Computational Modeling of the Three-Dimensional Flow in a Metallic Stator Progressing Cavity Pump
Emilio E. Paladino; João A. Lima, and Rairam F.C. Almeida, UFRN/DEM/PPGEM, Brazil
Benno W. Assmann, Petrobras, UN-RNCE, Brazil

Abstract
This work presents a novel computational model for the 3D flow in a rigid stator Progressing Cavity Pump (PCP), using an element based finite volume method, which includes the relative motion between rotor and stator. Usual flow models in PCPs consider a Poiseuille flow along the seal lines, i.e., along the positive clearance between cavities in order to predict the internal slip and then, the volumetric efficiency for different pressures, rotations and fluid viscosities. Furthermore, some attempts for more detailed models including computational solutions for the flow in simplified geometries can be encountered in literature. These approaches include, treating cavities as parallel plates or computing the flow between two static cavities, in all cases considering steady state flow, which is a strong hypothesis in this case. Nevertheless no models considering the solution for the full transient 3D Navier-Stokes equations and the relative motion between rotor and stator were encountered. The main challenge at this point was the imposition of the mesh motion and mesh generation process, mainly, because of the mesh quality control (element distortion) in regions near the seal lines, or in the clearance regions between rotor and stator. The model developed is capable to predict accurately the volumetric efficiency and the viscous losses as well as provide detailed information of pressure and velocity fields inside this device. Furthermore, the present model could be used to predict the hydraulic performance of an elastomeric progressing cavity pump after stator wear or deformation and allow for the development of a computational model for the fluid-structure interaction which permits the analysis of the non-rigid stator case.

Introduction
Progressing Cavity Pumping is being more and more used in oil production, mainly in heavy oil fields, due to its numerous technical advantages. Simplest models for PCP design, firstly presented by Moineau (1930), are based on calculating the slippage across the pump, considering a Hagen-Poiseuille flow in the sealing region, which is subtracted from the volume displaced, giving the volumetric flow pumped. As differential pressure increases, so does the slippage, and the relation between differential pressure across the pump and net volumetric flow pumped, can be calculated.

After Moineau’s models, several attempts for more precise fluid dynamic and fluid-structure interaction models have been presented. For oil production applications works due to Robello Samuel & Saveth (1998), Olivet et al. (2002), Gamboa et al. (2002) and Gamboa et al. (2003) constitute the main references in this field of research. Robello Samuel & Saveth (1998) developed optimal relationships between the pitch and the diameter of the stator to achieve a maximum flowrate for multilobe pumps. Olivet et al. (2002) performed an experimental study and obtained characteristic curves and instantaneous pressure profiles along metal to metal pumps for single- and two-phase flow conditions.

Gamboa et al. (2002) presented some attempts of flow modeling within a PCP using Computational Fluid Dynamics with the aim of getting a better comprehension of the flow inside the pump. Nevertheless, attempts for developing a three-dimensional model including rotor motion were failed even for rigid stator (this means constant clearance) due to the complexity of the geometry, mesh motion and (maybe) the inadequateness or limitations of the numerical approach used to solve the governing equations. In virtue of this, Gamboa et al. (2003) presented simplified models for single phase flow considering the possibility of variable gap due to elastomeric stator deformation. The basic approach does not differ too much from previous works based on metallic stator, but the slippage is calculated cavity by cavity and the possibility of a variation of the clearance as function of differential pressure is considered. In this way they were able to reproduce the characteristic non-linear behavior of volumetric flow versus differential pressure in a PCP with elastomeric stator.
Preliminary Analysis

A brief general discussion of the approaches followed by Moineau (1930), and more recently by works due to Gamboa et al. (2002) and Gamboa et al. (2003), previously discussed, is presented in this section. Although several hypothesis are considered for these simplified approaches, the analysis hereby presented is remarkable as allows to understand which geometrical parameters and fluid properties have more significant influence on pump performance and how do they affect it.

Assuming constant value for the clearance along the pump, the slip flow, i.e., the leakage which is subtracted from the theoretical displaced volume in order to calculate the actual volumetric flow rate, can be calculated considering the flow along a parallel flat plates channel, separated by a distance equal to the clearance.

Considering a channel between cavities, i.e., across the sealing lines (see Figure 2), which main geometrical parameters are showed in Figure 1, the pressure drop can be calculated as

\[ \Delta p = \rho \frac{U^2}{2} \frac{L}{D_h} \]  

where \( f \) is the friction factor and \( D_h \) is the hydraulic diameter, defined as four times the cross sectional area over the perimeter. Recalling that the mean velocity across the channel corresponds to the volume flow rate divided by the cross sectional area, the pressure drop can be calculated as,

\[ \Delta p = \rho \frac{L}{4b^2w^3} S^2 \]  

In equation above, it was considered that \( w \ll b \) for when calculating the hydraulic diameter. For laminar flow, the friction factor can be calculated as

\[ f = \frac{C}{Re} \] where \( Re = \frac{2\rho S}{\mu b} \]

where \( S \) is the slip flow or leakage through the pump. The generic constant \( C \) was used because, as this analysis does not intend to be quantitative, but just qualitative and the channel geometry is not, \textit{a priori}, known. Nevertheless, for laminar flow we can assume that friction factor is inversely proportional to the Reynolds number. At this point, it is important to say that one of the strongest hypotheses in these approaches is the simplification of the channel geometry as the actual sealing is produced in a convergent-divergent geometry.

Following previous equations, the slip can be calculated as,

\[ S = \frac{8bw^3\Delta p}{C\mu L} \]  

This slip flow can be subtracted from the theoretical volumetric flow rate displaced by the pump, which depend just on geometrical parameters, in order to obtain the actual flow rate as function of the pressure drop. Then, from this simple analysis the pump performance can be related, at least qualitatively, to the main pump geometric parameters and fluid properties.

In order to turn this analysis quantitative, the assumed channel dimensions have to be related to the pump geometric parameters. One of the main difficulties at this point is to define the channel length, \( L \) as, in the real geometry, this “channel” is actually a convergent divergent section.

Based on an approach similar to the hereby described, Gamboa et al. (2003) presented a more detailed model, which calculates the slip between cavities as the sum of two components; the flow “along” the pump and the flow “across” it. These components of the slip are showed schematically in Figure 2.
Nevertheless, it is also not clear in that work how the channel length was calculated (for both transversal and longitudinal flow). It is just stated that it must be much greater than the clearance. Then, at least in works reviewed by the authors, this channel length seemed to be more a fitting parameter, than a physical based one.

This is not a problem for the purpose of this analysis as it intends to be qualitative. From equation (4) obtained from a very simple analysis some points can be highlighted, which are also observed in experiments:

- For laminar flow, which has been assumed for this analysis, the slip depends linearly on the pressure difference across the pump ($\Delta p$)
- In this case, the fluid viscosity ($\mu$) has an inverse linear influence too
- The clearance ($w$) appears elevated to the cube, which means it has a strong influence on the volumetric efficiency
- Although quantitatively, the length $L$ can be related to the pump length, i.e., to the number of stages of the pump which has also a inverse linear influence on the volumetric efficiency
- Fluid density does not influence the volumetric efficiency.

The last conclusion arrives because fully developed Hagen-Poiseuille flow has been assumed along the seal region. This means a balance between pressure and viscosity forces, neglecting the inertial forces. Then, the rotation velocity does not influence too. Nevertheless, the rotation velocity may have some influence on volumetric efficiency as, for high rotation, the inertial forces at the sealing regions can be important. This fact can explain the failure of this approach in predicting performance for low viscosity fluids as water, reported in Gamboa et al. (2003), not just because of the turbulence developing at the sealing regions, but because the assumption of fully developed flow is not valid anymore. Note also that, even at the same rotation velocity, as viscosity goes down inertial forces became more important on the momentum balance.

Computational Model

Computational models for the flow in a Progressing Cavity Pump were implemented in CFX11 (ANSYS, 2008). This software is based on a discretization of the governing equations using an Element Based Finite Volume Method (Maliska, 2004; Ferziger & Peric, 2001) and a coupled approach for solving the pressure-velocity decoupling (Raw, 1985).

The main challenges of computational fluid dynamics modeling for the flow in a metallic stator PCP are the mesh generation process and the imposition of rotor-stator relative movement. The main problem in the mesh generation process is the mesh generation in regions near the sealing lines, because of the high aspect ratio of the resulting elements.

In addition, the mesh topology should be such that allows large deformation of the elements, which dimensions go from order of clearance to order of eccentricity along half revolution of the rotor. The element deformation is depicted in Figure 3.

In order to maintain the conservativeness of the numerical method used (Finite Volume Method) the mesh topology should be maintained. This means that the elements cannot be suppressed for the domain, as rotor approaches the stator. Then simulations with too small clearances and/or systems operating with interferences are not possible by this approach due to the arising of negative elements.

Several topologies were evaluated in order to get a full hexahedral mesh with good element quality. Independently of the numerical definition of “element quality” in a computational mesh, the concept is understood as, the highest the quality, the
lower is the element linear and angular distortion, which is known to generate bad conditioned matrices in equation discretization process and so numerical instabilities. In the extreme cases this lead to solver failure. This is a common problem in numerical simulations with moving meshes; even having a good “initial mesh”, mesh motion could lead to highly distorted elements if some cares are not taken.

As an initial approach, after generated a good quality mesh, simulations were performed by imposing the mesh motion directly on the solver used for simulations. In this case the rotor wall motion is imposed and the motion of the internal nodes of the computational mesh is calculated considering a linear elastic deformation. The problem observed with this approach is that, due to the effects of numerical diffusivity on the calculation of mesh motion (internal nodes) a hysteresis like process comes out leading to element distortion along rotor revolutions and the consequent solver failure due to element inversion. The meshes for the same rotor position at two subsequent revolutions are showed in Figure 4.

![Mesh for initial time](image1)
![Mesh for 8th time step](image2)

**Figure 4 – Element distortion due to numerical diffusivity on the mesh motion calculation**

The alternative used to impose the mesh motion was to specify each node position for each position of the rotor. This was accomplished by generating several meshes along one revolution for different rotor positions. Then meshes are read along the solver run for each timestep, and the new nodes positions are calculated. In order to do this, a bi-univocal mapping has to be imposed between nodes of meshes for subsequent timesteps. This process is defined through a FORTRAN user routine in CFX11 solver (Ansys, 2008). In addition, structured hexahedral meshes have been used to have control of number of nodes in each direction. Meshes were generated using ICEMHEXA package (Ansys, 2008), which generate hexahedral structured multiblock meshes. This does not means that a structured mesh solver was used but the structures multiblock philosophy was used in mesh generation process. In this way, the same block topology is used for all meshes, just adjusting the rotor position.

Figure 5 shows the topology used for mesh generation in the simulations presented in this work, which permitted the mesh generation for geometries with clearances about 0.1 millimeter, maintaining a good mesh quality.

![Entire pitch](image3)
![Quarter pitch](image4)

**Figure 5 – Block topology used for the structured multiblock mesh generation**
Figure 6 presents some snapshots of the computational mesh used for simulations, for different rotor positions, for a clearance of 0.185 mm. Other geometrical parameters as stator pitch, rotor diameter and eccentricity are the same as presented in Gamboa et al. (2002).

Figure 6 – Snapshots of the mesh at the inlet region

**Results**

Computational simulations were performed for the geometry used by Gamboa et al. (2002) and Olivet et al. (2002) in experimental studies, in order to validate the computational model implemented. Results for rigid stator and single phase flow were reproduced, as this is the situation which the actual model is able to represent, although these works present results for a big range of operational conditions and both rigid and elastometric are considered. Nevertheless, it is not clear in these works which geometrical parameters were used in each experiment. For instance, although the clearance used in rigid stator cases was reported as being 0.370 mm, in all experiments, there are some contradictions in the definition of the clearance value among works of the same researchers. In Gamboa et al. (2002) this parameter is defined as the difference between stator and rotor radius, which represent the minimum distance between rotor and stator. In a later work (Gamboa et al., 2003) the same parameter seems to be defined just as the difference between radius over two giving this minimum distance, for the geometry describes in these works 0.370/2=0.185 mm. From results obtained in this work for both values of interference, 0.370 mm and 0.185 mm, the last value seems to be the correct one, as results approximates much more to experiments using this value for the clearance. In addition it is also not clear the number of stator pitches (i.e. pump length) of the pump used in experiments (three or five).

In virtue of this situation, several cases were run, considering different values for clearance and number of pitches. It is important to leave clear, however, that the development of pump characteristic curves is beyond the objectives of this work. Results presented here intend to validate the computational model implemented and show its potential to solve design and operational problems which require a detailed representation of the flow within the pump.

Figure 7 (a) shows the volumetric flow pumped as function of differential pressure for different values of clearance, compared against experimental results from Gamboa et al. (2002). In all cases the rotation velocity was 300 rpm and a three pitches stator pump was considered. It is clear that, even though, pump performance seems to be a bit underpredicted for w=0.185 mm, results for w=0.370 mm are too far from experiments. Figure 7 (b) shows the same curves for different number of stator pitches. For the case of five pitches, there is some overprediction on volumetric flow, but results are very near from experiments. This is because, as commented, the pump performance has a cubic relation whit clearance but a linear one with number of pitches (or pump length). Then, although seems to be clear that the value of 0.370 mm for clearance was not used in Gamboa et al. (2003) experiments, the uncertainties about the number of pitches still persist.
Figure 7 – Volumetric flow rate obtained from simulations, compared with Gamboa et al. (2002) experimental results, for different clearances and pump lengths.

Figure 8 shows the results obtained for the presented computational model for different rotation velocities, compared with Gamboa et al. (2003) results. The number of pitches and clearance were considered as three and 0.185mm, respectively. The same behavior as previous results can be observed, for lower rotation velocities.

![Figure 8](image)

The described model was also run for the water flow case. Results for 300 and 400 rpm and three and five stator pitches are showed in Figure 9 (a) and Figure 9 (b), respectively. Once again there are some doubts related to the geometric parameters of pump used in experiments. Nevertheless, in this case, even for the longer pump (five pitches), results are still underpredicted. This fact was also related by Gamboa et al. (2003) as the model presented in that work failed for low viscosity fluid. Although the present results are much near from experiments than those obtained by Gamboa et al. (2003) with the approach presented in that work, the cases for low viscosity fluid should be object of further research.

![Figure 9](image)

Figure 9 – Volumetric flow rate obtained from simulations, compared with Gamboa et al. (2003) experimental results, for water flow.
The main advantage of the 3D CFD model is that it provides detailed information of the flow field inside the pump. Figure 10 shows the pressure distribution along the pump stator. This information can be important, for instance, to calculate the stator deformation in a polymeric stator PCP, which is one of the next steps of this research.

Figure 10 – Pressure distribution along the pump stator

Figure 11 shows a local zoom of the pressure distribution at the outlet region pump stator. The continuous and dashed lines represent respectively the seal lines for the longitudinal and transversal slip flows (see Figure 2). This confirms the hypothesis employed in Gamboa et al. (2003) that the slippage can be approximated by the sum of these two contributions. In addition, it can be seen that the “channel” along the longitudinal seal lines is subjected to a greater pressure drop than the transversal one, as was considered in Gamboa et al. (2003) model. For the case of rigid stator, where pressure distribution along cavities is approximately linear, the differential pressure between cavities at sides of longitudinal sealing is twice the value for the transversal case.

Figure 11 – Pressure distribution along the pump stator along three cavities

Figure 12 shows the pressure distribution along the pump length for the cases of three and five steps, with 400 rpm, and clearance equal to 0.185 mm. Water was used as the working fluid and the maximum differential pressure over the PCP was

Longitudinal slippage

Transversal slippage
\[ \Delta P = 15 \text{ psig.} \] It can be verified that the pressure distribution along cavities is actually linear (even for low viscosity fluid), which is consistent for the case of rigid stator.

![Figure 12 – Pressure distribution along pump length](image)

**Conclusions**

A detailed CFD model for a rigid stator PCP, considering the rotor motion, was successfully implemented. This model can provide detailed information about the pump performance for different geometrical parameters and operation conditions. This model represents the first attempt, at least in literature review, for a complete flow simulation in real pump geometry, considering the rotor motion, which in previous works was considered unrealizable.

Results obtained are consistent with experiments due to Gamboa et al. (2003) and Olivet et al. (2002). Although small discrepancies are observed between numerical results and experiments, there are some uncertainties regarding the geometrical parameters and fluid properties employed in experiments. For the case of low viscosity fluids, larger discrepancies are observed and will be subject of further investigations.

A simplified analysis was also presented which provide some qualitative information about the impact of the main PCP geometrical parameters and fluid properties on pump performance. Numerical results are in concordance with expected trends, pointed by this analysis.

The computational model developed can became a very useful tool for PCP design an operation optimization and control. Detailed pressure fields available allow the development of fluid-structure interaction models which could contemplate the elastomeric stator case. In addition, this model, based on the full solution of the Navier-Stokes equations within the PCP, can be extended for the multiphase flow situation which is the most common case in oil field applications.

**References**


Acknowledgements

Authors are grateful to CENPES/PETROBRAS for the financial support of the present research project.