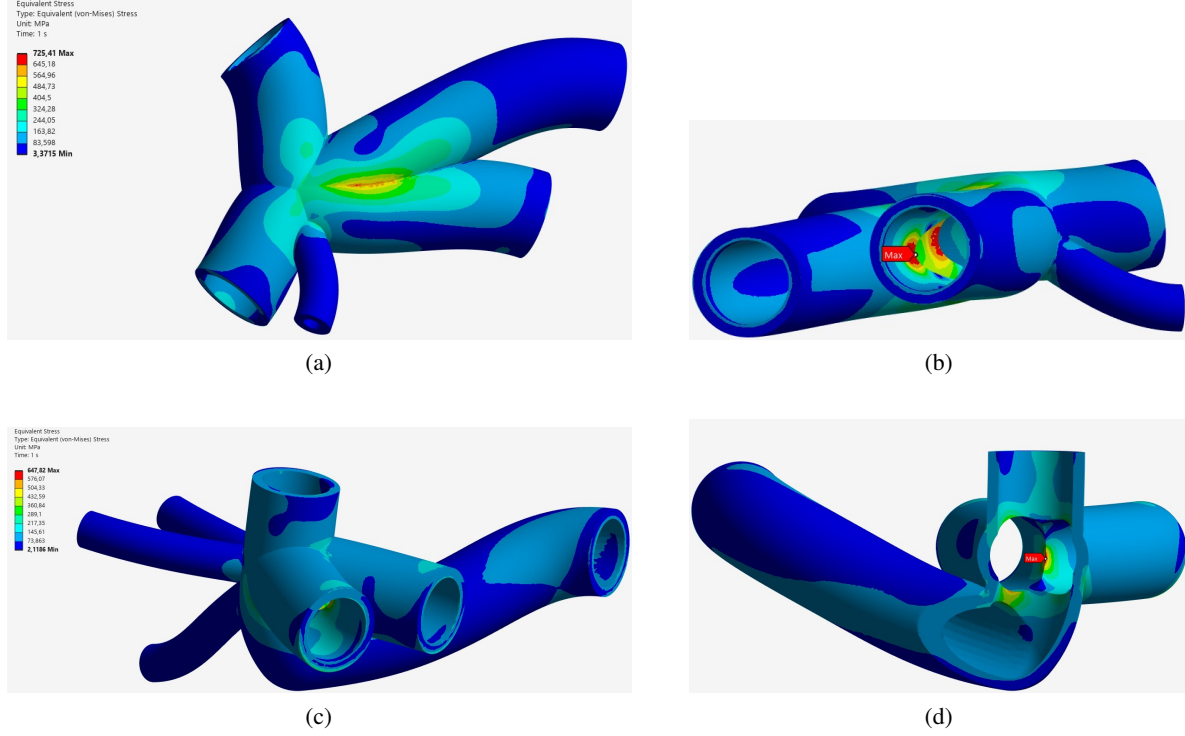


Figure 4.6. Analysis of the von Mises equivalent stress (σ_{vm}) distribution. Global structural view of the pre-filter block (a) and a detail on the maximum stress concentration (b). Corresponding global view for the post-filter block (c) and its localized stress peak (d).

was conducted to verify whether the analytical thickness-sizing equations, based on Lamé’s cylinder theory, maintain the stress tensors under maximum pressurization safely below the critical 1145 MPa structural limit. Figure 4.6 presents the resulting von Mises equivalent stress contours for both the optimized pre-filter and post-filter blocks under steady static load conditions. As inherently predicted by the theoretical surrogate model, the vast majority of curvilinear tubular domains exhibits a globally uniform low-magnitude stress field (mainly below 250 MPa, as indicated by the blue contours). This effectively validates Lamé’s 1D sizing assumption for the required nominal wall thickness. However, specific nodal stress concentrations naturally arise due to the complex 3D topology and the presence of unavoidable functional geometries. The pre-filter part registers an absolute structural peak of $\sigma_{vm,max} = 725.41$ MPa that occurs at an acute internal saddle point where two fluid branches merge (Figure 4.6b). Similarly, the post-filter region displays a localized equivalent stress peak of 647.82 MPa, positioned exactly at the sharp internal edge of a functional cross-bore that intersects the main housing plenum (Figure 4.6d). These localized peaks and the corresponding safety factors are summarized in Table 3. Even considering these purely localized geometric notch effects under the most critical loading scenario, the maximum recorded effective stress (725.41 MPa) remains strictly contained below the weakest orthotropic yield limit of the AM material. This leads to an intrinsic minimum safety factor of $SF_{min} = 1.58$, driven by the pre-filter saddle-point notch. It is worth noting that this local value is marginally below the nominal design safety factor ($SF = 1.7$) adopted in the 1D Lamé sizing: the analytical thickness rule holds for the bulk tubular walls, while the fully 3D notch at the branch merge locally erodes the margin, without ever reaching yield. This structural margin confirms that the chosen algorithmic layout yields a robust geometry, efficiently decoupling morphological optimization from computationally expensive iterative FEM simulations, and it is precisely why the 3D FEM cross-check complements the 1D surrogate.

Table 3: *Localized von Mises stress peaks and corresponding safety factors for the optimized blocks under the worst-case static load ($p_{\max} = 240$ bar). The safety factor is computed against the weakest orthotropic yield limit ($\sigma_{y,z} = 1145$ MPa); the design safety factor used in the 1D sizing is $SF = 1.7$.*

Block	$\sigma_{vm,max}$ [MPa]	$\sigma_{y,z}$ [MPa]	Safety factor [-]
Pre-filter	725.41	1145	1.58
Post-filter	647.82	1145	1.77

5. CONCLUSIONS

This paper presented a comprehensive full-scale 3D computational validation of MOTHeA, a recently developed framework for the automated synthesis of hydraulic manifolds via additive manufacturing. The core strength of the algorithm lies in its reliance on highly efficient analytical surrogate models for both fluid dynamic routing and structural sizing. As demonstrated, this approach completely bypasses the need for computationally prohibitive iterative simulations during the evolutionary loop, allowing the generation of a highly optimized, real-world topology in under 25 minutes on standard desktop hardware.

The full 3D CFD and FEM assessments definitively confirmed the physical reliability and structural viability of the algorithm’s assumptions. Operating under severe real-world conditions (maximum flow rate of 400 L/min and internal static pressure of 240 bar), the optimized free-form routing yielded an 84.1% reduction in global pressure drops and a 93.1% decrease in structural mass compared to the conventional cross-drilled baseline. Under these pressure loads, structural integrity was fully guaranteed with a minimum safety factor of $SF_{\min} = 1.58$. While the selected manufacturing material –the Inconel 718 nickel-based super-alloy– entails considerable production costs, its application is fundamentally justified in weight-sensitive sectors, such as aerospace and high-performance mobile machinery, where extreme mass containment translates directly to significant energy and emission savings over the vehicle’s life cycle.

Currently, the MOTHeA methodology successfully synthesizes the primary fluid dynamic channels and the load-bearing structural envelope. Interfacial valve-spool connections, standardized threaded fixtures, and external mounting brackets are deliberately deferred to a downstream mechanical refinement phase. Accordingly, future research will focus on the complete post-processing of the generated manifolds to conduct rigorous physical test campaigns on 3D-printed prototypes. Furthermore, the mathematical surrogate formulation will be expanded to include thermal gradients and dynamic fatigue stress validations under cyclic hydraulic loading.

References

- [1] Beltrami, D., Iora, P., Tribioli, L., Uberti, S.: Electrification of compact off-highway vehicles—overview of the current state of the art and trends. *Energies* **14**(17) (2021) <https://doi.org/10.3390/en14175565>
- [2] Mahato, A.C., Ghoshal, S.K.: Energy-saving strategies on power hydraulic system: An overview. *Proceedings of the Institution of Mechanical Engineers, Part I: Journal of Systems and Control Engineering* **235**(2), 147–169 (2021) <https://doi.org/10.1177/0959651820931627>
- [3] Quan, Z., Ge, L., Wei, Z., Li, Y.W., Quan, L.: A survey of powertrain technologies for energy-efficient heavy-duty machinery. *Proceedings of the IEEE* **109**(3), 279–308 (2021) <https://doi.org/10.1109/JPROC.2021.3051555>
- [4] Li, R., Sun, Q., Ding, X., Zhang, Y., Yuan, W., Wu, T.: Review of flow-matching technology for hydraulic systems. *Processes* **10**(12) (2022) <https://doi.org/10.3390/pr10122482>
- [5] Du, S., Zhou, J., Hong, J., Zhao, H., Ma, S.: Application and progress of high-efficiency electro-hydrostatic actuator technology with energy recovery: A comprehensive review. *Energy Conversion and Management* **321**, 119041 (2024) <https://doi.org/10.1016/j.enconman.2024.119041>
- [6] Padovani, D., Dimitriou, P., Minav, T.: Challenges and solutions for designing energy-efficient and low-pollutant machines in off-road hydraulics. *Energy Conversion and Management: X* **21**, 100526 (2024) <https://doi.org/10.1016/j.ecmx.2024.100526>
- [7] Abe, O., Tsukiji, T., Hara, T., Yasunaga, K.: Flow analysis in pipe of a manifold block. *International Journal of Automation Technology* **6**, 494–501 (2012)
- [8] Zhang, J.J., Wang, J.S., Wang, J.Y., Liu, G., Wang, J.: The optimal design on connection order of network in hydraulic manifold block. In: *Advanced Materials Research*, vol. 201–203, pp. 2190–2194 (2011). <https://doi.org/10.4028/www.scientific.net/AMR.201-203.2190>
- [9] Alshare, A.A., Calzone, F., Muzzupappa, M.: Hydraulic manifold design via additive manufacturing optimized with cfd and fluid-structure interaction simulations. *Rapid Prototyping Journal* **25**(9), 1516–1524 (2018) <https://doi.org/10.1108/RPJ-03-2018-0064>
- [10] Hong, Z., Chen-yu, W., Chen-chen, L.: Optimization of flow channel in 3d printing hydraulic manifold block based on response surface method. In: *2019 IEEE 8th International Conference on Fluid Power and Mechatronics (FPM)*, pp. 217–223 (2019). <https://doi.org/10.1109/FPM45753.2019.9035793>
- [11] Zhu, Y., Wang, S., Zhang, C., Yang, H.: Am-driven design of hydraulic manifolds: Enhancing fluid flow and reducing weight. In: *12th International Fluid Power Conference (12. IFK), Dresden*, vol. 1 (2020). <https://doi.org/10.25368/2020.23>
- [12] Rolinck, N., Schmitt, M., Schneck, M., Schlick, G., Schilp, J.: Development workflow for manifolds and fluid components based on laser powder bed fusion. *Applied Sciences* **11**(16) (2021) <https://doi.org/10.3390/app11167335>
- [13] Chao, Z., Yi, Z., Ruiwen, X., Cong, Z., Huayong, Y., Yi, Z.: Automatic optimisation layout design approach for lightweight and energy-saving additive manufactured hydraulic manifolds. *Virtual and Physical Prototyping* **19**(1), 2408407 (2024) <https://doi.org/10.1080/17452759.2024.2408407>

- [14] Zhang, C., Zeng, Y., Tang, Y., Gu, Y., Wu, M., Wang, W., Zhu, Y., Bing, X., Yang, H.: Three-dimensional follow-up layout design approach for lightweight hydraulic passage networks. Additive Manufacturing Frontiers (2026) <https://doi.org/10.1016/j.amf.2026.200295>
- [15] Itō, H.: Friction factors for turbulent flow in curved pipes. Journal of Basic Engineering **81**(2), 123–132 (1959) <https://doi.org/10.1115/1.4008390>
- [16] Swamee, P.K., Jain, A.K.: Explicit equations for pipe-flow problems. Journal of the Hydraulics Division **102**(5), 657–664 (1976) <https://doi.org/10.1061/JYCEAJ.0004542>

Experimental Characterization and Numerical Modelling of an Electric Vehicle Thermal Management System Integrating a Screw-Type e-Pump

Stefano Li Veli¹, Luca Romagnuolo², Valerio Di Cosmo², Emma Frosina^{1*}, Adolfo
Senatore²

¹ Department of Engineering, University of Sannio, Benevento, Italy

² Department of Industrial Engineering, University of Naples Federico II, Naples, Italy

*E-mail: frosina@unisannio.it

Abstract

Nowadays, the adoption of electric and hybrid-electric propulsion systems has become a common solution for the stringent decarbonization requirements. These technologies require advanced thermal management solutions capable of ensuring effective cooling of critical components while limiting energy consumption. Although centrifugal pumps are commonly used in cooling circuits, their performance can be affected by variable pressure losses and changing operating conditions, reducing flow-control accuracy. This work presents the experimental characterization, numerical modelling, and validation of a thermal management system for a high-performance electric vehicle, integrating a prototype screw-type positive-displacement e-pump. Tests were carried out on a dedicated test rig to hydraulically characterize the main circuit's components, including the electric motor and inverter, under different pump speeds and flow-rate distributions imposed by a needle-valve adjustment. The collected data were used to calibrate and validate a lumped-parameter model developed in Simcenter AMESim. The model accurately reproduced the measured pressure losses and the flow distribution across the different circuit branches, confirming the reliability of the numerical approach.

Keywords. Battery Electric Vehicle, Thermal Management System, Screw-type e-Pump, Experimental analysis and Numerical Validation

1. INTRODUCTION

In recent years, increasing attention has been paid to the environmental sustainability, with the aim of reducing pollution and, more specifically, CO₂ emissions. The transport sector has gained a prominent role and visibility in this context, although it is not the largest contributor [1]. However, this sector has historically shown some delay compared to others;

indeed, its emissions continued to increase until 2007, when the first micro-hybrid technologies, such as Start & Stop systems, began to spread. Emissions started to decrease more consistently only with the subsequent diffusion of hybrid vehicles, including full and mild hybrid electric vehicles, battery electric vehicles (BEVs), and fuel-cell electric vehicles (FCEVs) [2], [3].

The adoption of these new technologies, widely regarded as the most effective solutions to achieve environmental targets, has introduced new technological challenges. Among these, the thermal management of electrified powertrain components plays a crucial role. The main components of an electric vehicle (EV) powertrain, namely batteries, electric motors, inverters, and power conversion systems, operate within relatively narrow temperature ranges [4]. Exceeding these limits results in a significant reduction in performance, efficiency, and component lifetime, together with potential safety concerns. [4], [5]. For these reasons, the Thermal Management System (TMS) of electric and hybrid powertrains represents a key subsystem in EVs. The development and optimization of this subsystem have a strong impact on performance improvement, driving range extension, and reliability enhancement[6].

Nowadays, conventional thermal management systems for EVs, rely on liquid cooling circuits driven by centrifugal pumps. This well-established configuration is generally adopted because of its relative simplicity and low costs [7]. Nevertheless, centrifugal pumps may show limitations in terms of efficiency and flow-rate regulation, especially when not operating in their optimal working ranges or within systems affected by variable pressure drops. Under these conditions, cooling performance could be affected, with a subsequent power consumption increase and overall efficiency decrease [8].

This work proposes an innovative TMS, specifically intended for high-performance EVs, which replaces the conventional centrifugal pump with an innovative positive displacement electrically driven pump (e-pump). The performance of this type of pump is generally less sensitive to the operating conditions of the system when compared to centrifugal ones [9], [10], [11]. In this way, greater operational flexibility can be achieved, enabling a more effective response to different working conditions. Moreover, the proposed pump may also enhance vehicle comfort by reducing noise emissions, an aspect that is becoming increasingly relevant in EVs due to their inherently lower overall noise levels.

The research activity described in this paper focused on the development and use of experimental test rigs for characterizing the main components of the thermal management system. It also addressed, as said, the implementation and integration of the positive displacement pump, the definition of possible system architectures based on this pumping technology, and the development of a lumped-parameter numerical model using the Simcenter AMESim® environment.

This work is part of a research project supported by the Italian Minister “Ministero delle imprese e del made in Italy”, involving three Universities: Politecnico di Milano (PoliMI), CeSMA of University of Naples Federico II (UniNA), and University of Sannio (UniSannio) and the company Fluid-o-Tech srl, leader of the project. Preliminary results achieved by the authors were also presented in a previous publication [12].

2. EXPERIMENTAL ACTIVITIES

The aim of the experimental activity presented in this work was to perform a fluid-dynamic characterization of the main components cooled by an innovative TMS designed within the project, with particular focus on the electric motor and inverter. The experimental campaign was conducted at the laboratories of the University of Naples Federico II. Following the preliminary analysis and hydraulic tests of the individual components, a test rig was set up to measure pressure losses as a function of coolant flow rate, providing the data required for the simulation and calibration of the cooling circuit model. The resulting data were then processed and used to build a predictive numerical model.

The physical test-rig is outlined in Figure 1 and showed in Figure 2. A water-ethylene glycol (EG50W50) reservoir (1) feeds the e-pump under investigation (3). The pump speed is regulated via proprietary software. The coolant flow is then split into two branches serving the cooling circuits of the electric motor (9) and the inverter (10). A needle valve (7) is installed to regulate the flow rate distribution among the two branches, which is measured by two flow meters (5 – FM2, 6 – FM1). Downstream, two fan-coil units (12, 13), arranged in parallel, provide the heat-dissipation function of the circuit radiator. Three pairs of pressure and temperature sensors are located upstream of the inverter (8 – P1, T1), between the cooled components and the heat exchangers (11 – P2, T2), and downstream of the fan coil units (14 – P3, T3). An additional pressure sensor (4 – P0) is installed downstream of the pump. All the sensors installed in the circuit are listed in Table 1.

Table 1. Specifications of the sensors used in the experimental setup

ID	Variable	Type	Brand, Model	Range, Accuracy
FM1	Flowrate	Coriolis	Endress Proline 83F15	0 ÷ 6500 kg/h, ± 0.35%
FM2	Flowrate	Vortex	IFM SV5200	2-48 L/min, ±0.5%
P0,P1	Pressure	Transducer	Gesensing PMP1400	0 ÷ 4 bar, ± 0.15%
P2,P3	Pressure	Transducer	IFM PU5415	0 ÷ 6 bar, ± 0.5%
T1, T2, T3	Temperature	Thermocouple	Thermocouple K type	-200 ÷ 1100 °C, ± 1.5 °C

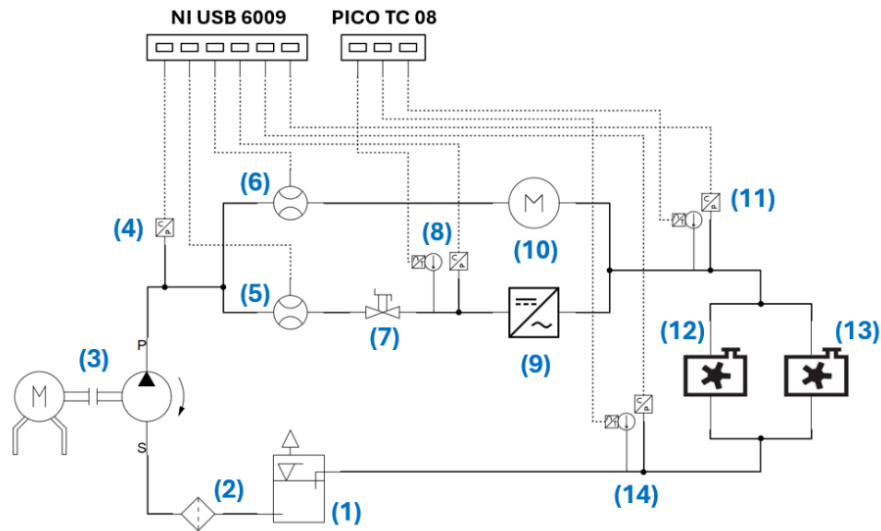


Figure 1. Schematic layout of the experimental test rig.

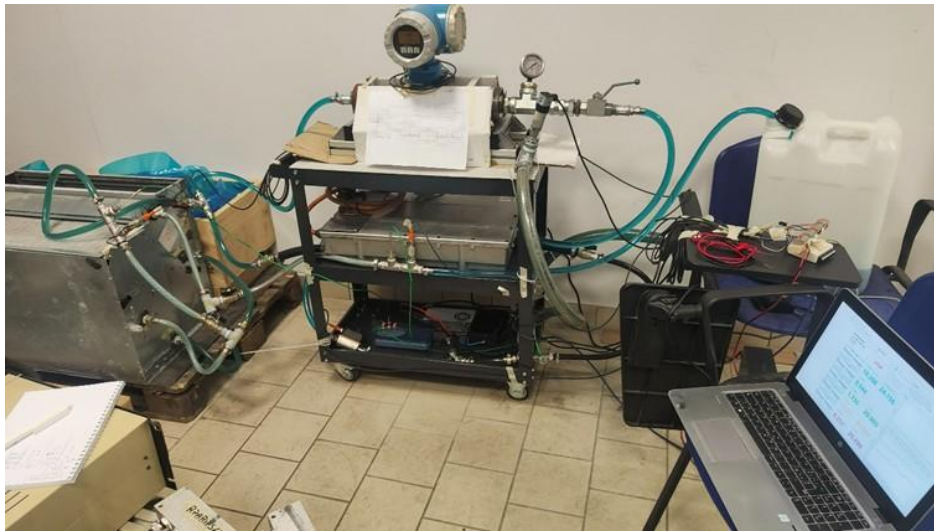


Figure 2. Photograph of the physical test rig

To physically assemble the circuit and connect components with different sizes and interfaces, several auxiliary components were used in addition to rubber hoses with nominal

diameters of 1/2", 1", and 1 1/4". These auxiliary components are listed in Figure 3, together with their locations in the circuit schematic.

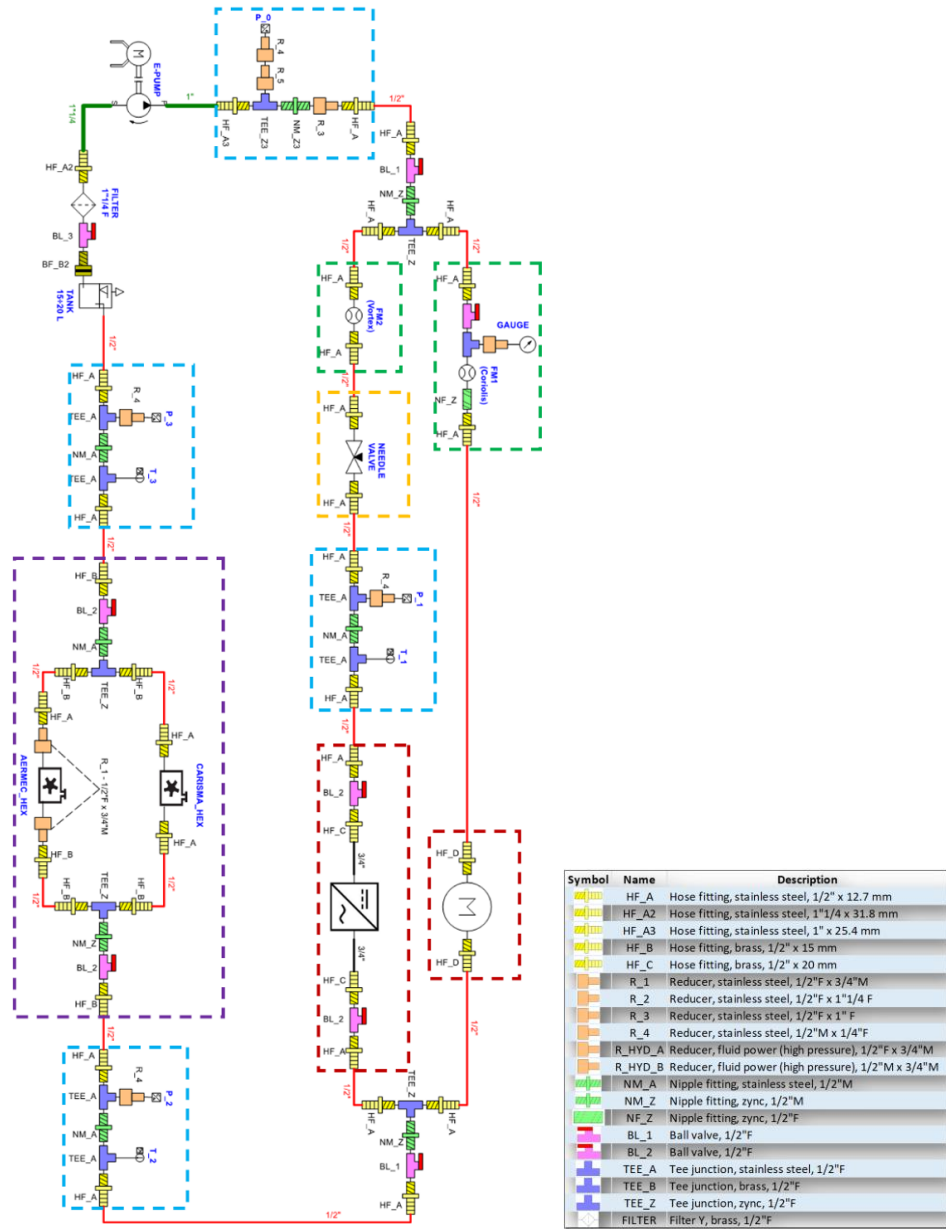


Figure 3. Circuit schematic with fitting definitions and locations

2.1. *Electric Motor*

The electric motor under investigation is the GVM (Global Vehicle Motor) 210 – 300 DPW, engineered, manufactured and sold by Parker Hannifin. The GVM series consists of high-performance permanent magnet synchronous motors specifically engineered for EV propulsion applications. Its main performance specifications are listed below:

- rated power: 155 kW; peak power: 277.8 kW;
- rated torque: 205 Nm; peak torque: 530 Nm;
- rated speed: 7220 rev/min; maximum speed: 8000 rev/min.

The motor is equipped with a liquid cooling circuit and can be cooled using water–glycol mixtures at different concentrations or, alternatively, oil, although motor performance is significantly reduced in the latter case. As specified in the provided datasheet, the maximum coolant temperature and pressure at the motor inlet must not exceed 65 °C and 5 bar, respectively. Furthermore, the maximum allowable coolant temperature rise between inlet and outlet is 10 °C. Since the GVM 210–300 DPW operates with an efficiency higher than 92% over most of its operating range, a coolant flow rate of approximately 20 L/min can be considered sufficient to ensure compliance with these operating limits.

2.2. *Inverter*

The inverter installed in the test-rig is the Parker GVI H650, a high-performance traction inverter designed to supply the electric motor described above. It converts the DC voltage provided by the battery into a controlled three-phase AC voltage, enabling motor speed regulation, torque control, and regenerative braking energy recovery. Its main specifications, reported in the provided datasheet, are reported below:

- nominal voltage: 650 V;
- peak current: 500 A_{rms};
- peak power: 300 kVA.

The inverter is equipped with an internal liquid cooling circuit and is compatible with water–ethylene glycol mixtures at different concentration ratios. According to the datasheet, the coolant temperature and pressure at the inverter inlet must not exceed 60 °C and 2 bar, respectively. In addition, the maximum allowable coolant temperature rise between inlet and outlet is 5 °C.

2.3. *E-Pump*

A prototype screw-type positive-displacement electric pump, provided by the project partner Fluid-o-Tech (FoT), was used [12], [13], [14]. The pump has a volumetric displacement of 13.14 cm³/rev and integrates an electric motor driven by an inverter supplied at 12 V, with a maximum current absorption of 50 A. The pump is controlled through a speed feedback loop, with the target rotational speed set via dedicated software. The main performance limits of the pump are:

- rotational speed: 6000 rev/min;
- flow rate: 50 L/min;

- pressure difference: 3.2 *bar*.

The performance curves of the electric pump were provided by the manufacturer and are shown in Figure 4.

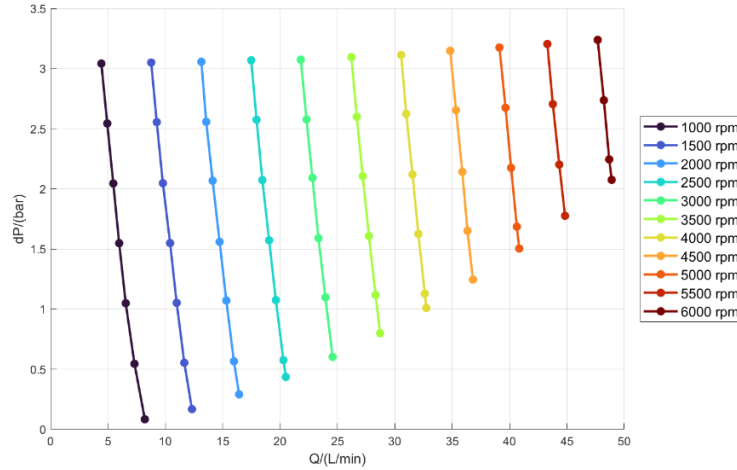


Figure 4. Pump performance curves.

2.4. Fan Coils

The two fan coil units referred to as “Carisma_HEX” and “Aermec_HEX” in Figure 3 respectively represents the selected fan coil units “Carisma CRC44” produced by Sabiana and “FCZ350PO” manufactured by Aermec. Their arrangement in parallel and its series connection with the circuit were adopted to ensure proper heat dissipation while avoiding excessive circuit pressure losses.

3. NUMERICAL MODEL

The aim of the modelling activity was to reproduce the experimental test-rig using a one-dimensional lumped-parameter approach in the Simcenter AMESim environment. The data obtained from the experimental characterization of the individual components were implemented in the model, and the resulting model was subsequently validated against the experimental measurements. Once validated, the model could support future thermal analyses, which were not performed experimentally since all tests were carried out to hydraulically test components without thermal loading. An overview of the model is presented in Figure 5.

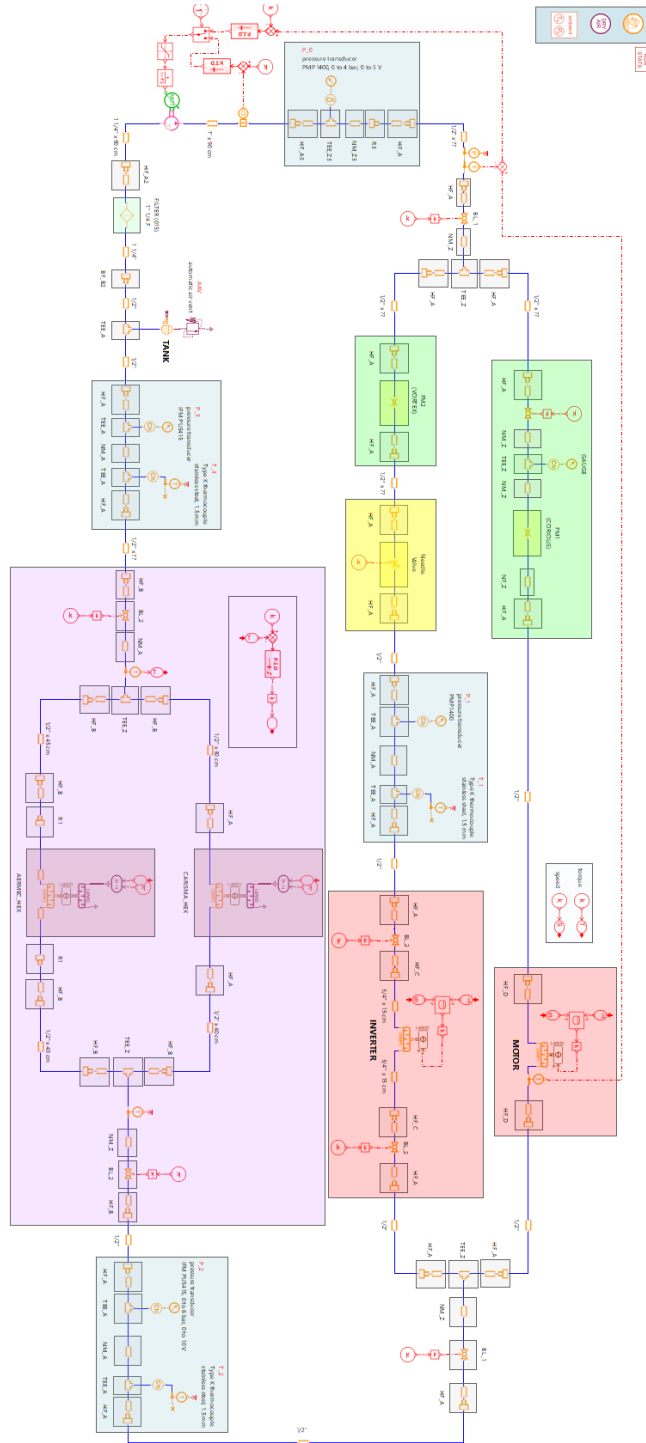


Figure 5. Numerical model of the test-rig reproduced.

3.1. Tank and E-Pump

The system reservoir is modelled as a thermo-hydraulic accumulator, in which the liquid volume is coupled with a pneumatic volume. Air release is reproduced by means of an overpressure valve.

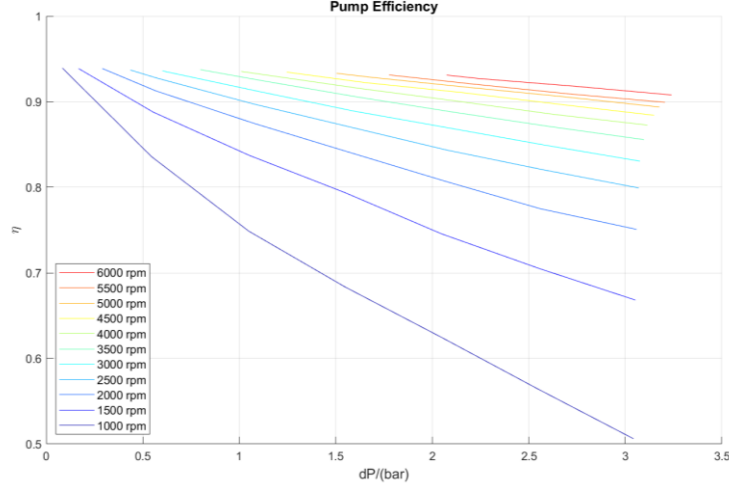


Figure 6. FoT screw pump's efficiency characteristics implemented in the model

The pumping unit is modelled to reproduce the prototypal Fluid-o-Tech positive-displacement pump, with a displacement of 13.14 cm³/rev. The pump efficiency curves implemented in the software are shown in Figure 6. The pump rotational speed is regulated by two separate PID controllers associated with a switching logic based on the simulation time. During the initial phase, the pump speed is regulated according to the error between the flow rate measured by the sensor and the reference value imposed by the user. Once the initialization phase is completed, the control is transferred to the thermal-management PID controller, which adjusts the pump speed to maintain a temperature difference of 10 °C between the inlet and outlet of the electric motor.

The theoretical volumetric flow rate of the pump is given by equation (1), where V is the pump displacement (13.14 cm³/rev) and ω is the shaft speed. The real delivered flow rate is obtained from the volumetric efficiency shown in Figure 6, according to equation (2):

$$Q_{th} = V \cdot \omega \quad (1)$$

$$Q_{real} = Q_{th} \cdot \eta_{vol} \quad (2)$$

As shown in Figure 6, the pump volumetric efficiency drops markedly at the lowest tested rotational speed (1000 rev/min). This behaviour is consistent with the operating principle of a fixed-displacement positive-displacement pump, whose volumetric efficiency is governed by internal leakage flow, which is largely independent of shaft speed. Since the theoretical flow rate is proportional to speed while the leakage flow is not, the leakage represents a

progressively larger fraction of the theoretical flow as speed decreases, and this effect is further amplified at high pressure differentials. It should be noted that, within the range of operating conditions investigated experimentally, the combination of 1000 rev/min and high-pressure differential that would be required to fully expose this efficiency drop was not actually reached; the maximum pressure differential recorded at 1000 rev/min remained well below the level associated with the steepest efficiency reduction.

3.2. Motor and Inverter

The electric motor (Parker GVM210–300) and inverter (Parker GVI-H650) are integrated into the numerical model as simplified heat exchangers. The pressure losses in these two components are implemented through $Q = f(dP)$ curves (Figure 7), derived from processed experimental data collected during the flushing tests conducted in the CeSMA/UniNA laboratories.

The heat exchange in these components is modelled by means of a thermal-hydraulic capacitive element, whose thermodynamic state is obtained from the mass and energy balances of the half-heat-exchanger submodel:

$$\frac{dm}{dt} = dm_1 + dm_2 \quad (3)$$

$$\frac{d(mu)}{dt} = dmh_1 + dmh_2 + \dot{Q} \quad (4)$$

where m is the mass and u is the internal energy in the capacity, dm_1 , dm_2 and dmh_1 , dmh_2 are the mass and enthalpy flow rates at the two hydraulic ports, and $\dot{Q}$ is the heat flux exchanged with the outside, imposed at the opposite side of the element.

In the current configuration, the imposed thermal load can either be set to zero or defined according to one operating point of the electric motor reported in its datasheet.

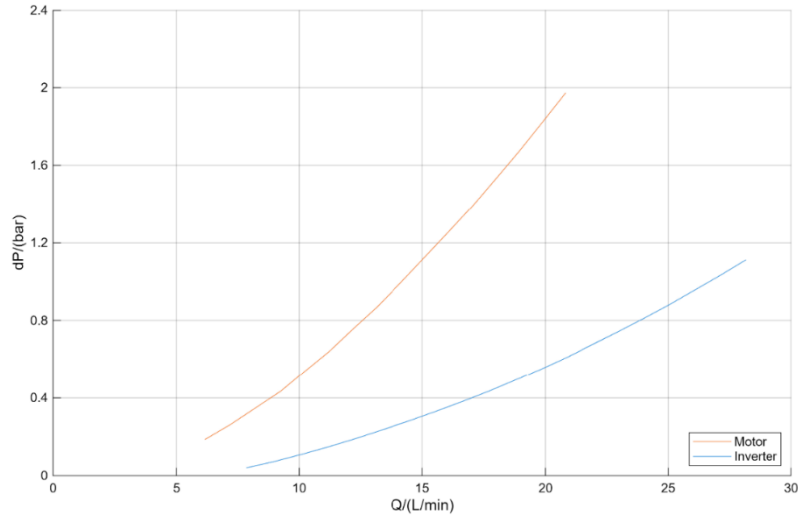


Figure 7. $Q = f(dP)$ curves implemented for motor and inverter (from experimental data)

3.3. Fan Coil Units

The two fan coil units, namely Carisma and Aermec, were implemented in the model as simplified heat exchangers. Each unit consists of two half-heater elements connected through a thermal exchange block, which evaluates the heat transfer between air (cold side) and the coolant (hot side) using the same mass and energy balance formulation described in section 3.2. The air flow rate in the cold side of both heat exchangers is controlled by a PID controller, whose error signal is defined as the difference between the coolant temperature measured downstream of the parallel fan coil connection and the reference value of 25 °C. The pressure drops of the two units are implemented through $Q = f(\Delta P)$ curves, obtained from experimental data and shown in Figure 7.

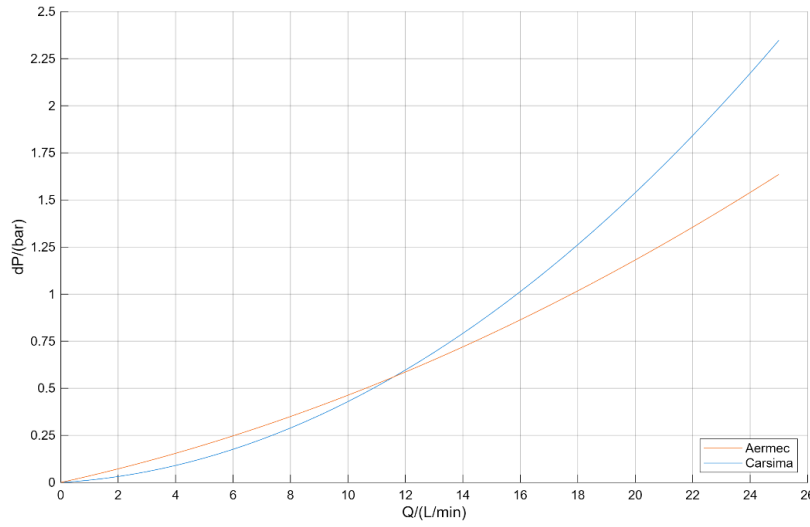


Figure 8. $Q = f(\Delta P)$ curves implemented for Carisma and Aermec (from experimental data)

4. RESULTS AND MODEL VALIDATION

As previously described, the test rig is equipped with a needle valve to regulate the flow distribution between the inverter and motor branches. The valve installed in the experimental setup is the Nieruf NV010003 needle valve. To evaluate the hydraulic behaviour of the system under different flow-splitting conditions, several tests were repeated by varying the valve opening from fully open, corresponding to position 0, to fully closed, corresponding to position 11.5. Each position increment corresponds to one complete turn of the needle valve, up to its complete closure.

As shown in Figure 9 and Figure 10, the flow rate distribution between the two parallel branches, namely the motor branch and the inverter branch, is only slightly affected by the needle valve opening up to position 5. Beyond this threshold, the distribution becomes significantly more uneven, with a marked reduction in the flow rate through the inverter branch. As expected, the total flow rate through the two parallel branches increases with pump rotational speed.

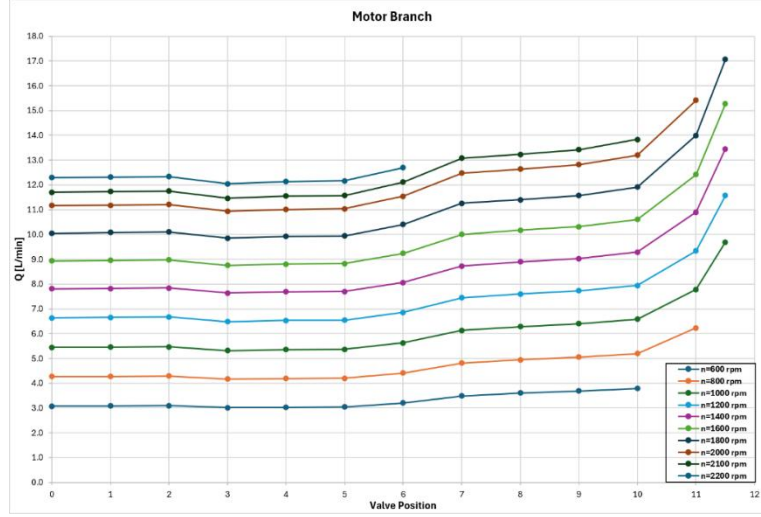


Figure 9. Measured flowrate as function of needle valve opening in the motor branch.

The needle valve was implemented in the model as a variable-section orifice, modelled according to the K_v coefficient of 0.66 reported in the datasheet. However, since the physical valve was manually adjusted during the experimental tests, a calibration procedure was required to correlate each experimental valve position with the corresponding control signal of the variable orifice.

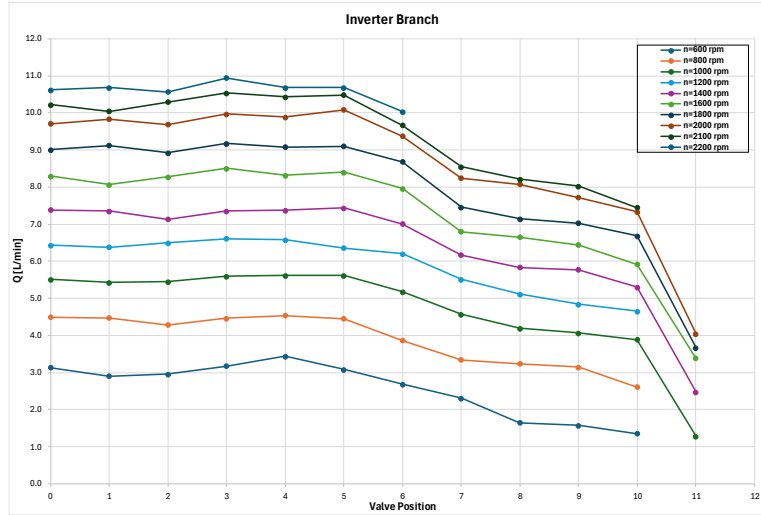


Figure 10. Measured flowrate as function of needle valve opening in the inverter branch.

Since the variable orifice used to represent the physical needle valve does not directly reproduce its mechanical actuation law, an inverse approach was adopted to map each experimental valve position to an equivalent input signal. This mapping was obtained by iteratively adjusting the orifice control signal until the simulated flow distribution between

the two parallel branches matched the experimental measurements. During this procedure, the PID controller regulating the pump speed was used to impose the same total flow rate measured during the experimental tests, calculated as the sum of the flow rates through the two parallel branches.

As shown in Figure 11. and Figure 12., after calibration, the model accurately reproduces the flow rate distribution between the two parallel branches over the investigated operating range. The calibration procedure was carried out and validated for all tested valve positions; however, for brevity, only the results corresponding to position 10 are reported here, selected as a representative example.

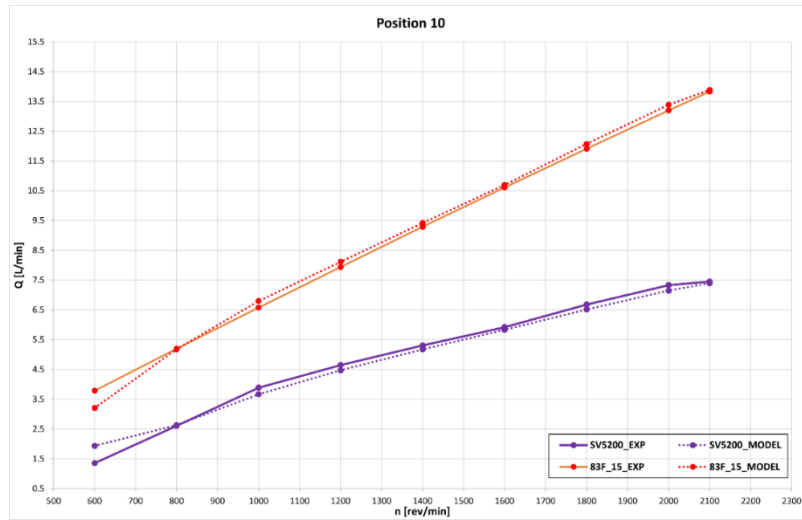


Figure 11. Numerical vs Experimental flowrates comparison with valve position 10.

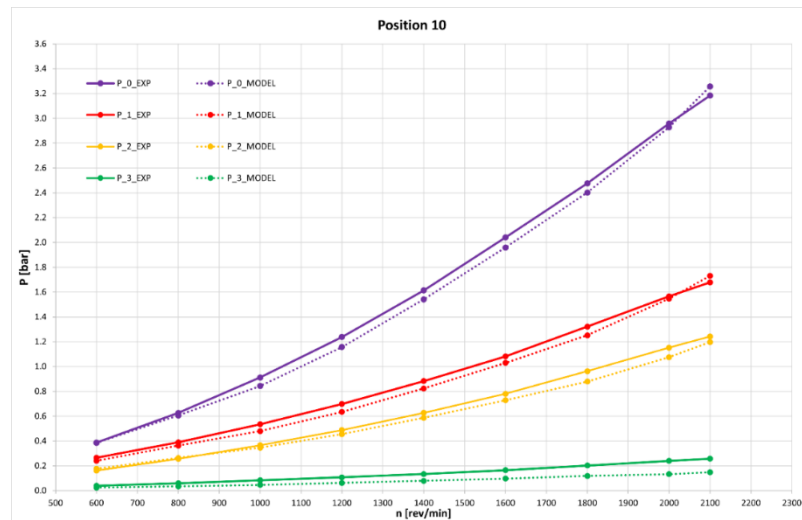


Figure 12. Numerical vs Experimental pressure drops comparison with valve position 10.

A slight divergence between the numerical and experimental flow rates can be observed at the lowest tested pump speed (600 rev/min). This discrepancy is mainly attributed to the pressure-loss characteristics implemented in the model for the pipe fittings, which tend to overestimate local losses at low flow rates compared to their actual behaviour. Since the inverter branch includes a larger number of fittings than the motor branch, this effect is more pronounced on that branch, resulting in a slightly lower simulated flow rate through the inverter branch at the lowest investigated speed.

5. CONCLUSIONS AND FUTURE PERSPECTIVES

This work presents the experimental characterization, numerical modelling, and validation of an innovative thermal management system for a high-performance electric vehicle. The work investigates the use of a prototype positive displacement e-pump as an alternative to conventional solutions that involve centrifugal pumps.

An experimental test rig has been built to replicate the behaviour of a TMS for electric motor and inverter, which have been placed in parallel with respect to the cooling flow, made of a water-glycol mixture (50% vol.). A screw-type e-pump has been tested for water-glycol supply, whose speed is regulated by proprietary software. Flow in the two branches has been regulated using a needle valve. Experimental test-rig measurements confirmed the suitability of the pump for the considered TMS prototype.

A lumped-parameters numerical model has been developed in Simcenter AMESim[®] environment and validated with experimental data provided by the flow tests, in terms of pressure losses and flow distribution between the motor and the inverter branches. Since variable thermal loads can be imposed on the motor and inverter models, the validated model will support future analyses of the circuit thermo-hydraulic behaviour and enable the comparison of alternative layouts. Future work will also address the pump volumetric efficiency at low rotational speed combined with high pressure differentials, a combination of operating conditions only partially covered by the present experimental campaign.

FUNDING

This research was supported by Italian Ministry of Enterprises and Made in Italy (MIMIT) within the project CLIM (Grant No. F/310129/01-04/X56, CUP B89J23006720005).

6. REFERENCES

- [1] L. Chapman, "Transport and climate change: a review," *J. Transp. Geogr.*, vol. 15, no. 5, pp. 354–367, Sep. 2007, doi: 10.1016/J.JTRANGE0.2006.11.008.
- [2] J. Sporkmann, Y. Liu, and S. Spinler, "Carbon emissions from European land transportation: A comprehensive analysis," *Transp. Res. D Transp. Environ.*, vol. 121, p. 103851, Aug. 2023, doi: 10.1016/J.TRD.2023.103851.
- [3] J. Buberger, A. Kersten, M. Kuder, R. Eckerle, T. Weyh, and T. Thiringer, "Total CO₂-equivalent life-cycle emissions from commercially available passenger cars," *Renewable and Sustainable Energy Reviews*, vol. 159, p. 112158, May 2022, doi: 10.1016/J.RSER.2022.112158.

- [4] R. B. Gupta, Deveshkumar, and T. Dutta, "Review of Battery Thermal Management Systems for Electric Vehicles," *Lecture Notes in Mechanical Engineering*, pp. 325–335, 2025, doi: 10.1007/978-981-97-7759-4_27.
- [5] A. Dhanasekaran *et al.*, "Battery thermal management systems for electric vehicles: A review of current status, opportunities, and future challenges," *J. Energy Storage*, vol. 166, p. 122330, Jul. 2026, doi: 10.1016/J.EST.2026.122330.
- [6] G. Previati, G. Mastinu, and M. Gobbi, "Thermal Management of Electrified Vehicles—A Review," *Energies* 2022, Vol. 15, Page 1326, vol. 15, no. 4, p. 1326, Feb. 2022, doi: 10.3390/EN15041326.
- [7] H. Jeon, D. Hyun, H. Lee, S. Son, and J. Han, "Optimization of Blades and Impellers for Electric Vehicle Centrifugal Pumps via Numerical Analysis," *Energies* 2024, Vol. 17, Page 853, vol. 17, no. 4, p. 853, Feb. 2024, doi: 10.3390/EN17040853.
- [8] A. Petrov and R. L. G. Dias, "Variable speed control in centrifugal pumps for closed systems," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 963, no. 1, Nov. 2020, doi: 10.1088/1757-899X/963/1/012026.
- [9] P. Borriello *et al.*, "A fault detection strategy for an ePump during EOL tests based on a knowledge-based vibroacoustic tool and supervised machine learning classifiers," *Meccanica* 2024 59:3, vol. 59, no. 3, pp. 279–304, Feb. 2024, doi: 10.1007/S11012-024-01754-W.
- [10] G. Di Giovine *et al.*, "A Preliminary Design of a Thermal Management System for High-Performance Electric Vehicles: Numerical Simulation and Experimental Validation," *J. Phys. Conf. Ser.*, vol. 3143, no. 1, p. 012068, Dec. 2025, doi: 10.1088/1742-6596/3143/1/012068.
- [11] P. Borriello, E. Frosina, P. Lucchesi, and A. Senatore, "Comparative Analysis of Simulation Methodologies for Spindle Pumps," *Fluids* 2024, Vol. 9, Page 44, vol. 9, no. 2, p. 44, Feb. 2024, doi: 10.3390/FLUIDS9020044.
- [12] E. Frosina, L. Romagnuolo, S. Li Veli, A. Simeone, C. Roselli, and A. Senatore, "A Preliminary Design of a Thermal Management System for High-Performance Electric Vehicles: Numerical Simulation and Experimental Validation," *J. Phys. Conf. Ser.*, vol. 3143, no. 1, p. 012068, Dec. 2025, doi: 10.1088/1742-6596/3143/1/012068.
- [13] P. Borriello, E. Frosina, P. Lucchesi, and A. Senatore, "A Study on a Twin-Screw Pump for Thermal Management Systems by means of CFD using SimericsMP+®: experimental validation and focus on Pressure Pulsation".
- [14] P. Borriello, E. Frosina, A. Senatore, and F. Monterosso, "Numerical Modelling and Experimental Validation of Twin-Screw Pumps Based on Computational Fluid Dynamics using SCORG® and SIMERICS MP+®, " *E3S Web of Conferences*, vol. 312, Oct. 2021, doi: 10.1051/E3SCONF/202131205007.

Numerical analysis of the influence of ball-joint clearance on the elastohydrodynamic performance of slipper interfaces in axial piston motors

Julien Ntede Florent^{1,2}, Ishan Suvarna¹, Lizhi Shang¹, Andrea De Pascale², Davide Moro², Alberto Gatti³

¹*Maha Fluid Power Research Center, Purdue University, West Lafayette, Indiana 47906, USA,*

²*University of Bologna, Department of Industrial Engineering, 40136, Bologna, Italy,*

³*Hydraulic Motor Platform, Bonfiglioli Riduttori S.p.A. Via Enrico Mattei, 12 47122 Forlì (FC)*

Abstract

The tribological behavior of the slipper–swashplate interface is a primary factor limiting the operational lifespan and power density of swashplate axial piston motors. Under high hydraulic loading, the assumption of rigid body motion becomes invalid because elastic deformations within the slipper assembly substantially modify the geometry of the lubricating film. This study examines the typically overlooked influence of non-conformal contact mechanics in the piston-ball/slipper-socket joint and its role in driving deformation of the sealing land. Hertzian contact theory was employed to characterize the pressure distribution across the ball–socket interface, and a series of finite element simulations were conducted in ANSYS to quantify the effects of varying joint clearances. The results demonstrate that increasing the clearance significantly reduces the effective contact area, thereby generating localized stress concentrations. These stress concentrations impose a more concentrated mechanical load on the slipper body, inducing significant elastic warping and geometric nonlinearity across the sealing land interface.

The resulting increase in local contact pressures not only amplifies structural distortion but also may promote higher frictional losses within the ball–socket joint, as higher normal loads between relatively moving surfaces intensify boundary and mixed lubrication conditions. This effect contributes to greater energy dissipation and accelerated wear, further affecting the overall tribological performance of the assembly.

To evaluate the influence of structural deformation on the interface's load-carrying capacity, a hypothetical contact region was defined to simulate the mechanical interaction within the ball-socket joint. This was achieved by isolating the slipper and performing a parametric study in which the contact angle, measured radially from the central axis, was systematically varied. For each discrete angular increment, the specific pressure distribution required to equilibrate the piston's axial force was numerically determined and applied as the primary boundary condition. A comparative analysis shows that a clearance of 15 μm , corresponding to a contact angle of 60°, minimizes sealing land distortion. The resulting deformation field was then incorporated into the MULTICS Multiphysics framework to evaluate its influence on the load capacity of the slipper–swashplate interface.

Keywords: axial piston motor, ball socket interface, slipper swashplate interface, ball clearance

1. Introduction

Axial piston motors are central components in modern hydraulic systems due to their high-power density and dynamic performance. Their efficiency and durability depend strongly on the tribological behavior of internal interfaces, particularly the slipper–swashplate contact, where a micrometer-scale lubricating film must sustain high loads while limiting leakage and friction. As operating pressures and speeds continue to rise, accurately modeling the coupled fluid-structure interactions at this interface has become increasingly important. In most axial piston motor designs, the socket is located on the slipper side and is manufactured from a softer material such as brass, while the ball is integrated into the piston and made from a harder material such as steel. The ball diameter is typically a few micrometers smaller than the socket diameter. Experimental studies have shown that even a minor clearance offset has a significant influence on the performance and lubrication behavior of the slipper interface. For instance, abnormal changes or loosening at the ball-socket joint directly compromise the structural integrity and boundary conditions of the component, leading to premature fatigue or mechanical detachment under high-pressure dynamics [2]. However, despite its practical relevance, there is currently no systematic analysis or numerical model capable of fully explaining the physical mechanisms governing this effect. The objective of this work is therefore to investigate the influence of ball-socket geometry on slipper performance using a novel coupled simulation approach that captures the interaction between joint mechanics and the hydrodynamic behavior of the slipper–swashplate interface.

While early studies treated the slipper–swashplate gap as a classical hydrodynamic bearing, recent research has revealed a far more complex behavior. Wu et al. (2023) demonstrated that transient squeeze film effects induced by pressure pulsations can dominate the lubrication response, significantly altering film thickness and load capacity [1]. Geometric influences have also been emphasized: Haidak et al. (2018) showed that the ratio between the slipper’s inner and outer diameters strongly affects power losses, identifying an optimal geometry that balances leakage control and viscous dissipation [3].

Slipper attitude and microgeometry further influence lubrication performance. Meng et al. (2020) found that small angular deviations can drastically modify pressure distribution and proposed a grooved sealing land design to enhance stability [4]. At the rough-surface scale, Sharif et al. (2012) analyzed mixed and micro EHL regimes, showing that decreasing the lambda ratio leads to asperity interactions and fatigue mechanisms not captured by smooth-surface models [5].

The piston–slipper joint introduces additional mechanical constraints. Shen et al. (2020) used dynamic simulation and ball-in-socket contact modeling to show that piston radius and radial clearance strongly influence contact stiffness, thereby affecting slipper motion and film formation [6]. Material innovation has also emerged as a promising direction: Schlegel and Schmitz (2025) demonstrated that high-performance plastics can replace copper-based slippers while maintaining acceptable deformation characteristics and reducing environmental impact [7]. Finally, Betti et al. (2026) extended Hertzian theory to conformal spherical contacts, providing improved stiffness predictions for geometries typical of piston–slipper joints [8].

Together, previous studies show that slipper–swashplate lubrication is governed by tightly coupled hydrodynamic, structural, and material effects. However, despite substantial progress, current models still struggle to capture the mechanical consequences of wear within the piston–slipper joint, particularly how increasing internal clearance modifies the deformation of the slipper sealing land and, in turn, the hydrodynamic performance of the interface. Most existing approaches treat the joint as perfectly rigid or impose simplified kinematic constraints that do not reflect the evolving geometry of worn components, leaving the relationship between internal joint clearance and external film behavior insufficiently quantified.

The present work addresses this gap by establishing, for the first time, a direct numerical link between ball–socket clearance and the resulting deformation of the sealing land. By parametrically reducing the contact angle, used here as an effective surrogate for increased joint clearance, the study reveals a clear transition in the deformation pattern from a stable, nearly uniform displacement field to a distinctly dished configuration. This shift has direct implications for film thickness distribution, load carrying capacity, and ultimately motor efficiency.

2. Mathematical model

The behavior of the slipper–swashplate interface fundamentally governs the tribological performance of an axial piston motor. Accurately predicting efficiency and durability requires a mathematical model that couples the slipper’s motion and orientation with the fluid dynamics of the lubricating film. During operation, the slipper is subjected to time varying forces that continuously modify its attitude, making the local film thickness a dynamic quantity that must be evaluated at every point on the sealing land. This film thickness distribution is directly governed by the balance of forces acting on the piston–slipper assembly.

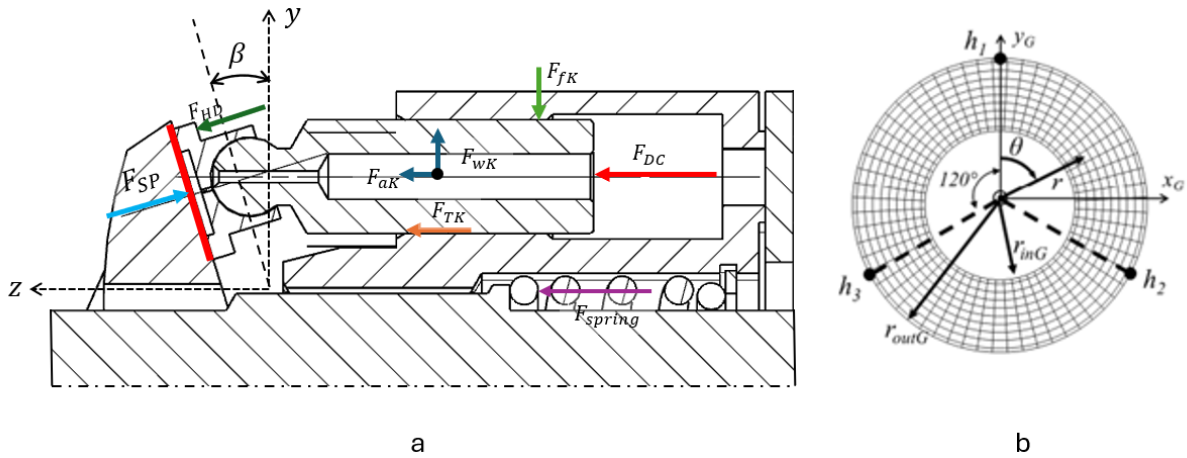


Figure 1: a) Dynamic force equilibrium of the piston-slipper assembly shown in an axial cross section of the motor, with the slipper–swashplate interface highlighted in red, the ball-joint inner lubrication channel, and the plate angle β indicated; b) slipper’s sealing land frontal view with key variables for the oil film thickness in a local polar coordinate system

In the context of the slipper–swashplate interface, only the force components acting along the shaft axis (z coordinate in Figure 1a) contribute to the axial load on the slipper that determines the reaction pressure within the lubricating film. For this reason, the equilibrium is formulated exclusively along the shaft axis,

where the dominant contributions arise from the hydraulic force generated by the chamber pressure F_{DC} , the inertial force F_{aK} associated with the reciprocating motion of the piston–slipper assembly, and the friction force F_{TK} transmitted through the piston and its bore. These three components constitute the primary load that must be supported by the hydrodynamic film at the slipper interface and therefore govern the resulting pressure distribution and tribological behavior. Based on the vectors defined in Figure 1a, the dynamic behavior of the piston-slipper assembly is governed by a single force balance equation along the piston axis. By accounting for both the external operating loads, fluid film reaction and the inertial effects, the complete dynamic equilibrium equation can be expressed as:

$$F_{DC} + F_{TK} + F_{HD} - F_{SP} \cdot \cos\beta + F_{aK} = 0 \quad (1)$$

Where $F_{HD} = F_{spring}/z$.

In modelling the squeeze film effect, only the oil pressure acting on the piston and the swashplate reaction are considered, as the remaining forces affect only the magnitude of the compression force and do not influence the squeeze film mechanism itself.

The reaction force F_{SP} exerted by the swashplate on the slipper is the sum of two distinct contributions: the steady state hydrostatic support and the squeezed film effect, which are detailed in the following.

- The hydrostatic support force is governed by the pressure differential across the slipper's sealing land, which acts as the primary load bearing mechanism under steady state conditions. The fluid from the displacement chamber flows radially through the slipper's sealing land toward the low pressure region, generating a pressure gradient across the interface. The resulting pressure distribution over the sealing land, when integrated, yields the total hydrostatic lift force that stabilizes the slipper assembly:

$$F_{hyd} = \frac{\pi}{2} \left(\frac{r_{outG}^2 - r_{inG}^2}{\ln(r_{outG}/r_{inG})} \right) p_{DC} \quad (2)$$

where r_{outG} and r_{inG} are external and internal radii of the sealing land, respectively, see Figure 1b, and p_{DC} is the chamber pressure.

- The squeeze film support force acts as a dynamic stabilizing mechanism that responds to the rapid, cyclic fluctuations inherent in hydraulic motor operation. As the piston reciprocates, the pressure in the chamber varies continuously, causing the slipper to move toward or away from the swashplate with a velocity $v = \frac{dh}{dt}$. Due to the lubricant's viscosity and near incompressibility, this motion generates a squeeze effect, producing a localized high-pressure film that provides additional hydrodynamic lift during pressure peaks expressed in polar coordinates as:

$$F_{squeeze} = \int_{r_{inG}}^{r_{outG}} \int_0^{2\pi} p r dr d\theta \quad (3)$$

The fluid transport through the annular sealing land is modeled by considering a micro-elementary ring at a radius r with a radial width dr . Assuming steady, laminar, and isothermal flow, the volume flow rate q through this ring is governed by the pressure gradient term $\frac{dp}{dr}$.

The flow rate through the micro element ring at radius r is expressed as:

$$dq = -\frac{h^3}{12\mu} \left(2\pi r \frac{dp}{dr} \right) dr \quad (4)$$

Following the analytical formulation presented in [1], the pressure distribution along the radial land can be expressed analytically as:

$$p = \frac{3\mu}{h^3} \frac{dh}{dt} \left[(r^2 - r_{inG}^2) - (r_{outG}^2 - r_{inG}^2) \frac{\ln(r/r_{inG})}{\ln(r_{outG}/r_{inG})} \right] \quad (5)$$

Equation 3 can be rewritten as:

$$F_{squeeze} = \frac{3\pi\mu}{2h^3} \frac{dh}{dt} \left[r_{outG}^4 - r_{inG}^4 - \frac{(r_{outG}^2 - r_{inG}^2)^2}{\ln(r_{outG}/r_{inG})} \right] \cos\beta \quad (6)$$

The lubrication gap h deviates from a perfectly parallel configuration due to a complex nonlinear interaction between mechanical tilting moments, structural elasticity, and localized thermal expansion [3]. Consequently, in swashplate motors, the slipper orientation is inherently dynamic; continuously evolving in response to the transient pressure field and kinematic constraints.

The relationship between the slipper's tilt angle and interface performance is governed by a fundamental trade-off:

- Hydrodynamic enhancement: within a certain range, an increase in tilt angle promotes the formation of a converging-diverging wedge, enhancing the hydrodynamic lift and increasing the load carrying capacity of the oil film.
- Instability and leakage: conversely, excessive tilt leads to a pronounced film thickness gradient and an uneven pressure distribution. This condition increases the risk of local film rupture and significantly amplifies peripheral leakage, thereby reducing the motor's volumetric efficiency.

Furthermore, the mean film thickness introduces an additional optimization challenge. While a reduction in gap height enhances the squeeze film effect, providing damping and load carrying capacity, it also increases the viscous shear stress within the lubricant. This results in higher frictional losses and increased mechanical power dissipation. Most critically, excessively low film thickness shifts the lubrication regime toward boundary conditions, significantly increasing the likelihood of asperity contact and surface degradation.

To ensure the structural reliability of the axial piston motor, a stable lubrication regime must be maintained to prevent asperity contact between the slipper and the swashplate. This is achieved by ensuring that:

- The combined hydrodynamic and hydrostatic lift forces must exceed the net axial load acting on the slipper:

$$F_{squeeze} + F_{hyd} > F_{DC} + F_{ak} + F_{TK} + F_{HD} \quad (7)$$

- The minimum film thickness must satisfy the following condition:

$$h_{min} > \Lambda \cdot (\sigma_{slipper}^2 + \sigma_{sp}^2)^{1/2} \quad (8)$$

where Λ , referred to as the lambda ratio, and varies over the range 0.1 – 2.0, is a parameter used to characterize the transition from full film lubrication to mixed and micro elastohydrodynamic lubrication regimes, and their associated fatigue damage [4].

The ball-socket clearance is a key design parameter governing the interfacial equilibrium of the slipper assembly. In addition to defining the kinematic range of the slipper tilt, it significantly affects the internal pressure distribution within the joint. Under high operating pressures, an improper clearance leads to localized elastic deformation, resulting in a crowning of the slipper sealing lands. Both insufficient and excessive clearance cause geometric distortion of the sealing land, disrupting the hydrostatic pressure balance and inducing fluid film instability. Therefore, the joint clearance becomes a critical factor in the transition from stable hydrodynamic lubrication to contact dominated regimes, characterized by asperity interaction and increased friction.

3. Numerical model of the ball-joint contact and simulation boundary conditions

To quantify the mechanical interactions governing load transfer from the piston ball to the slipper socket, the joint is modeled as a conformal spherical contact based on an extension of Hertzian contact theory. This approach is used to evaluate the resulting contact pressure distribution and the associated elastic distortion of the slipper, which directly affects the gap geometry [7]. The model is based on the following assumptions and boundary conditions:

- Linear elasticity and constitutive behavior: the piston ball and the slipper socket are modeled as homogeneous and isotropic continua. The material response follows Hooke's law, assuming that the stresses remain within the elastic regime and no plastic deformation occurs.
- Small displacement assumption: the contact radius is assumed to satisfy the condition $\delta \ll a \ll R_1$ (see Figure 2 for the meaning of the geometrical quantities). By ensuring that deformation remains localized, this justifies the use of small strain theory, whereby the global geometry of the components is not significantly altered.
- Frictionless contact interface: the contact between the ball and the socket is assumed to be frictionless ($\mu = 0$), allowing the analysis to focus solely on normal contact stresses and their role in transferring axial piston loads into structural deformation of the slipper sealing lands.

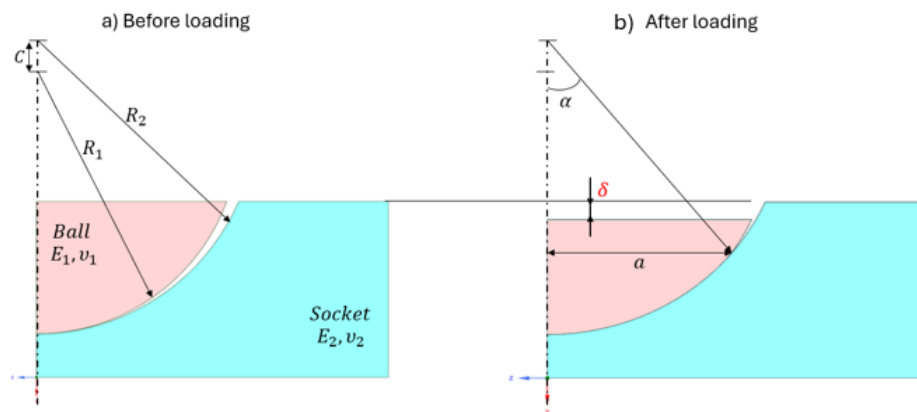


Figure 2: Conformal contact between two spherical bodies

While the frictionless assumption simplifies the formulation to isolate normal contact stresses, it introduces a limitation. In physical applications, hydrodynamic shear and boundary friction within the ball-socket joint generate tangential tractions. These tractions exert a micro tilting moment on the slipper body, which can slightly alter the symmetric bending of the sealing land.

Figure 2 illustrates the contact interface between the piston ball and the slipper socket, characterized by their respective radii of curvature, R_1 and R_2 . According to Hertzian contact theory, the normal force F_n applied at the ball-and-socket interface results in a characteristic load displacement relationship. The corresponding compressive deformation δ , occurring along the loading direction, is given by [5]:

$$\delta = \left(\frac{9F_n^2}{16R'E'} \right)^{1/3} \quad (9)$$

Where $E' = 2 \left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \right)^{-1}$ is the equivalent elastic modulus, $R' = \frac{R_1 R_2}{R_2 - R_1} = \frac{(R_1 + C)R_1}{C}$ is the equivalent radius and C is the ball-joint clearance.

Based on contact equilibrium, the radius of the contact area a depends on the applied load and the geometric compatibility of the contacting surfaces, as given by:

$$a(\delta) = \sqrt{R'\delta} \quad (10)$$

The contact pressure distribution is not uniform and follows a semi-ellipsoidal profile. The maximum pressure, occurring at the center of the contact area, is a critical parameter for assessing the onset of local yielding. It is given by:

$$p_{max} = \left(\frac{6F_n E'}{\pi^2 (R')^2} \right)^{1/3} \quad (11)$$

p_{max} depends on the applied normal force as well as on the material and geometric properties. If it exceeds the yield strength of the socket material, localized plastic deformation may occur, permanently altering the socket geometry. As the ball is pressed into the socket, the resulting pressure is transmitted through the slipper thickness, leading to a slight deflection of the outer sealing lands.

To investigate the sensitivity of the slipper's structural integrity to joint tolerances, a high-fidelity finite element model was developed in ANSYS Mechanical. The model includes only the piston ball, socket, and slipper, as the deformation of the sealing land is governed by local contact mechanics rather than the global motor structure. The surrounding components, namely the cylinder block, the swashplate, and the housing, are orders of magnitude stiffer and therefore behave as rigid supports relative to the slipper. Modeling the entire motor would add computational cost without changing the deformation field of interest. For this reason, the cylinder side surface is fixed, while the slipper is allowed to translate in the plane but is constrained in the piston direction and in rotation, reproducing the effective kinematic environment imposed by the hydrodynamic film during operation. The simulation aims to characterize the mechanical coupling between the piston-ball interface and the resulting deformation of the slipper sealing land. A parametric study was conducted by keeping the socket geometry constant while varying the piston ball diameter, in order to obtain different ball-joint clearance values. To ensure accuracy and mesh independence of the structural quantities of interest, a mesh convergence study was performed. The final model utilizes high-order tetrahedral elements with an average global element size of 1.2 mm. Due to the

high stress gradients and local contact conditions, local refinement was implemented at the ball-socket contact zone, where the element size was reduced to 0.1 mm. The mesh sensitivity evaluation demonstrated that further refining the contact zone element size to 0.08 mm resulted in less than a 1.2% variance in peak contact pressure and less than a 0.5% change in the maximum sealing land displacement, confirming that the main structural results are robustly mesh independent.

The piston–slipper relative motion was assumed to be no relative rotation for the structural finite element model. While physical slipper assemblies experience micro rotation relative to the piston during a complete shaft cycle, the macroscopic elastohydrodynamic loading acting on the slipper sealing land is axisymmetrically dominated during the high-pressure phase. Consequently, the localized stress concentrations and resulting dishing deformation profiles are rigidly linked to the slipper body itself and rotate concurrently with it relative to the swashplate interface. By locking the relative angular orientation between the piston ball and slipper socket during the static snapshot simulations, the model accurately isolates the structural compliance mode without sacrificing the fidelity of the transferred boundary conditions to the MULTICS framework.

The numerical model relies on standard industrial material configurations to accurately replicate load transfer and structural deflection. Table 1 lists the specific isotropic elastic properties applied to the components in the finite element framework. Among these parameters, the elastic modulus (E) of the bronze socket is the most critical for the subsequent analysis, as its lower stiffness directly governs the structural compliance, warping, and dishing characteristics of the slipper sealing land under high hydraulic forces.

Table 1: Material properties

<i>Material</i>	<i>E(GPa)</i>	<i>ν</i>
Structural steel	200	0.3
Bronze CuSn3Zn1	120	0.345

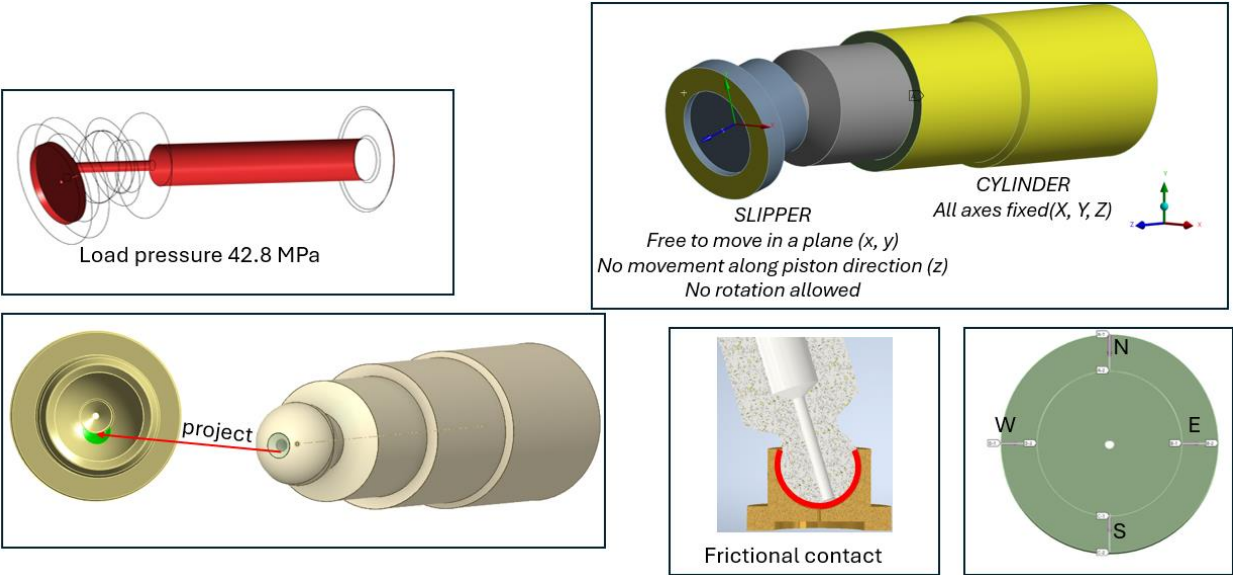


Figure 3: Simulation framework for structural compliance analysis, illustrating loading conditions and the location of node paths used for result extraction.

4. Results and discussion

i) Parametric FEM analysis of the joint clearance effect

The simulation results show a clear correlation between the joint clearance C and the interfacial contact pressure. This behavior is driven by the reduction in the effective contact area as the piston ball diameter decreases.

The contour plots in Figure 4 illustrate the local pressure distribution in the ball-socket joint and the peak pressure for four clearance levels, namely $0\ \mu\text{m}$, $15\ \mu\text{m}$, $35\ \mu\text{m}$ and $100\ \mu\text{m}$:

- $C = 0\ \mu\text{m}$: The contact pressure is distributed over a wider and more uniform area, resulting in relatively low peak pressure.
- $C = 100\ \mu\text{m}$: The contact area becomes highly localized near the apex of the socket. This concentration leads to a significant increase in peak pressure, reaching values as high as $804.79\ \text{MPa}$.

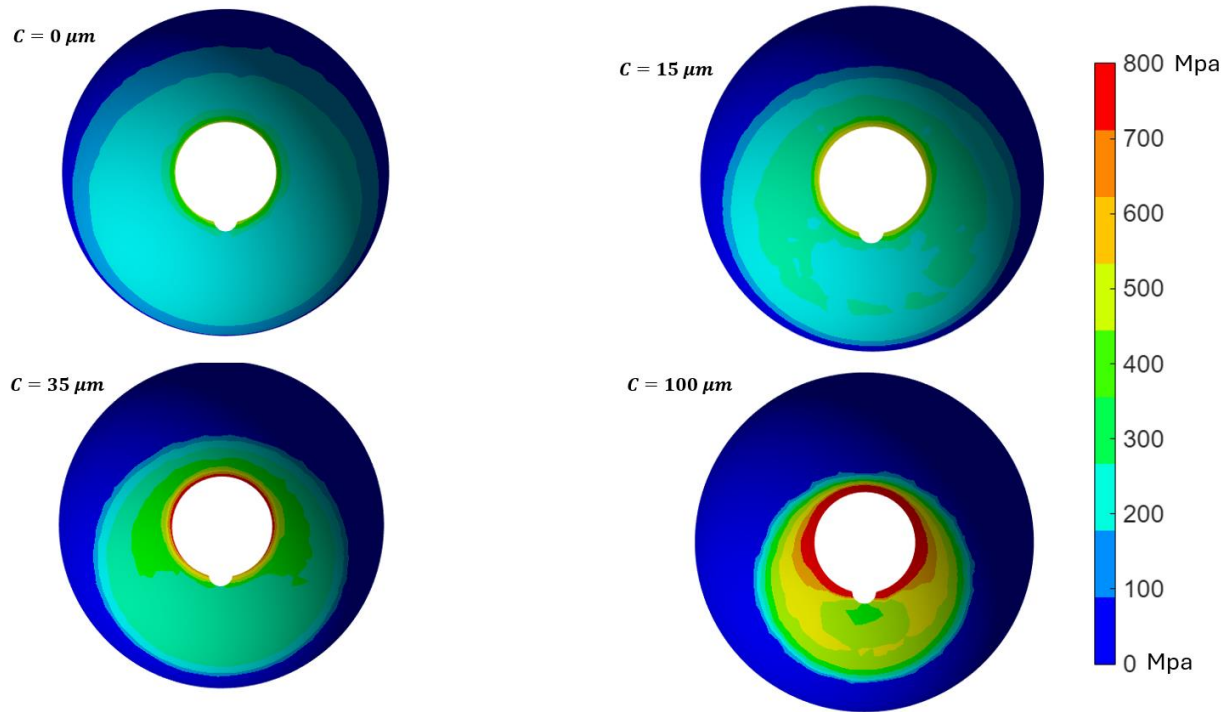


Figure 4. Comparative analysis of contact pressure distribution in the ball-socket interface: evolution of stress concentration and contact area reduction as a function of clearance

As clearance increases, the conformity between the ball and the socket diminishes, reducing the load bearing contact area. Since the axial hydraulic force F_{DC} remains constant across all cases, this reduction in contact area results in higher stress concentrations required to maintain equilibrium. Consequently, the peak contact pressure p_{max} increases accordingly. The increase in p_{max} , which approaches nearly three times its initial value, indicates a substantially higher local load on the bronze slipper and suggests that the

material may approach its elastic limits in the most stressed regions. The associated pressure concentration also leads to non-uniform deformation of the slipper body, which in turn affects the stability of the lubrication film at the swashplate interface. The resulting deformation pattern along the sealing land exhibits a characteristic tilted shape (see Figure 5): the inner radius is displaced slightly downward due to the concentrated load transmitted through the ball–socket interface, while the outer radius shows a modest upward deflection caused by bending of the thin sealing land. This combination of central compression and peripheral lift-off represents the dominant deformation mode observed in the analysis.

This structural distortion also affects the hydraulic and tribological behavior of the interface. The slight lift of the outer radius increases the local gap height, promoting additional leakage through the sealing land. Conversely, the reduced film thickness near the inner radius brings the interface closer to mixed or boundary lubrication conditions, where the likelihood of localized wear and frictional heating becomes more significant. Together, these effects illustrate how even modest elastic deformation can subtly modify the operating conditions of the slipper–swashplate interface.

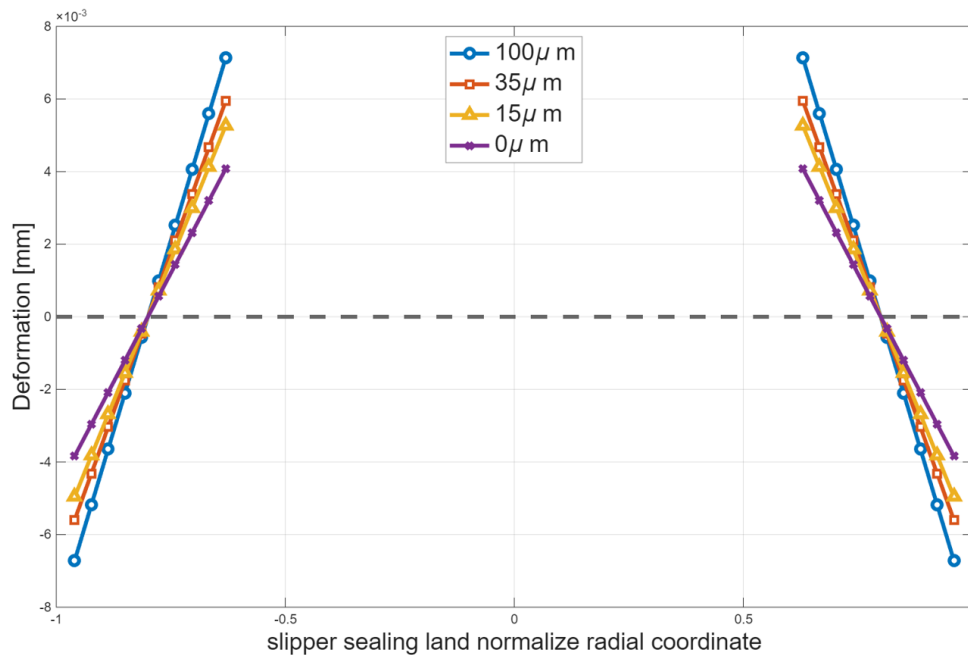


Figure 5. Structural deformation of the slipper sealing land

ii) Elastohydrodynamic and Structural Coupling in Piston-Slipper Interface Simulations

To quantify these coupled effects, the structural deformation is integrated into a fluid dynamic simulation using MULTICS, version 4.1.0 [15]. This is a multiphysics simulation environment for hydraulic machines developed at the Maha Fluid Research Center at Purdue University. Hydraulic units such as pumps and motors involve multiple interacting physical domains fluid chambers, lubricating films, and solid bodies as well as phenomena such as elastic deformation, fluid friction, and pressure induced motion. Multics incorporates dedicated physics models for each of these behaviors, and its modular architecture enables the addition and coupling of new models as needed. The tool, which has been successfully applied to a wide range of hydraulic units, including gear machines, gerotors, vane pumps, and axial piston machines [16], is used here to investigate the coupled fluid-structure problem at the slipper-swashplate interface,

under dynamic conditions (Figure 6). In this study, the slipper–swashplate interface is modeled using the Reynolds equation, extended to account for asperity contact between the surfaces. The fluid dynamics within the pump/motor chambers and orifices are represented using a lumped parameter model, which significantly reduces computational cost compared to full CFD. The dynamics of the rotating components are also included and coupled with the fluid models, ensuring that all relevant interactions influencing the slipper–swashplate interface are captured. A dedicated timeline interpolation scheme ensures continuous data exchange between the models without increasing the overall simulation time.

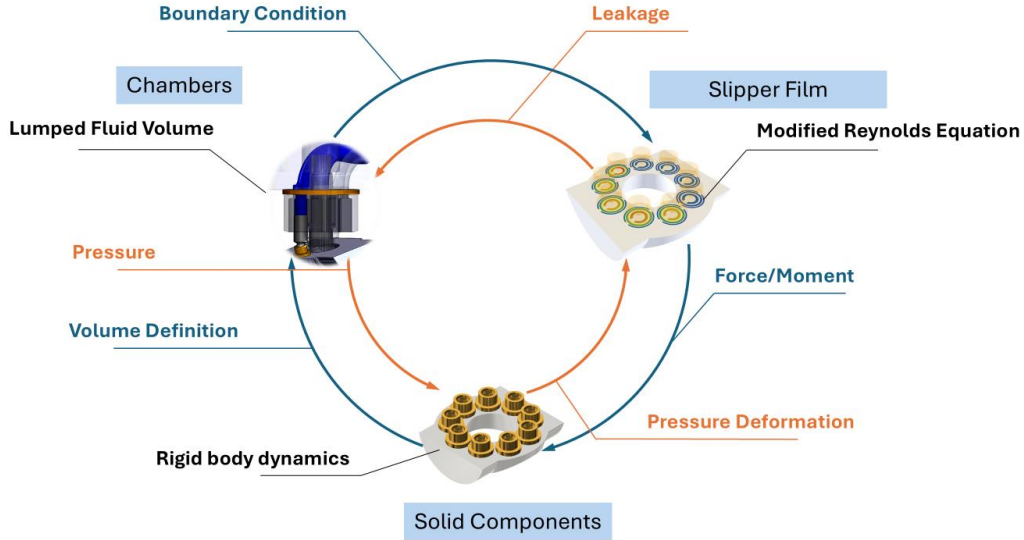


Figure 6. Multics Workflow for Coupled Fluid–Structure–Dynamics Simulation of the Slipper–Swashplate Lubrication Interface

Within Multics, the influence matrix method [17] is used to represent the elastic response of the slipper. This approach requires defining a representative contact area over which the piston ball pushes against the socket. While the effective contact area A_{eff} is directly obtained from the FEM simulations, the corresponding projected contact radius a (see Figure 2) needed for comparison with Hertzian analytical formulations is not explicitly provided [7]. To establish this connection, the contact between the piston ball (radius R_1) and the socket is approximated as occurring over a finite circular region characterized by a contact radius a , associated with a contact angle α . Assuming a constant piston force, the load is distributed over this effective area, allowing the corresponding contact pressure to be evaluated consistently in both the FEM based and Hertzian frameworks.

$$p_{contact} = \frac{F_{DC}/\cos\beta}{A_{eff}} \quad (12)$$

The following geometric relations define the relationship between the simulated contact area and the projected contact radius:

$$A_{eff} = 2\pi R_2^2(1 - \cos\alpha) \quad (13)$$

$$a = R_2 \sin\alpha \quad (14)$$

Where α represents the contact half angle, defined as the angle between the axis of symmetry of the joint and the line connecting the center of the undeformed spherical socket to the boundary of the contact region (see Figure 2).

The proposed methodology maps physical joint clearance to an equivalent contact angle, establishing an effective geometric surrogate within the MULTICS framework. Although this approach leverages a statically equivalent, uniform pressure distribution over a predefined circular boundary, a primary limitation is its inability to dynamically capture the non-uniformity and inherent asymmetry of the contact patch. This limitation is particularly critical during high-speed tilting or transient dynamic strokes, when the contact topology exhibits significant temporal variation. Mechanistically, this correlation serves as a structural transfer function that remains robust across a wide range of operating conditions, because the load boundary profile is strictly dictated by the geometric conformity of the ball-socket pair rather than by fluctuating operating pressures. However, its envelope of validity is bounded by high-pressure operations that ensure continuous, positive contact between the ball and socket. Under severe transient dynamic reversals or low-pressure suction strokes, where inertial forces can induce micro separation or partial joint liftoff, this static equivalence must be augmented with dynamic adjustments to accurately resolve the evolving contact topology.

Table 2. Input for the parametric FEM study of the slipper

$2 * \alpha$ (°)	a (mm)	A_{eff} (mm ²)	$p_{contact}$ (MPa)
30	3.5	41.25	358.38
45	4.95	90.20	163.90
60	6.06	153.93	96.04
75	6.76	228.20	64.78
90	7	307.87	48.02

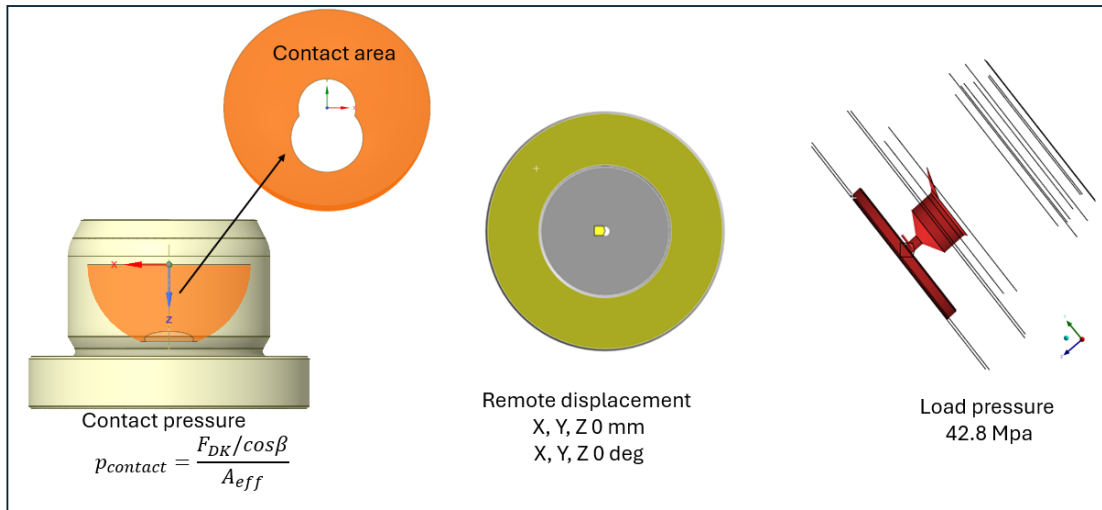


Figure 7. Contact Area Definition and Pressure Calculation Framework for the Ball–Socket Interface

A series of structural simulations was performed on the slipper to assess how variations in joint geometry influence the deformation of the sealing land. In this parametric study, the contact radius and the contact angle were selected as the primary independent variables. The objective was to determine which angular increments, reported in Table 2, produce deformation levels comparable to those obtained in the

assembly level analysis for different radial clearances. Establishing this correspondence enables a direct comparison between the two parameters, ball-socket clearance and contact angle, in terms of their effect on the overall structural response of the slipper.

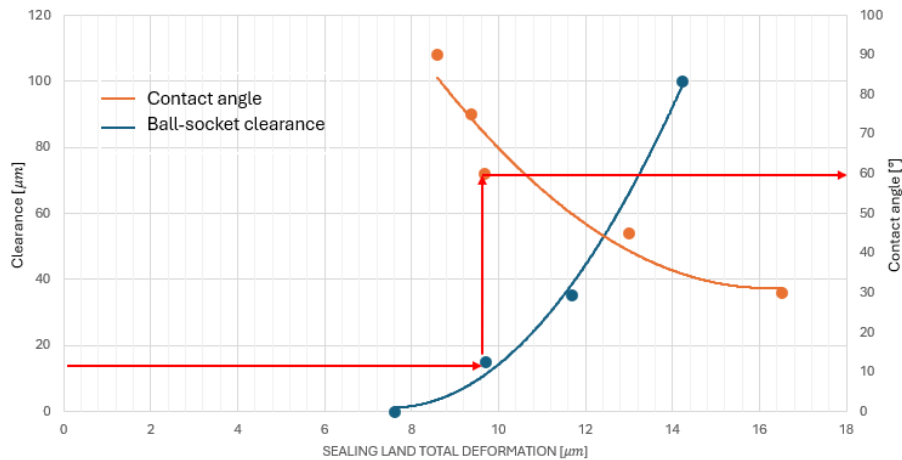


Figure 8. Mutual Influence of Clearance and Contact Angle on Slipper Sealing Land Deformation (simulation points and trend lines)

The comparison between the two parameters reveals that increasing clearance leads to a monotonic rise in sealing-land deformation, whereas increasing the contact angle has the opposite effect, progressively reducing deformation. By cross-referencing the two trends, an equivalence point emerges: a clearance of roughly 15 μm produces the same deformation as a 60° contact angle, both resulting in a total deformation of approximately 9.7 μm . This establishes a direct quantitative relationship between the two geometric parameters and provides a consistent basis for translating joint wear, i.e. clearance increase, into an equivalent change in contact angle. To further illustrate how the assumed contact geometry influences the structural response, Figure 9 compares the deformation field obtained using a full socket-contact assumption with that derived from the influence-matrix approach, where the contact radius corresponds to a 60° engagement angle.

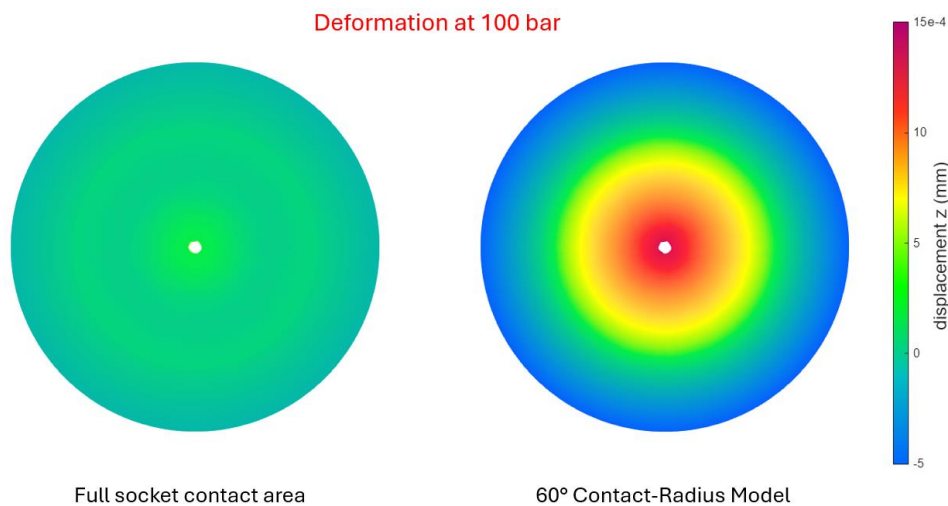


Figure 9. Deformation Response at 100 bar: Full Contact Patch vs. Contact Radius Derived from 60° Engagement

The comparison reveals a clear shift in the location and magnitude of the maximum deformation. Under the full contact assumption, the deformation remains nearly uniform across the sealing land, with only a slight central depression. In contrast, when the contact radius is restricted to the 60° region, the deformation becomes strongly localized: the maximum displacement occurs near the center of the slipper, directly under the reduced contact patch where the load is applied. This concentrated deformation produces a steeper radial gradient, with progressively smaller displacements toward the outer radius. The resulting profile is consistent with the tilting mode identified earlier, where central compression is accompanied by peripheral lift-off.

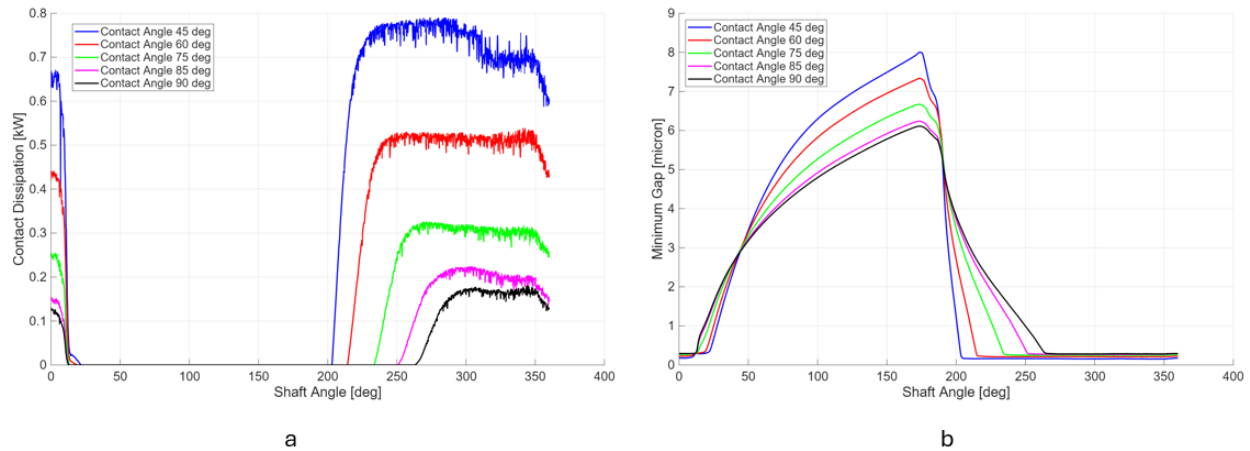


Figure 10. a) Angular Evolution of Contact Dissipation b) Minimum Gap Height Over a Revolution for Different Contact Angles

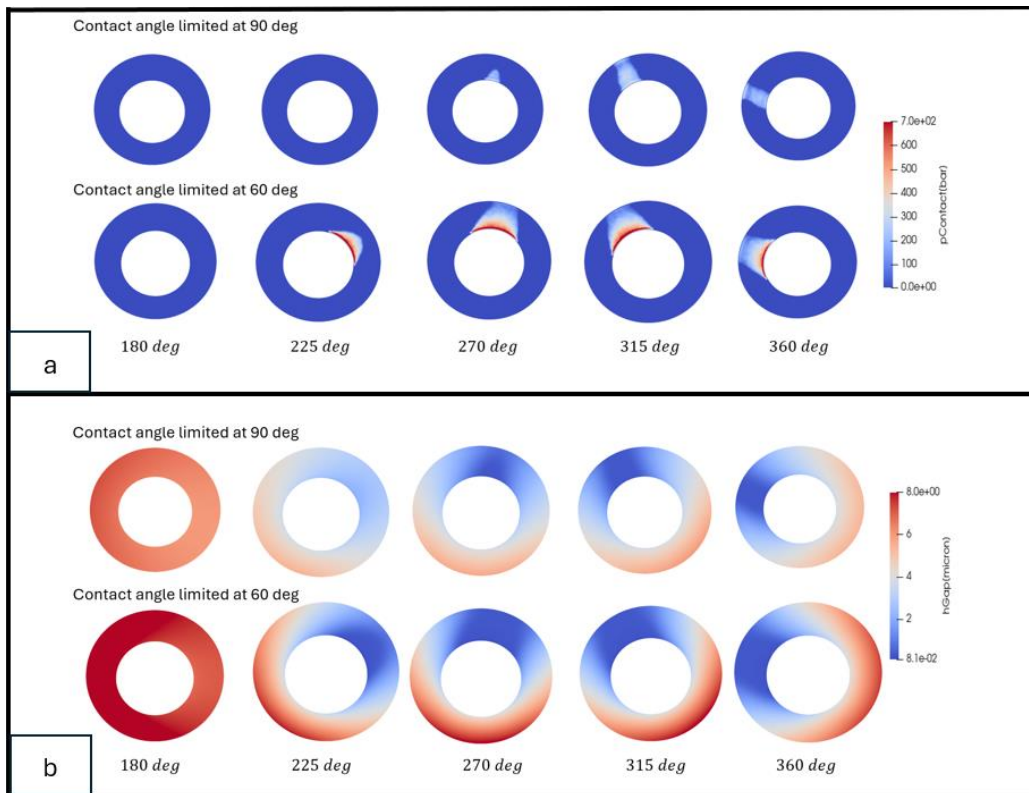


Figure 11. Angular Evolution of a) Contact Pressure, b) Film Thickness in the Slipper Pad Interface During the High-Pressure Stroke for a 90 degrees & 60 degrees Limited Contact Area

Figures 10 and 11 illustrate the strong coupling between film thickness, contact pressure development, and energy losses as the slipper rotates during the high-pressure stroke. Figure 10 shows that the minimum gap exhibits a clear dependence on the imposed contact area angle. When the contact area becomes more restricted, the minimum gap decreases more sharply and approaches near contact conditions over a wider angular interval. Correspondingly, the contact dissipation remains low when the film is still relatively thick but rises significantly once the gap collapses and stays elevated throughout the portion of the stroke where the interface operates in mixed or boundary lubrication.

Figure 11 further confirms this behavior by showing that the regions of minimum film thickness coincide with the angular positions where the mechanical load is concentrated. In these zones, the hydrodynamic pressure increases within the converging wedge and diminishes toward the diverging region, revealing a pronounced angular dependence of both pressure generation and film formation. The comparison between the 90° and 60° contact area constraints also highlights that a more limited contact area intensifies the pressure peaks and reduces the film thickness, thereby promoting higher dissipation and more severe lubrication conditions.

A smaller contact angle makes the slipper more unstable because the reduced load-carrying area cannot maintain a sufficiently thick hydrodynamic film. When the motor operates at high pressure, the slipper tends to tilt, locally squeezing out the oil film. This collapse of the lubricating wedge forces the interface into mixed or boundary lubrication, where the edges experience concentrated contact pressure. Consequently, frictional dissipation increases, and the risk of severe wear becomes significantly higher.

5. Conclusion

This work has demonstrated that the clearance between the piston ball and the slipper socket plays a decisive role in governing the mechanical and tribological behavior of the slipper swashplate interface. The coupled simulation framework developed here shows that even small variations in ball–socket clearance can significantly alter the slipper tilt, load distribution, and lubrication conditions throughout the high-pressure stroke. These geometric effects directly influence film formation, the balance between hydrodynamic and contact pressures, and the resulting energy-dissipation mechanisms. The results further reveal that the angular evolution of minimum film thickness, hydrodynamic pressure, and contact dissipation is strongly affected by the imposed 60° contact area constraint, underscoring the interface’s sensitivity to both design and operating parameters. Overall, the findings confirm that ball–socket clearance is a critical design variable with direct implications for efficiency, wear, and long-term reliability in axial piston machines, and they provide a foundation for more accurate predictive models and improved motor optimization strategies.

Nomenclature

- F_{SP} Swashplate force [N]
- F_{HD} Hold on force [N]
- F_{aK} inertial piston force [N]
- F_{WK} Weight or normal reaction force [N]
- F_{TK} Friction force [N]
- F_{DC} Piston force [N]
- F_{SP} Swashplate reaction force [N]

- F_{spring} Spring Force [N]
- β Tilt angle between the slipper and swashplate [°]
- μ (dynamic viscosity) Dynamic viscosity of the lubricant (Pa·s)
- h (film thickness) Instantaneous fluid-film thickness between slipper and swash plate [m]
- dh/dt (time derivative of film thickness) Rate of change of film thickness [m/s]
- r_{outG} (outer radius of the gap region) Outer radius of the annular lubrication region [m]
- r_{inG} (inner radius of the gap region) Inner radius of the annular lubrication region [m]
- $\sigma_{slipper}$ RMS roughness of the slipper surface [m]
- σ_{sp} RMS roughness of the swash-plate surface [m]
- Λ safety factor/separation parameter [adim]

References

1. Wu, J., Ma, L., An, G., & Han, H. (2023). *Squeeze film effect and its influence on the lubrication characteristics of slipper oil film under pressure pulse*.
2. Li, D., Ma, X., Wang, S., Lu, Y., & Liu, Y. (2024). *Failure analysis on the loose closure of the slipper ball-socket pair in a water hydraulic axial piston pump*. *Engineering Failure Analysis*, 155, 107718. <https://doi.org/10.1016/j.engfailanal.2023.107718>
3. Haidak, G., Wang, D., Shiju, E., & Liu, J. (2018). *Study of the influence of slipper parameters on the power efficiency of axial piston pumps*.
4. Meng, X., Ge, C., Liang, H., Lu, X., & Ma, X. (2020). *Lubrication characteristics of the slipper–swashplate interface in a swashplate-type axial piston pump*. *Proceedings of the Institution of Mechanical Engineers, Part J: Journal of Engineering Tribology*.
5. Sharif, K. J., Evans, H. P., & Snidle, R. W. (2012). *Modelling of elastohydrodynamic lubrication and fatigue of rough surfaces: The effect of lambda ratio*.
6. Shen, H., Zhou, Z., Guan, D., Liu, Z., Jing, L., & Zhang, C. (2020). *Dynamic contact analysis of the piston and slipper pair in axial piston pump*.
7. Schlegel, F., & Schmitz, K. (2025). *Geometric design of axial piston machine slipper-bearings made of high-performance plastics*.
8. Betti, A., Forte, P., & Ciulli, E. (2026). *An extension of Hertz's formula for the stiffness of conformal spherical contacts*.
9. Harris, R. M., et al. (1963). *The design and performance of hydrostatic slipper bearings*. *Proceedings of the Institution of Mechanical Engineers*.
10. Hooke, C. J., & Kakoullis, Y. P. (1978). *The effects of non-parallelism on the performance of slipper bearings*. In *Proceedings of the 4th International Fluid Power Symposium*.
11. Schenck, A., & Ivantysynova, M. (2016). *A transient thermo-elastic swashplate and slipper model*. *International Journal of Fluid Power*.
12. Tang, H., et al. (2019). *Impact of manufacturing tolerances on the lubrication performance of the slipper–swashplate interface*. *Tribology International*.
13. Chao, Q., et al. (2015). *Effects of ball-joint clearance on the slipper behavior*. *Journal of Mechanical Science and Technology*.
14. Canbulut, F., et al. (2014). *Effects of slipper geometry on the performance of an axial piston pump*.
15. [Multics Documentation — Multics v4.1.0](#)
16. <https://engineering.purdue.edu/Maha/assets/docs/MulticsBrochure2022.pdf>.
17. T. Ransegnola, L. Shang, and A. Vacca, "A study of piston and slipper spin in swashplate type axial piston machines," *Tribology International*, vol. 151, p. 106403, Nov. 2020.