# A Study on a Twin-Screw Pump for Thermal Management Systems by means of CFD using SimericsMP+®: experimental validation and focus on Pressure Pulsation

**Pasquale Borriello[1,3], Emma Frosina[2], Pierpaolo Lucchesi[3], Adolfo Senatore[1]**

[1]*Department of Industrial Engineering University of Naples Federico II - Naples, Italy, pasquale.borriello@unina.it, senatore@unina.it*

[2]*Department of Engineering University of Sannio - Benevento, Italy, frosina@unisannio.it*
[3]*Fluid-o-Tech s.r.l.- Milan, Italy, p.lucchesi@fluidotech.it*

## Abstract.

This research is about an innovative methodology for simulating spindle pumps using a full 3D-CFD transient approach by means of Simerics MP+®. Spindle pumps are known from the literature to be very reliable, efficient, and low-noise emission machines. Latest EU Regulations have been ambitious by pushing the transportation sector increasingly towards the reduction of primary harmful pollutants and $CO_2$ emissions. In this context, the thermal management of BEVs (Battery Electric Vehicle) is gaining a new technological interest. In this scenario, improvements on pump efficiency are needed to determinately reduce the absorbed energy during real on-the-road operation. Commonly, centrifugal pumps are used for this kind of application, but their efficiency is highly vulnerable to rotational speed, wasting energy in real exercise, even if they are designed to have a very high efficiency at the design point. Thus, adopting an accurately designed spindle pump for thermal management systems could fit the new needs coming, in particular, from the automotive industry. Computational Fluid Dynamic (CFD) allows the real-time monitoring of pressure distribution, velocity field, mass flow rate, rotor torque, and other performance indicators resulting in a very powerful tool to lead design and optimization phases of such kind of machines. In this paper, a new Simerics MP+® template which allows for the simulation of spindle pumps, never presented in literature, has been adopted and validated by comparing the model results with tests coming from an experimental campaign. In the last section, data on pressure-pulsation of a spindle-pump using CFD are shown for the first time: considerations are carried out both in time-domain and frequency-domain.

**Keywords**. Screw pump, spindle pump, thermal management, automotive, CFD, SimericsMP+, experimental validation, pressure pulsation.

# 1. INTRODUCTION

During the recent years, primary pollutants and $CO_2$ emissions caused by on the road transportation sector have been reduced worldwide through increasingly strict regulations. The European Union (EU) with the European Green Deal aims to grab climate neutrality by 2050. Regulation (EU) 2021/1119 of the European Parliament and of the Council of 30 June 2021 place new targets: 55% net greenhouse gas emissions reduction compared to 1990 by 2030, better known as “Fit for 55” package. To face this issue, vehicle manufacturers are expanding the production of hybrid and electric vehicles [1], which may help in cutting down primary pollutants in urban areas [2]. Also, the recent assimilation of different thermal needs in the engine cooling system (charge air cooling [3, 4, 5], air conditioning condenser [6], separated cooling between engine head and block [7], cabin heater, etc.) outcomes an increased complexity, which claims for increased degrees of freedom in designing new cooling strategies [8, 9, 10]. Generally, centrifugal pumps are qualified in engine cooling systems. In traditional Internal Combustion Engine (ICE) cooling system, centrifugal pumps, mechanically coupled with the crankshaft, were used. This technology does not let on to vary the pump speed separately from the engine speed, preventing a high pump efficiency, which highly relies on the impeller speed. Indeed, centrifugal pumps grant the best performances at the design point, known as at Best Efficiency Point (BEP), commonly located at high flow rate and rotational speed, corresponding to the maximum engine mechanical power. However, the pump works far from the BEP in the course of homologation cycles, resulting in a decreased pump efficiency (between 15-20%), and so in a compelling power absorbed compared to the propulsion one [11]. An electrically actuated centrifugal pump would deliver a faster engine warm-up, accountable for around 2/3 of the whole harmful emissions in typical driving cycles [12], but not a pump efficiency increase.

Rather, spindle pumps are rotary, positive displacement pumps that can have one or more spindles to deliver high or low viscosity fluids along an axis. Like all positive displacement pumps, efficiency is essentially independent on rotational speed. Additionally, spindle pumps are known for simplicity, small volume, high ability of self-priming, smooth working, easy disassembly, low vibration, pressure fluctuation and noise emissions [13]. Such features make them a good fit for the new demands coming from automotive sector, as explained already in [14]. Yet, the spindle kinematics and internal flow field of these machines are very complicated, so the number of accessible flow studies on spindle pumps is quite narrow, and the understanding of the internal flow is yet lacking. Consequently, to meet the modern needs coming from the market, the full recognition of the process within the pump is vital. Spindle machines have been extensively investigated in terms of thermodynamic modelling [15, 16, 17], even for multiphase flow [18, 19, 20]. Spindle pump performances can be predicted as well by lumped-parameters models. A triple spindle pump for an aerospace application has been considered in [21], where a parametric analytical model was developed and validated, arranging the boundary-layer theory [22] to get the volumetric capabilities of the pump. The indirect correlation between backflow and wear resistance in triple spindle pumps was studied developing an accurate theoretical model in [23]. That research was later expanded computing the pressure loads acting on the pump rotors in [24], and finally simulated and experimentally validated even considering dynamic effects in [25]. Lumped parameter models have been developed also for multiphase spindle pumps: in [26] is presented an exhaustive thermodynamic model experimentally validated, model further upgraded considering gas-liquid mixtures with very high gas volume fractions (up to 99 %) [27]. The literature resources credited above give a good understanding of the working process but points out that enhancements are achievable specially for novel applications of multiphase pumps. Recapitulating, most of the current works are based on the thermodynamic chamber, mathematical models which neglect kinetic energy and simplify the analysis on the main and leakage flows. Few published works rely on steady state CFD which considers static mesh of the moving flow domains [13, 28, 29, 30]: by adopting a static grid, however, the transient essence of the working process and the actual velocity field of the main flow are ignored. Breakthrough in employing CFD for the study of positive displacement spindle machines was made by Kovacevic et al. [31] who generated a structured moving mesh for spindle compressor rotor based on a rack generation approach arranged by Stosic et al. [32]. This revolutionary work in grid generation for spindle machines allowed for the CFD simulations and performance prediction of spindle pumps and compressors [33, 34, 35], even catching cavitation phenomena [36, 37].

This paper shows an innovative methodology for simulating spindle pump all-in-one commercial software, never showed before. In fact, differently from what described by the authors in [37], in which the commercial software SCORG® was adopted to mesh the rotors, both the structured moving mesh of rotor fluid domain and the mesh for stationary domains such as ports and pipes are generated using the grid generator built into the commercial software SimericsMP+®. The resolution of the structured moving mesh in the CFD solver is supported by use of the user-defined function and interface, as described in [38] [39]. In particular, the General Gear Template implemented in SimericsMP+® v.5.2.13 has been used to manage the mesh and the movement of the deforming in time field of the rotors. A standard k-ε turbulence model to include turbulent phenomena and a full cavitation model available in SimericsMP+® have been empowered as well to accurately predict the pressure distribution, velocity field, real-time mass flow rate, rotor torque, and other important performance indicators. In the next sections, the 3D-CFD model will be introduced and validated with experimental results. In the end, data on pressure-pulsation of a spindle pump by means of CFD are presented for the first time: results are carried out both in time-domain and frequency-domain.

This research is part of a particular program, established by the Italian government, named PON Research & Innovation where Italian Universities develop Ph.D. programs in collaboration with Italian firms and foreign universities. The parties involved in this program are the University of Naples Federico II, University of Sannio, Fluid-o-Tech, located in Corsico (MI) - Italy, and the Maha Fluid Power Research Center, Purdue University (USA).

## 2. TWIN-SCREW PUMP MODELLING

### *2.1. Reference Machine*

Spindle pumps transfer the fluid medium in perpetual volume cavities, labelled chambers, from suction to delivery side. Of course, like all positive displacement machines, spindle pumps do not create pressure but rather simply move the medium from one spot to another; the output pressure of the pump is dependent on the back pressure downstream. Due to the helical shape of the spindles, chambers are regularly identified by closely intermeshed counter-rotating spindles and moved linearly towards the discharge port. The chambers are not perfectly sealed but connected to each other via different small gaps, termed circumferential, radial and flank gaps where a leakage flow results [40], shown in Fig.2.1. The clearance between the tip of one spindle and the shaft of the other spindle coincide to the radial gap, while the clearances between the tip of the spindles and pump housing consist in the circumferential gaps. Differently to radial and circumferential gaps, flank gap exhibits a complex 3D geometry which brings a considerable difficulty regarding the set-up of a CFD model. In this application case, the driving screw consists of a double-start thread trapped 900° around its corresponding shaft. The driven spindle is a three-start thread resulting in a 2/3 transmission ratio. It is qualitatively shown in Fig. 2.1; no more geometrical parameters are reported for confidentiality reason. The screws are just entrained inside a cylindrical support bordered by the pump housing, no bearings are adopted to facilitate the pump manufacture, as ordinarily demanded in automotive applications, but permits axes misalignments, and thus casual dry friction phenomena between rotors and casing may take place during pump operation. But ordinarily, the film of fluid entrained in the circumferential gap hydrodynamically supports the rotating spindles. The flow conditions of this thin film of fluid are determined by pump speed and pressure rise. So, for very low pump speed operation or very high pressure rise the pump may face risk of damages.

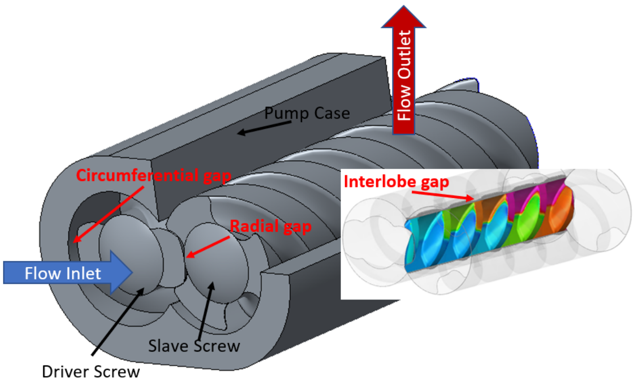


Figure 2.1. Main leakage paths and reference twin-screw pump.

### *2.2. Experimental Set-Up*

The hydraulic schematic of the test rig is presented in Fig.2.2; the medium is a 50:50 mixture of water and glycol (IP Antifreeze Red) at room temperature, with a density of 1070 kg/m3 and viscosity of 0.004 Pa s. A variety of working points are achieved by using a control valve on the discharge side. Rotational speed can be regularly controlled and modified by a dual-range torque meter (Kistler Type 4503A). The pressure at discharge pipe is recorded by an amplified pressure transducer (Unik 5000 pmp). The uncertainty of pressure measurements is estimated to be lower than 0.4 %. The flow rate is measured in the discharge pipe by the means of a Coriolis flowmeters (Sitrans F C Mass 2100 Di 3-40) with an uncertainty below 0.15 % in the involved mass flow range.

The experimental test champaign has been carried out at the Laboratory of Fluid-o-Tech, an Italian company with over 70 years of experience in the engineering and manufacturing of positive displacement pumps and fluidics system.

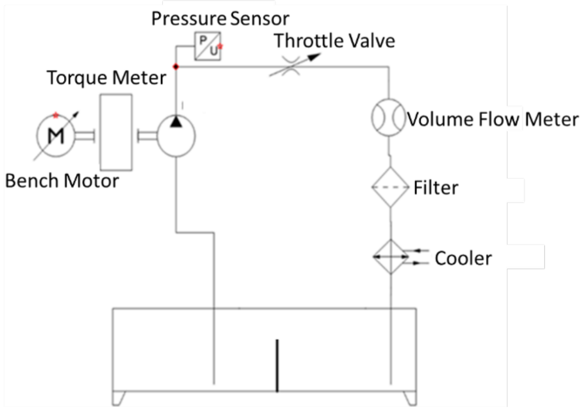


Figure 2.2. Hydraulic schematic of experimental results.

## 3. SIMULATION METHODOLOGY

Positive displacement spindle pumps act on the basis of changing the size and position of a working domain which therefore causes variation in the pressure of the working domain thereby moving the fluid. To determine performances of a spindle pump, quantities such as mass, momentum, energy, etc. require to be numerically modelled. A closely coupled, time dependent set of partial differential equations (PDEs), based on a finite volume method (FVM) represent the governing equations to be solved. The solution of the governing equations is achieved using the commercial software Simerics MP+®. Technicalities on the solution scheme can be found in [41] and are shortly condensed here. The numerical scheme is based on the Reynolds-averaged Navier-Stokes (RANS) equations, and a standard k - ε model is used to consider turbulence effects. In Simerics MP+®, as in the original Singhal et al model [42], the fluid in cavitating flows is always considered to be a mixture of liquid, vapor, and some non-condensable gases. By default, the cavitation model accounts for both liquid-vapor phase change and the effect of non-condensable gases. Based on the modelling approach for non-condensable gas (NGC) effect, Equilibrium Dissolved Gas Model (EDGM), in which the mass fraction of the non-condensable gas dissolved in the liquid is equivalent to the equilibrium value, was employed. Simulations have been conducted with a reference pressure of 1.01 bar as pump inlet pressure, while the pump outlet pressure has been set at different discharge pressure up to 4.01 bar. By modifying the pressure difference between the inlet and the outlet, the required flow rate is obtained. Rotating speed of the driver rotor ranges from 500 rev/min up to 4000 rev/min. Stagnation inlet and pressure outlet are used correspondingly for the inlet and outlet boundaries. The starting pressure and initial velocity are 1.01 bar and 0 m/s, respectively. The turbulence intensity is 1% and the turbulence viscosity ratio is 10.

### *3.1. Grid Generation*

Generating the mesh consists in discretizing a working domain in control volumes for which a solution of local fluid properties is to be established. In this work, the Grid Generator built in SimericsMP+® permits the discretization of the computational domain. The volume of fluid (VOF) domain, shown in Fig.3.1, has been extracted from the CAD 3D drawing using the commercial software Creo®, and spilt in sub-volumes, which will be interfaced afterward in Simerics MP+®. Finally, the VOF has been imported in the CFD code in STL format and then meshed with appropriate grid sizes.

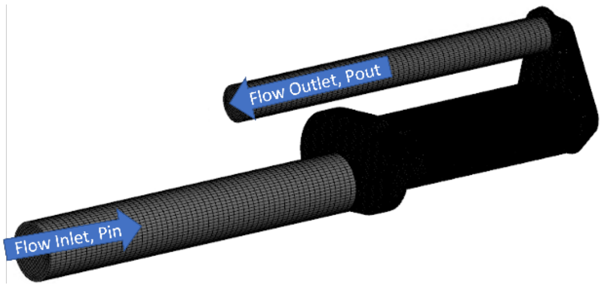


Figure 3.1. VOF domain.

An unstructured body-fitted binary tree approach is employed for the static mesh of the inlet port and outlet port using SimericsMP+® general grid generator. For the spindles, whose working domain has to change and deform time accordingly following the aspect of the operating principle, the new General Gear rotor template mesher for spindle machines have been adopted. A mesh sensitivity analysis has been managed to obtain the outlet volumetric flowrate independence of cell size, as shown in Fig. 3.2. The VOF counts around 2.87 M cells, in particular the rotor mesher for the structured spindle volumes has been set as follows: 120 cells in the circumferential direction, 8 in the radial direction and 600 in the axial direction. Models with finer mesh have been explored, but the results did not sustain the increment in the computational time.

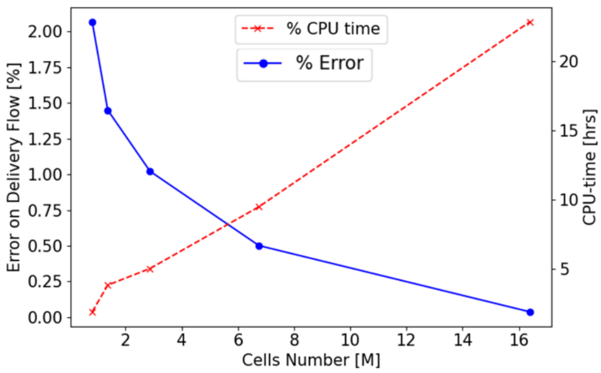


Figure 3.2. Mesh sensitivity analysis.

# 4. RESULTS

After completing the mesh sensitivity analysis, the CFD model can be validated on experimental data. Simulation have been run on a desktop workstation equipped with a 32 cores Intel® Xeon® Gold 6142 CPU 2.60GHz processor with 128 Gb memory, where each run took about five hours for a pump revolution. Each spindle pump rotation has been simulated by means of 180 time-steps, resulting in 2 degrees main rotor rotation per step, as previously done by the authors [37]. An opportune convergence criterion of four order of magnitude has been chosen to satisfy mass flow conservation between inlet and outlet and achieved in 50 inner iterations at each time step. As regards the flow rate, time averaging is performed for one spindle revolution, after it has been guaranteed that initial transients have been abated.

## *4.1. Model Validation*

In Fig. 4.1 experimental and numerical results are compared on a flow-pressure diagram; comparison among four rotational speeds are showed (500, 1500, 3000, 4000 rev/min). These results show the benefit obtained considering the micro-motion of the spindles due to the unbalanced pressure field in the radial cross because of the helical shape of the spindles, as shown in Fig.4.2. Basically, to recreate the realistic situation of the spindles being pushed by the hydrostatic force [42], both spindles have been shifted from the center of the pump housing, resulting in the spindles eccentric with respect to the pump housing, with the smaller gap set at 5 µm. In fact, in [37], the maximum error in the averaged flow rate was 10%, with a standard deviation of the same order of magnitude. In this study the mean error in the averaged flow rate is 0.3% with a maximum error of 1.2% and a standard deviation of 0.7%. Therefore, a first critical achievement obtained by the fluid-dynamic model developed in the present paper is the considerable reduction of the error with respect to the previous modeling approach.

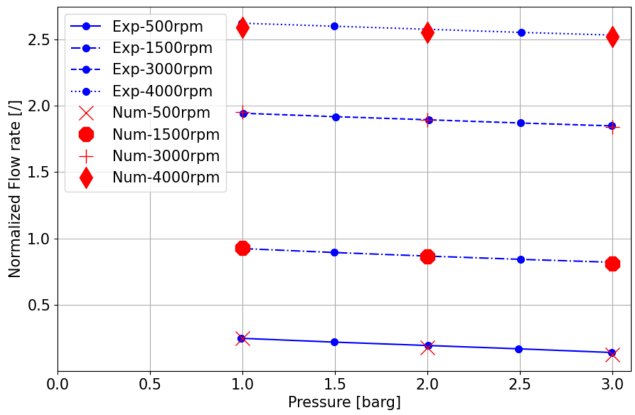


Figure 4.1. Numerical model validation – Normalized Flow rate vs Pressure.

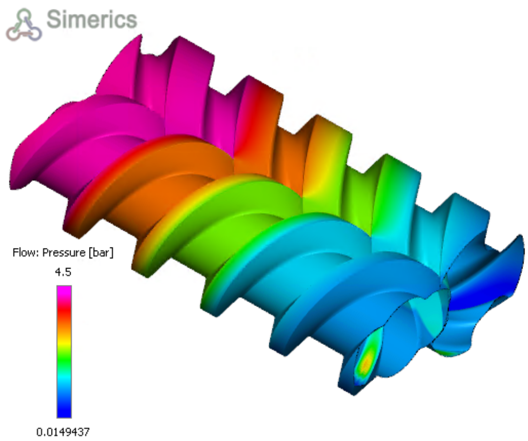


Figure 4.2. Pressure distribution: unbalanced in radial cross section.

Fig.4.3 shows the comparison between the predicted and measured torque at two different pressures rises and over four different pump's speed: excluding 500 rev/min, the maximum error is 11.2% while average error is 4% with a standard deviation of 6.9%.

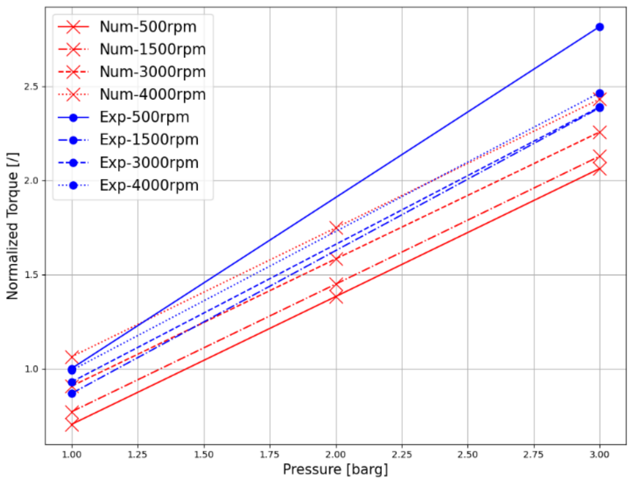


Figure 4.3. Numerical model validation – Torque vs Pressure.

This is an interesting result, meaning that torque due to friction and contact between the spindles is a very small part respect to torque due to pressure and viscous effect that are instead kept into account by the CFD model. From Fig.4.3 is also visible that, while at 1500 and 3000 rev/min torque predicted from CFD tends to underestimate the real torque, as expected, at 4000 rev/min the trend is the opposite. This can be explained looking at Fig.4.4:

in a) six probing points along the axis direction of the spindles have been put; in b) the pressure of each of those points has been plotted. It is evident that the higher the speed the higher the pump's suction capability, resulting in a lower pressure at spindles' inlet side, eventually meaning that the delta pressure the spindles have to overcame at 4000 rev/min results bigger than at 500 rev/min.

At 500 rev/min, instead, higher torque absorptions can be explained due to a not fully developed lubrication regime, as anticipated in Section 2.1.

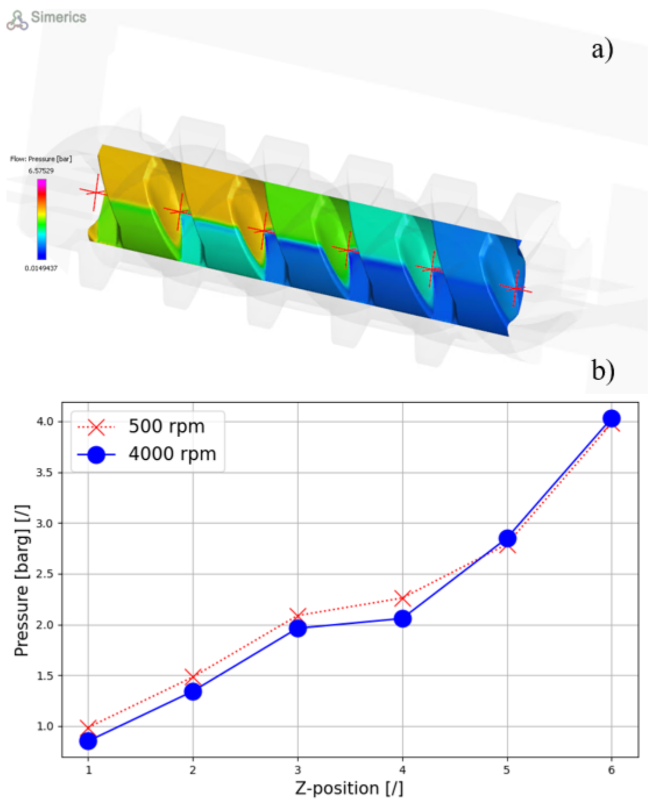


Figure 4.4. Torque analysis: a) six probing points have been put along the axis of the spindle in the CFD model; b) local pressure has been plotted for each point. At 400 rev/min pressure at spindle's inlet side is lower due to higher suction capability resulting in a bigger pressure drop.

### *4.2. Pressure Pulsation*

In this section, for the first time in literature, pressure pulsation of spindle pump by means of CFD are presented. The fluid domain described above has been modified according to [44-45].

As showed in Fig. 4.5, avoiding any discontinuity in the diameter, the system implements a perfect constant diameter duct coupled with a calibrate orifice: this design creates a system load, that can be easily replicated in the numerical model. In future, numerical ripples will be compared with experimental data to be done by modifying the test rig and placing a pressure transducer in the same position of the red probing point visible in Fig. 4.5.

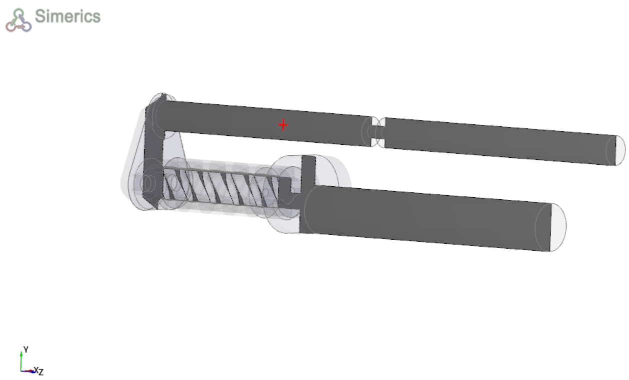


Figure 4.5. VOF on a X-plane – Detail on the delivery line

In Fig.4.6 is reported a numerical comparison of the pressure ripple, both in time domain and frequency domain, for three different imposed circumferential gaps: a) baseline pump, b) double than baseline (later called 2* Baseline), c) four times bigger than baseline (later called 4* Baseline). All the three models have been run at 2641 rev/min. Looking at frequency-domain, main contributes are imputable to geometrical parameters of the spindles. In fact, considering that:

$$f_1 = {}^{2641}/_{60} = 44.02\ Hz \tag{4.1}$$

and recalling that the driver spindle has 2 starts while the driven spindle has 3 starts, frequencies such as:

$$2 * f_1 = 88.03\ Hz \tag{4.1}$$

$$^{2}/_{3} * f_1 = 29.34\ Hz \tag{4.1}$$

$$^{3}/_{2} * f_1 = 66.03\ Hz \tag{4.1}$$

and their multiples such as:

$$3 * f_1 = 132.05\ Hz \tag{4.1}$$

$$4 * f_1 = 176.07\ Hz \tag{4.1}$$

$$6 * f_1 = 264.1\ Hz \tag{4.1}$$

$$8 * f_1 = 352.13\ Hz \tag{4.1}$$

can be individuated.

Analyzing in time domain results from Fig. 4.6, it is clear that Baseline and 2*Baseline shapes are very much comparable. Even in frequency domain, the contributions are consistently at the same values, with a much higher component at $2f_1$. Going at much higher circumferential gap, like the 4*Baseline model, contributions on frequency domain are spread across more frequencies, as guessable looking at the shape in time domain. Since the three different models are exposed to a different downstream pressure because they have different volumetric efficiency, the Non-Uniformity Grade (NUG) [46] is reported in Table I for comparison. Thus, the analysis confirms that, as known from literature, fluid born noise in spindle pumps has a very little contribute on the overall noise emission since pressure pulsation are very low.

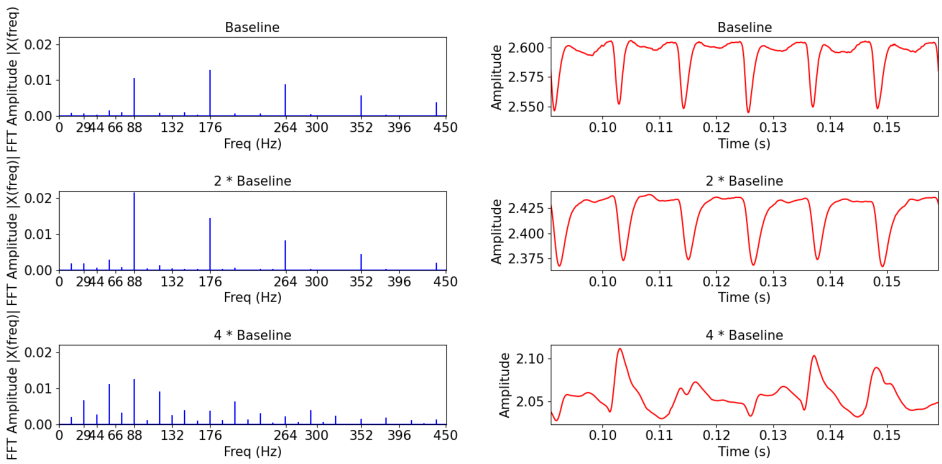


Figure 4.6. Pressure pulsation in time-domain and frequency-domain for three different circumferential gap: a) Baseline; b) 2*Baseline; c) 4*Baseline.

TABLE I. NUG AMONG THREE DIFFERENT CFD MODELS

| **MODELS** | **NUG** |
|---|---|
| *a) Baseline* | 0.024 |
| *b) 2* Baseline* | 0.030 |
| *c) 4* Baseline* | 0.041 |

### *4.3. Cavitation*

3D-CFD simulation allows the user to monitor important performance indicators such as pressure distribution, mean velocity distribution, velocity distribution in gap areas, parameters related to turbulence model and many others. In this section considerations related to cavitation phenomena are carried out.

Cavitation may occur at the spindle leading edge, the radial gaps and the flank gaps, as already described in [37]. Over flank gaps, as visible in Fig.4.4.a), high pressure differences lead local velocities to reach high values consequently lowering the local static pressure. Along circumferential gaps, local velocities reach high values because of high rotational speed. That higher circumferential velocity causes higher pressure losses in the lower part of the chamber resulting in a consequent reduction of total pressure, also increasing the dynamic pressure. The reduction of total pressure and the rise of the dynamic pressure on the other hand leads to lower static pressure at the bottom side of any chamber. As showed in Fig.4.7, Gas Volume Fraction in proximity of the 5 μm clearances along circumferential gaps are more important to what showed in [37] since rotors are moved in y-direction accounting for the unbalanced pressure in the radial cross, as discussed in Section 4.1.

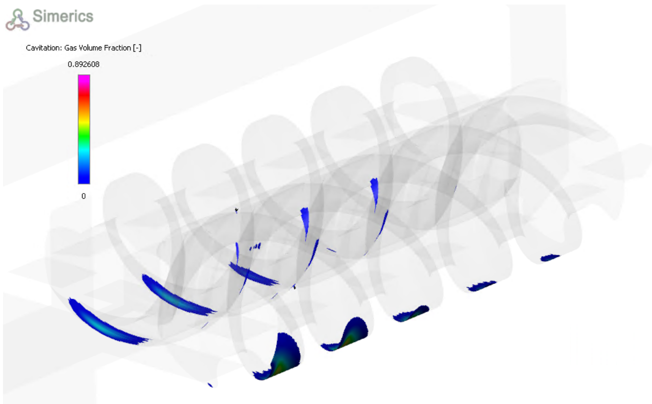


Figure 4.7. Isosurface of the Gas Volume Fraction at 3000 rev/min and ΔP=3 bar.

## 5. CONCLUSIONS

This paper presented a methodology for simulating spindle pumps by means of 3D-CFD employing the commercial software SimericsMP+®. Turbulence and cavitation have been included in the numerical model to capture any aspect during the real operation. An experimental test campaign was specifically performed with the aim of validating the developed tool. Comparisons between experimental results and numerical predictions were presented as concerns the unit steady-state operating features.

Numerical results were also presented concerning the fluctuations of the outlet pressure for three different circumferential gaps, that is a parameter of primary importance when studying the fluid borne noise for the reduction of the noise emissions in hydraulic machines. The presented comparisons highlight the potentiality of the tool for the prediction of pressure ripple as effects of the operating conditions and to identify the main design. Finally, it is possible to conclude that the presented tool has powerful capability in predicting with a good accuracy the real-time mass flow rate and rotor torque; by also looking at pressure distribution, velocity field, and other important performance indicators, the presented numerical approach can lead design and optimization phase of such kind of machines.

## 6. ACKNOWLEDGMENTS

The authors would like to thank the support from Giuseppe Ricucci of Fluid-o-Tech, from Federico Monterosso and Micaela Olivetti of OMIQ s.r.l.

## 7. REFERENCES

[1] IEA, Global EV Outlook 2021.

[2] S.I. Ehrenberger, M. Konrad, F. Philipps, "Pollutant emissions analysis of three plug-in hybrid electric vehicles using different modes of operation and driving conditions", Atmos. Environ. 234, 2020.

[3] R. Cipollone, D. Di Battista, A. Gualtieri, "A novel engine cooling system with two circuits operating at different temperatures", Energy Conversion and Management, 75, pp. 581-592, 2013.

[4] H. Chen et al., FCA US LLC, "Chiller system for an engine with a forced induction system", United States Patent, US10006339B2, 2018.

[5] D. E. Hornback et al., FCA US LLC, "Vehicle thermal system for reduced fuel consumption", United States Patent, US 10216258B1, 2019.

[6] D. Di Battista, R. Cipollone, High efficiency air conditioning model based analysis for the automotive sector, International Journal of Refrigeration, 64, pp. 108-122, 2016.

[7] R. Cipollone, D. Di Battista, A. Gualtieri, "Head and block split cooling in ICE", IFAC Proceedings Volumes (IFAC-PapersOnline), 45 (30), pp. 400-407, 2012.

[8] T. Castiglione, P. Morrone, L. Falbo, D. Perrone, S. Bova, Application of a model-based controller for improving internal combustion engines fuel economy, Energies, 13 (5), art. no. 1148, 2020.

[9] A. K. Haghighat, S. Roumi, N. Madani , D. Bahmanpour, M. G. Olsen, "An intelligent cooling system and control model for improved engine thermal management", Applied Thermal Engineering 128, 253–263, 2018.

[10] H. Mu, Y. Wang, H. Teng, Y. Jin, X. Zhao, X. Zhang, "Cooling system based on double-ball motor control valve", Advances in Mechanical Engineering, Vol. 13(5) 1–12 , 2021.

[11] L. Mariani, M. Di Bartolomeo, D. Di Battista, R. Cipollone, F. Fremondi, R. Roveglia, "Experimental and numerical analyses to improve the design of engine coolant pumps", E3S Web of Conf. 197, 06017, 2020.

[12] J. Gao, G. Tian, A. Sorniotti, A. E. Karci, R. Di Palo, "Review of thermal management of catalytic converters to decrease engine emissions during cold start and warm up", Applied Thermal Engineering 177–187, 2019.

[13] Li, W., Lu, H., Zhang, Y., Zhu, C., Lu, X., & Shuai, Z., "Vibration analysis of three-screw pumps under pressure loads and rotor contact forces", Journal of Sound and Vibration, 360, 74-96, 2016.

[14] G. Di Giovine, L. Mariani, M. Di Bartolomeo, D. Di Battista, R. Cipollone, "Triple-Screw pumps for ICE cooling systems in the transportaction sector", E3S Web of Conferences 312, 07024, 2021.

[15] Li F. Screw pump [M]. Beijing, China: Machine Press, 2010.

[16] Vetter, G., and Wincek, M, "Performance Prediction of Twin-Screw Pumps for Two- Phase Gas/Liquid Flow," Pumping Machinery, Fluid Engineering Conference (FED), ASME, Washington, DC, 154, pp. 331-340, 1993.

[17] Patil A., “Performance evaluation and CFD simulation of multiphase twin-screw pumps”, PhD Thesis, Texas A&M University, College Station, TX, 2013.
[18] Mewes D, Aleksieva G, Scharf A, et all., “Modelling twin screw multiphase pumps – a realistic approach to determine the entire performance behaviour”. [2nd International EMBT Conference, Hannover, April 2008, pp.104–116, 2008].
[19] Rabiger K. “Fluid dynamic and thermodynamic behavior of multiphase screw pumps handling gas-liquid mixtures with very high gas volume fractions”, PhD Thesis, Faculty of Advanced Technology, University of Glamorgan, 2009.
[20] Hu B., Cao F., Yang X. “Theoretical and experimental study on conveying behavior of a twin-screw multiphase pump”, [Proceedings of the Institution of Mechanical Engineers, 230(4), pp. 304-315, 2016].
[21] A. Putira, “Parametrization of Triple Screw Pumps for Aerospace Applications”, https://commons.erau.edu/edt, 2018.
[22] H. Schlichting, K. Gerten, “Boundary-Layer Theory”, Ninth Edit, Springer, 2017.
[23] G.C. Mimmi, P.E. Pennacchi, “Design of three-screw positive displacement rotary pumps”, Trans. Eng. Sci. 7, 1995.
[24] G.C. Mimmi, P.E. Pennacchi, “Computation of pressure loads in three screw pump rotors”, J. Mech. Des. Trans. ASME, 581–588, 1998.
[25] G.C. Mimmi, P.E. Pennacchi, “Dynamic effects of pressure loads in three screw pump rotors”, J. Mech. Des. Trans. ASME, 12, 589–592, 1998.
[26] C. Feng, P. Yueyuan, X. Ziwen, S. Pengcheng, Thermodynamic performance simulation of a twin-screw multiphase pump, Proc. Inst. Mech. Eng. Part E J. Process Mech. Eng. 215, 157–163, 2001.
[27] K. Räbiger, “Fluid dynamic and thermodynamic modelling of multiphase screw pumps, operating on the threshold of an exclusive gas compression”, [7th International Conference on Compressors and their Systems 2011, Pages 385-400, 2011].
[28] R. Ma and K. Wang, “CFD Numerical Simulation and Experimental Study of Effects of Screw-Sleeve Fitting Clearance upon Triangular Thread Labyrinth Screw Pump (LSP) Performance”, Journal of Applied Fluid Mechanics, Vol. 3, No. 1, pp. 75-81, 2010.
[29] Tang Q and Zhang Y, “Screw optimization for performance enhancement of a twin-screw pump”, Proc IMechE. Part E: J Process Mechanical Engineering, 228, pp.73–84, 2014.
[30] Qian Tang, Yuanxun Zhang, Linqing Pei, Jiong Tang, “Optimal design of screw and flow field analysis for twin-screw pump”, [Proceedings of the ASME 2011 International Design Engineering Technical Conferences & Computers and Information in Engineering Conference IDETC/CIE 2011 August 28-31, Washington, DC, USA , 2011]
[31] Kovacevic´ A, Stosic´ N and Smith IK. Screw compressors: three-dimensional computational fluid dynamics and solid fluid interaction. New York, NY: Springer.
[32] Stosic N, Smith IK and Kovacevic A. Screw compressors: mathematical modelling and performance calculation. New York, NY: Springer, 2005.
[33] E. Mujic, A. Kovacevic, N. Stosic, I. Smith, “Advanced Design Environment for Screw Machines”, Int. Compress. Eng. Conf., 2010.
[34] Y. Lu, A. Kovacevic, M. Read, N. Basha, “Numerical Study of Customised Mesh for Twin Screw Vacuum Pumps”, Designs, 3(4), 52, 2019.

[35] Yan D., Kovacevic A., Tang Q, "Numerical modelling of twin-screw pumps based on computational fluid dynamics", Proceedings of the Institution of Mechanical Engineers, 231(24), pp. 4617-4634, 2017.
[36] D. Yan, A. Kovacevic, Q. Tang, S. Rane, Numerical investigation of cavitation in twin-screw pumps, Institution of Mechanical Engineers Part C, 2017.
[37] P. Borriello, E. Frosina, A. Senatore, F. Monterosso, "Numerical Modelling and Experimental Validation of Twin-Screw Pumps based on Comuputational Flud Dynamics using Scorg® and SimericsMP+®", E3S Web of Conferences 312, 05007, 2021.
[38] Frosina, E.; Senatore, A.; Rigosi, M, "Study of a High-Pressure External Gear Pump with a Computational Fluid Dynamic Modeling Approach", Energies 2017, 10, 1113, 2017.
[39] A. Corvaglia, M. Rundo, P. Casoli, A. Lettini, "Evaluation of Tooth Space Pressure and Incomplete Filling in External Gear Pumps by Means of Three-Dimensional CFD Simulations". Energies 2021, 14, 342, 2021.
[40] Moghaddam, A. H., Kutschelis, B., Holz, F., Börjesson, T., and Skoda, R, "Three-Dimensional Flow Simulation of a Twin-Screw Pump for the Analysis of Gap Flow Characteristics." ASME. J. Fluids Eng, March 2022.
[41] Inc. S. Simerics MP+'s User Manual – v 5.2.13.
[42] A. Singhal, M. Athavale, H. Li and Y. Jiang, "Mathematical basis and validation of the full cavitation model", J. Fluids Eng., 124, pp. 617-62, 2002.
[43] Vacca A., Guidetti M., "Modelling and Experimental Validation of External Spur Gear Machines for Fluid Power Applications", Simulation Modelling Practice and Theory, Vol. 19, No.9, pp.2007-2031, 2011.
[44] E. Frosina, G. Marinaro, A. Senatore, Experimental and Numerical Analysis of An Axial Piston Pump: A Comparison Between Lumped Parameter And 3D CFD Approaches, Proceedings of the ASME-JSME-KSME 2019, 8th Joint Fluids Engineering Conference AJKFluids2019, July 28-August 1, San Francisco, CA, USA, 2019.
[45] E. Frosina, G. Marinaro, A. Senatore, M. Pavanetto, "Effects of PCFV and Pre-Compression Groove on the Flow Ripple Reduction in Axial Piston Pumps", 2018 Global Fluid Power Society PhD Symposium, GFPS 2018.
[46] J. Ivantysyn and M. Ivantysynova, "Hydrostatic Pumps and Motors: Principles, Design, Performance, Modelling, Analysis, Control and Testing", Tech Books International, 2003.

## Biographies

**Pasquale Borriello** received the bachelor's degree in mechanical engineering from University of Naples Federico II in 2017, the master's degree in mechanical engineering

from University of Naples Federico II in 2019 after a 7 months Visiting Scholar experience at Purdue University. He pursued the philosophy of doctorate degree in Industrial Engineering from University of Naples Federico II in 2024 having as partners Purdue University, where he spent six months as Research Visiting in 2022-2023, and University of Sannio.
Along with his academic career, he developed an industry track starting with a 2-month internship at F-lab in 2019 and working from 2020 as Product Innovation Engineer at Fluid-o-Tech in Milan. He currently works at Fluid-o-Tech as Product Innovation Manager leveraging simulation and computer science knowledges to develop new products and technologies.