11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

AN INVESTIGATION OF THE PERFORMANCE OF A POSITIVE DISPLACEMENT RECIPROCATING PUMP AT LOW PRESSURE NPSH INCORPORATING A THREE PHASE CAVITATION MODEL

ALDO IANNNETTI[*], MATTHEW T. STICKLAND[†] AND WILLIAM M. DEMPSTER[†]

[*] Mechanical and Aerospace Engineering Department
University of Strathclyde
16 Richmond Street, Glasgow G1 1XQ, Scotland, United Kingdom
e-mail: aldo.iannetti@strath.ac.uk

[†] Mechanical and Aerospace Engineering Department
University of Strathclyde
16 Richmond Street, Glasgow G1 1XQ, Scotland, United Kingdom
email: matt.stickland@strath.ac.uk

[†] Mechanical and Aerospace Engineering Department
University of Strathclyde
16 Richmond Street, Glasgow G1 1XQ, Scotland, United Kingdom
email: william.dempster@strath.ac.uk

Key Words: *Singhal et al. cavitation model, Multiphase flows, PD reciprocating pump, self-actuated valve model, moving mesh.*

Abstract. The full cavitation [1] multiphase CFD model of a positive displacement reciprocating pump is presented to investigate performance during the pumping cycle through 360° of the crank shaft rotation. This paper discusses the cavitation appearance and dynamics inside the pump chamber at 100kPa, 50kPa, 25kPa and 0kPa inlet gauge pressure and evaluates the Singhal et al. [1] cavitation model in conditions of incipient cavitation, partial cavitation and full cavitation. The paper also investigates the role of pump inlet valve inertia on cavitation dynamics. The transient CFD model takes into account a three phase flow composed of water, water vapour and 15 parts per million (ppm) of non-condensable ideal gas mass fraction, and utilizes the moving mesh technique to deal with the inlet and outlet valve dynamics. A User Defined Function (UDF) is utilized to couple the pressure field and the valve force and displacement-time histories so that the valves are "self-actuated". A second UDF handles the compressibility model of water which is essential for high outlet pressure and to stabilize the simulation in the situation when the valves are both closed. The paper shows the feasibility of such a complete CFD model of a PD pump, equipped with the Singhal et Al. cavitation model, and its capability to assess the rate of phase change, the efficiency loss and the prediction the valve lift history.

1 INTRODUCTION

Technical literature usually classifies the cavitation phenomenon into three different regimes as Opitz [2] pointed out and discussed, they are incipient cavitation, partial cavitation and full cavitation. They are usually recognised by their different features and loss of performance that they cause. In positive displacement (PD) reciprocating pumps the most important external factors which lead to cavitation are usually the inlet pressure and the crank angular velocity. Increasing the speed and/or decreasing the inlet pressure the pump can reach conditions from incipient to full cavitation.

According to Opitz [2], [3] incipient cavitation occurs in a very short time at the beginning of the inlet stroke when the inlet valve is still close and the chamber decompression creates a wide zone of low pressure where bubbles are randomly distributed in the chamber and they do not impact on the surfaces. According to other authors such as Franc [4] incipient cavitation is a more general situation where the generation of bubbles is not sufficiently significant to affect the main flow. All authors agree on the fact that incipient cavitation does not produce damages to the device and it seems it does not affect the pump performance either. In fact, as Pawan recalled [5], the Hydraulic Institute Standards fixed at 3% the maximum allowable loss of volumetric efficiency and, incipient cavitation does not exceed that limit.

Partial cavitation [2] is characterized by complete bubble degeneration within the inlet stroke. It can be classified further into the three sub-categories of incipient partial, advanced partial and distinctive partial cavitation depending on the features and when they arise within the inlet stroke. The full cavitation is characterised by the complete degeneration of the vapour cavities at the beginning of the outlet stroke, a significant volumetric efficiency loss is to be expected.

Many tools have been designed over the years to analyse the behaviour of PD pumps, from analytic to experimental. Shu [6] and Edge [7], for instance worked on an analytic tool to predict the performance of PD pumps while Opitz [2] more recently chose an experimental approach. The work presented in this paper is CFD oriented because the authors believe CFD techniques nowadays are ready to model the physics and functioning of PD pumps in cavitation conditions.

To correctly simulate the features of the cavitation regimes, a CFD model of a PD reciprocating pump should consider the independent movement of the valves which have to be actuated by the pressure field rather than the analyst’s external action as explained in [8].

A self-actuated CFD model of the valves should also be equipped with a reliable cavitation model which calculates the modified pressure field when the pump cavitates. It is very important to have a comprehensive cavitation model as well as a multiphase model. In fact, cavitation is affected by the grade of pureness of water i.e. the non-condensable gas mass fraction dissolved in water as discussed by [9] and this has to be considered in the model. The work which this paper presents aims at creating that comprehensive CFD model of one chamber of a PD pump and demonstrating its feasibility and capability in predicting the features among the different cavitation regimes and the loss of performance.

2 CFD MODEL

A complete pumping cycle composed of the inlet and outlet stroke of the pump represented in figure 1 was simulated. The starting point of the simulation was the initial point of the inlet stroke when the plunger was at its Top Dead Centre (TDC) position (see figure 1) with the crank angle at 0°. From 0° to 180° the plunger performed the entire inlet stroke moving backwards and reaching the Bottom Dead Centre position (BDC), it moved forward compressing the chamber again in the crank angle ranges from 180° to 360°. The inlet valve lifted off during the suction stroke as the decompression of the plunger caused a non-null pressure forces to act on the valve, it returned back on the seat in the final part of the inlet stroke as the pressure difference was restored, pushed by the spring. In a similar manner the outlet valve opened when the forward stroke of the plunger created a pressure force sufficiently high to exceed the outlet valve spring preload and returned on the seat when the pressure was restored in the final part of the outlet stroke. The layering moving mesh algorithm took care of the displacement history of the plunger as well as the lift history of the valves. The plunger displacement was fixed by the reciprocating motion depending on the crank angular velocity (130 rpm), crank diameter and connecting rod length. The inlet and outlet valve's lift was governed by a User Defined Function (UDF) as described by Iannetti [8], figure 2 describes how the UDF worked. The fluid model was decomposed into the pattern of sub-volumes shown in figure 3 in order to correctly handle the moving mesh algorithms in a similar manner to that described by Iannetti [8]. The UDF also took into account the spring stiffness characteristic curve and a further UDF dealing with the compressibility model of water was also provided.

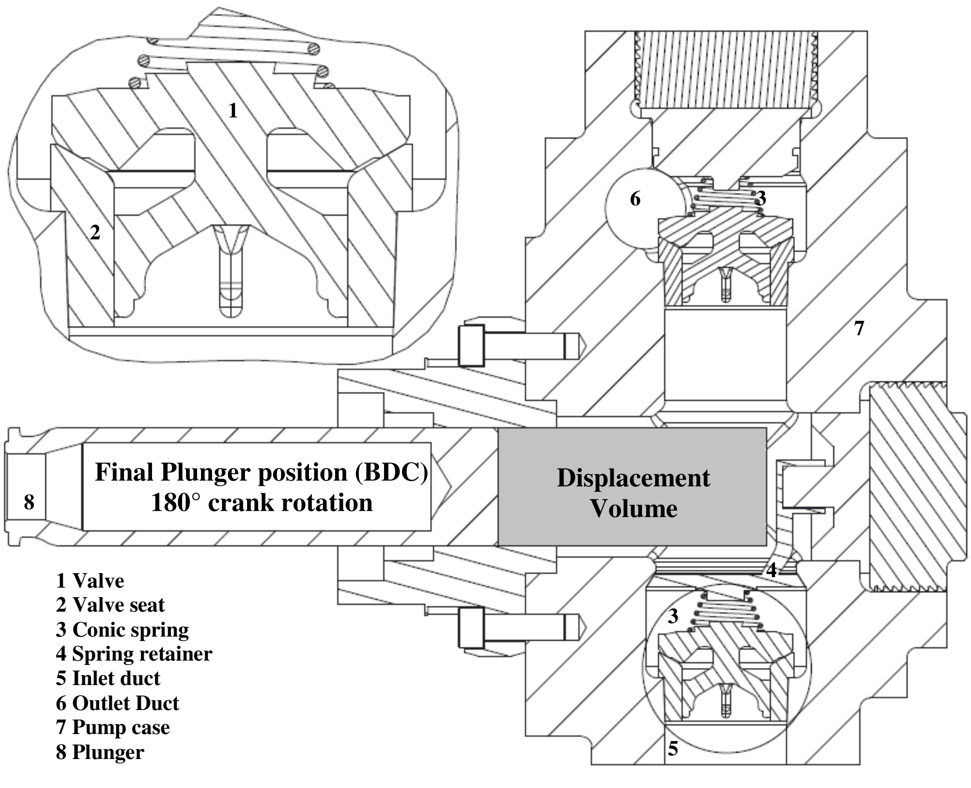

Figure 1: PD pump geometry and nomenclature, TDC and BDC plunger position

The new feature which was not present in the model discussed by Iannetti [8] that is a peculiar feature of the work being presented in this paper is the Singhal et al. cavitation model [1] which was used in conjunction with the mixture multiphase model [10]. This cavitation model is usually called the "full" cavitation model because it takes into account all the first order terms of the Rayleigh-Plesset equation [4] as well as the surface tension term. It also takes into account the effects of the non-condensable gas.

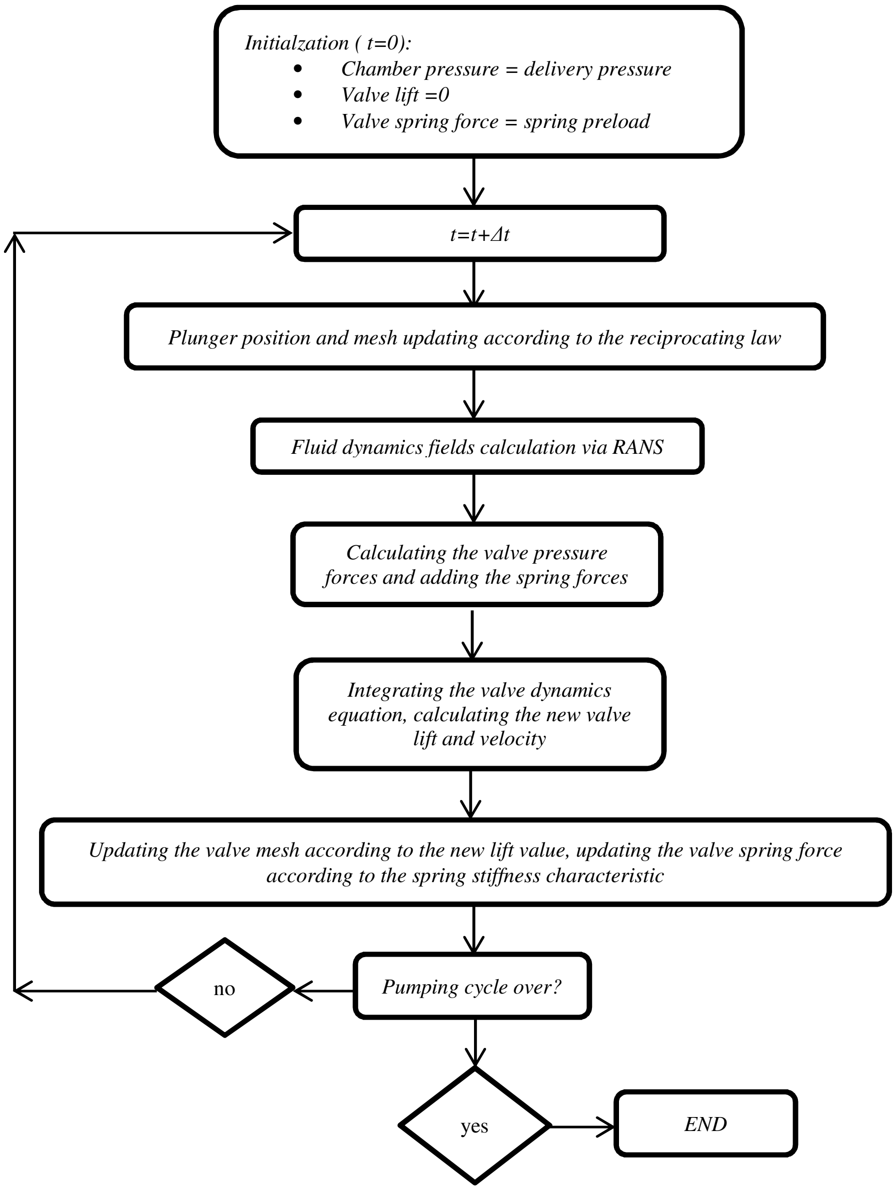

Figure 2: A UDF which interacts with the RANS solver to govern the valve's lift and the moving mesh

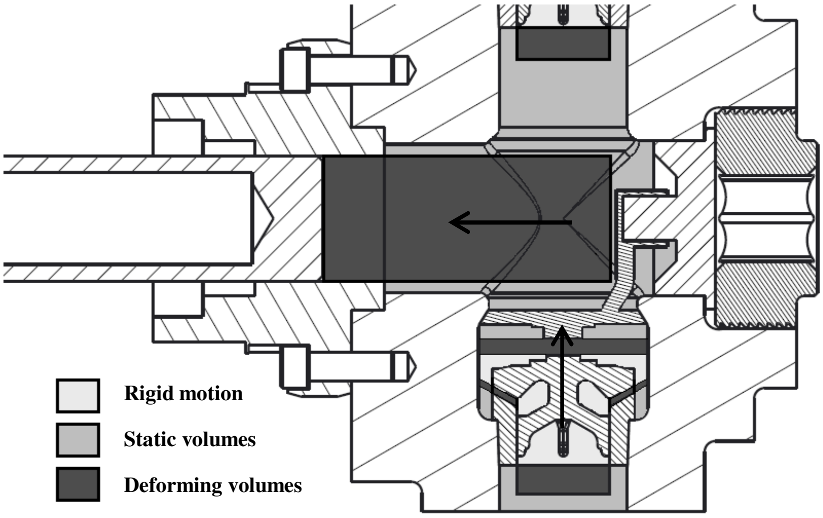

Figure 3: Moving mesh: Decomposition pattern of fluid volumes, the arrows indicate the direction of creation of new mesh layers when the plunger is moving backwards (induction) and the valve is lifting up

2.1 Mesh

The complex geometry of the pump simulated did not allow the use of a full hexahedral mesh, the best solution in this case was the hybrid mesh. As already discussed by Iannetti [8] for the volumes involved in the moving mesh the choice of hexahedral cells was made while tetrahedral cells were utilised for the static volumes.

2.2 Solver settings

Table 1 summarises the solver settings chosen for the simulations as well as the solution controls.

Table 1: Solver settings summary

Solver		RANS, pressure based, transient		
Models	Multiphase	Mixture model [10]		
		Phases	Water liquid	Primary phase
			Water vapour	Secondary phase
	Turbulence	K-ε Standard	Enhanced wall treatment	
	Cavitation	Singhal et al.	15 ppm air (ideal gas)	
Pressure-Velocity coupling		SIMPLE		
Spatial discretization		Momentum	Second order upwind	
		Vapour	First order upwind	
		Turbulent kinetic energy	Second order upwind	
		Turbulent dissipation rate	Second order upwind	
Transient formulation		First order implicit		

Under relaxation factors	Pressure	0.3
	Momentum	0.7
	Vapour	0.5
	Turbulent kinetic energy	0.8
	Turbulent dissipation rate	0.8
Residuals	10^{-3}	
Time step	0.125° crank rotation ≡ 1.6 x 10^{-4} s @130 rpm	
Max Iteration per time step	35	
UDFs	Compressibility of water [8]	
	Valve dynamics, see Figure 2	

2.3 Boundary conditions

To create the cases mentioned in the abstract, four different boundary conditions were chosen. Since the model simulated the whole pumping cycle, inlet and outlet pressure conditions were needed. For simplicity's sake the hypothesis of using the same pressure at inlet and outlet was used within each case as one can see in Table 2, the same plunger displacement-time history was chosen for all cases. Figure 4 shows the plunger displacement over crank rotation angle for the chosen shaft angular speed of 130 rpm.

Table 2: Inlet pressure, outlet pressure and plunger displacement boundary conditions

Case	Pressure: function of the mass flow rate		Plunger
	Inlet pressure [kPaG]	Outlet pressure [kPaG]	Displacement
Case 1	0	0	see Figure 4
Case 2	25	25	see Figure 4
Case 3	50	50	see Figure 4
Case 4	100	100	see Figure 4

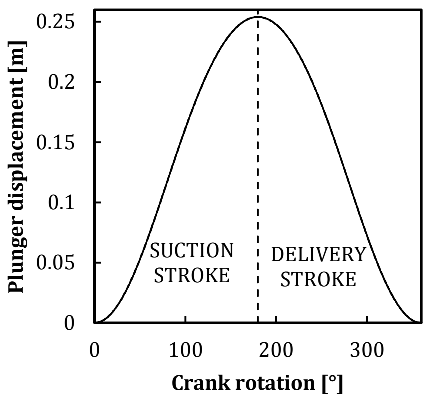

Figure 4: Boundary conditions, plunger displacement hystory

3 RESULTS DISCUSSION

Figure 5 shows the chamber pressure history over crank rotation of all the four case being discussed. To record the shown temporal trend a monitor point inside the chamber was chosen, it was placed in the vicinity of the TDC position of the plunger and was considered static throughout the pumping cycle. Figure 5 shows that case 1 and 2 approached the vapour pressure, case 1 did this for a longer time (100°-170° of crank rotation) while case 2 varied around the vapour pressure for a shorter time (roughly 110°-130° of crank rotation). This resulted in the greater vapour generation as shown in Figure 6 which quantifies the vapour fraction calculated on the plunger top surface for the four cases. The estimated peaks occurring between 100° and 120° were 11% and 6% for case 1 and 2 respectively and this confirmed the pressure trend of Figure 5, the longer the chamber pressure remained in the vicinity of the vapour pressure the higher was the generation of vapour. Table 3 shows the volumetric efficiencies of the cases 1 to 4, it also points out the relationship between the inlet valve closing time and efficiency. The higher was the delay of the inlet valve closing the higher was the volumetric efficiency loss. For case 1, Figure 6 together with the Table 3 data demonstrates that the delay of the inlet valve closing was due to the presence a significant amount of vapour in the last part of the inlet stroke, it implied that the initial part of the outlet stroke was needed to compress the vapour and turn it back to liquid, this caused the loss in volumetric efficiency. According to Opitz [2] one can say that case 1 describes a typical situation of full cavitation. Furthermore, according to Opitz and Tackett [11] one can say that case 2 describes the characteristic features of partial cavitation as the volumetric efficiency loss was within the 3%.

Figures 5 and 6 show that case 4 chamber pressure remained safely distant from the vapour pressure and a negligible amount of vapour was generated. This resulted in the lower inlet valve closing time delay and the consequent smallest volumetric efficiency loss as highlighted by Table 3. One can say that case 4 was not affected by cavitation. Case 3 showed an intermediate behaviour, figure 5 shows that case 3 chamber pressure remains distant from the vapour pressure limit but figure 6 shows a maximum vapour fraction of 3%, this is a piece of evidence of the incipient cavitation situation. Although the monitor point recorded a pressure sufficiently higher than the vapour pressure, localized zones exceeded the limit and a slightly amount of vapour was generated, it was not sufficient to decrease the volumetric efficiency and to delay the inlet valve closing time as one can see in Table 3. According to Franc [4] case 3 describes an incipient cavitation situation.

Table 3: Volumetric efficiencies calculated relative to the standard condition (plunger stroke volumen multiplied by the density of wáter at ambient pressure)

	Inlet Max pressure [kPa]	Volumetric efficiency Vs standard conditions [%]	Inlet valve closing time [crank rotation °]
Case 1	0	93.36	205.3
Case 2	25	97.93	194.6
Case 3	50	98.26	185.8
Case 4	100	98.87	183.7

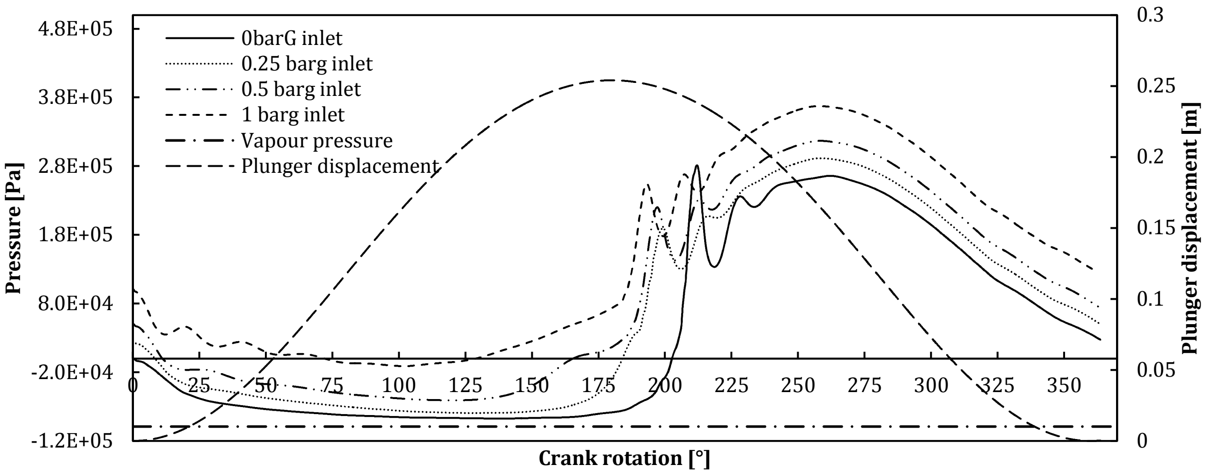

Figure 5: Chamber pressure-time histories, case 1 to 4. The pressure is monitored in a fixed point near the TDC plunger position

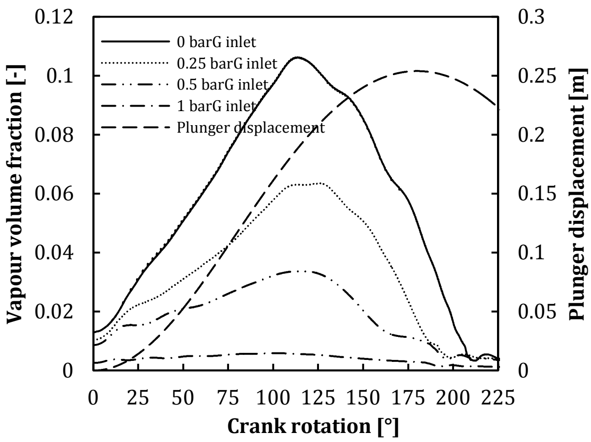

Figure 6: Plunger surfaces mean vapour fraction, case 1 to 4

4 CONCLUSION

A set of four CFD test cases of a PD pump have been created. They were designed to show the typical features of the cavitation condition of incipient, partial and full cavitation plus a further case of null cavitation. The different cavitation regimes were achieved by decreasing the inlet manifold pressure from 100kPa to 0 kPa. The CFD model utilized was a comprehensive cavitation algorithm which included the effects of phase change and the presence of non-condensable gas. The dynamic model of the valves simulated the connection between the pressure field which was affected by the valve lift history and the valve dynamics which depended on the pressure field. This correctly simulated the behaviour of the pump in the full cavitation regime providing reliable information on its volumetric efficiency and on the generation of water vapour by cavitation prediction which can help the designers to extend PD pump operational life.

REFERENCES

[1] A. K. Singhal, M. M. Athavale, H. Li, and Y. Jiang, “Mathematical Basis and Validation of the Full Cavitation Model,” *J. Fluids Eng.*, vol. 124, no. 3, p. 617, 2002.

[2] K. Opitz and E. Schlücker, “Detection of Cavitation Phenomena in Reciprocating Pumps using a High-Speed Camera,” *Chem. Eng. Technol.*, vol. 33, no. 10, pp. 1610–1614, Jul. 2010.

[3] K. Opitz, E. Schlücker, and O. Schade, “Cavitation in reciprocating positive displacement pumps,” in *Twenty-seventh international pump users symposium*, 2011, pp. 27–33.

[4] J. P. Franc and J. M. Michel, *Fundamental of Cavitation*. Kluwer academic publishers, 2004, pp. 1–300.

[5] S. D. A. Pawan J. Singh, “Determination of NPSHR for reciprocating positive displacemenmt pumps: a new approach,” in *Proceedings of the thirteenth international pump user symposium*, 1996, pp. 131–139.

[6] J. Shu, C. R. Burrows, and K. A. Edge, “Pressure pulsations in reciprocating pump piping systems Part 1 : modelling,” *Proc Inst Mech Engrs*, vol. 211 Part I, no. April, pp. 229–237, 1997.

[7] K. a Edge, O. P. Boston, K. C. S. Xiao, K. C. M. J. Longvill, and K. C. C. R. Burrows, “Pressure pulsations in reciprocating pump piping systems Part 2: experimental investigations and model validation,” *Proc. Inst. Mech. Eng. Part I J. Syst. Control Eng.*, vol. 211, no. 3, pp. 239–250, Jan. 1997.

[8] a. Iannetti, M. T. Stickland, and W. M. Dempster, “A computational fluid dynamics model to evaluate the inlet stroke performance of a positive displacement reciprocating plunger pump,” *Proc. Inst. Mech. Eng. Part A J. Power Energy*, vol. 228, no. 5, pp. 574–584, Apr. 2014.

[9] T. Baur, J. Köngeter, and R. Leucker, “Effects of dissolved gas on cavitation inception in free surface flows,” in *Third International Symposium on Cavitation*, 1998, no. April.

[10] ANSYS, “ANSYS Fluent Theory Guide,” vol. 15317, no. November. ANSYS Fluent, 2011.

[11] H. H. Tackett, J. A. Cripe, and G. Dyson, “Positive displacement reciprocating pump fundamentals- power and direct acting types.,” in *Proceedings of the twenty-fourth international pump user symposium*, 2008, pp. 45–58.