A CFD Model for a Progressing Cavity Pump

Emilio E. Paladino*, João Alves de Lima†

PPGEM - Graduate Program in Mechanical Engineering, Federal University of Rio Grande do Norte, CEP 59072-970- Natal - RN

* e-mail: emilio@ct.ufrn.br
† e-mail: jalima@dem.ufrn.br

ABSTRACT

The growth of Progressing Cavity Pump as artificial lift system in the last years lead to the development of models for the flow behavior within these devices. Based on the ideas of the system creator, Rene Moineau, usual flow models attempt to establish relations between differential pressure and flow rate by considering a Poiseuille flow along the seal lines between cavities in order to predict the internal slip which is subtracted from the displaced volumetric flow rate. In addition, some attempts for more detailed models including computational solutions for the flow in static simplified geometries can be encountered in previous works. Nevertheless no models considering the solution for the full transient 3D Navier-Stokes equations and relative motion rotor and stator were found in literature. This work presents a computational model for the unsteady 3D flow, using an element based finite volume method, which includes the relative motion between rotor and stator, in a Progressing Cavity Pumps. The computational model for the flow in a Progressing Cavity Pump was implemented in CFX11 [1]. This software is based on a discretization of the governing equations using and Element Based Finite Volume Method ([2]; [3]; [4]) and a coupled approach for solving the pressure-velocity decoupling ([5]).

The numerical flow computation within positive displacement pumps is, in general, a challenging task as it normally requires moving mesh simulations. Furthermore, depending on the pump type, the need of discretization of small clearances between rotor and stator introduces serious difficulties into the mesh generation process. In addition, for the specific case of PCPs, the pump kinematics are very complex (when compared with a reciprocating pump, for instance). the main challenge in this work was the imposition of the mesh motion and mesh generation process, mainly, because of the need of mesh quality control (element distortion) in regions near the seal lines.

The model developed is capable of the accurate prediction of volumetric efficiency and viscous losses as well as provide detailed information of pressure and velocity field inside this device. This could allow, for example, the prediction of local stator deformation in order to predict how this influences on slip, the accurate treatment of turbulence effects, by using advanced turbulence models, and the model extension for the case of multiphase flows, which is a common case in artificial lift.

The model was validated against experimental results from literature, as can be seen in Figure 1. The model successfully predicts the performance for high (oil 42 cP) and low (Water) viscosity fluids. For the case of low viscosity fluids, simplified models based on Pouseuille flow in seal regions, use to fail [6]. Some aspects related to the dynamic behavior of the flow, not captured by the simplified models, are analyzed using this model. In Figure 2, the pressure distribution along the pump stator is showed. This information, no available in simplified models, allows the calculation of the stator deformation, in the case of elastomeric stators.
**Figure 1** - Comparison of model results with experimental results from [7]

**Figure 2** - Pressure distribution along the stator for different rotor positions

**References**