Journal of Petroleum Science and Engineering 78 (2011) 178–192

Contents lists available at ScienceDirect

Journal of Petroleum Science and Engineering

journal homepage: www.elsevier.com/locate/petrol

A computational model for the flow within rigid stator progressing cavity pumps

Emilio E. Paladino *, João A. Lima, Paulo A.S. Pessoa, Rairam F.C. Almeida

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

ARTICLE INFO

Article history:
Received 22 October 2010
Accepted 1 May 2011
Available online 11 May 2011

Keywords:
Progressing Cavity Pump
3D Model
transient Model
moving Mesh

ABSTRACT

This work presents the development of a novel computational model for the 3D-transient flow in a Progressing Cavity Pump (PCP) that includes the relative motion between the rotor and stator. The governing equations are solved using an Element-based Finite Volume Method in a moving mesh. The model implementation is only possible due to the meticulous mesh generation and motion algorithm described herein, which is considered to be one of the main contributions of the present work. The model developed in this study is capable of precisely predicting volumetric efficiency and viscous loses in addition to providing detailed information about the pressure and velocity fields inside the device. Turbulence effects are accurately treated with advanced turbulence models. In addition, although the presented results are for single phase flow, the model can be extended to account for multiphase flows using models available in CFD software. In addition, some aspects related to inertial effects that are not captured by simplified models are analyzed using this model. The results presented herein consider a rigid stator pump. The model was validated against experimental results from literature.

1. Introduction

Progressing Cavity Pumps (PCP) are positive displacement pumps with an operation principle based on the eccentric motion of a single rotor, which displaces the fluid contained in cavities from low to high pressure regions. Hydraulic seals are required between moving and static parts to avoid counter-flow (from high to low pressure regions), which depletes the pump efficiency.

These seals are promoted in two ways: 1) by generating interference between the rotor and stator, in which case the stator must be deformable (elastomeric), or 2) by leaving a small clearance between them, where the sealing is dynamically accomplished through the viscous pressure drop along the clearance. Even in the case of interference between the rotor and stator, because the stator is deformable, a clearance could eventually appear as the pressure increases, as will be explained later.

Since their invention by Moineau (1930), PCPs have been successfully used for pumping high viscosity fluids or slurries, mainly in food and cosmetic industry, but it was the growth of the application of these pumps for oil artificial lifts in low to medium depth oil wells since the 1970s (in several cases substituting for traditional reciprocating pumps) that lead to the development of more detailed flow models and experimental studies within these devices in recent years. Among the main advantages of this system for artificial lifts are its ability to pump heavy oils, tolerate high percentages of free gas and high efficiency can be quoted (Revard, 1995, Cholet, 1997).

This work presents a computational model for unsteady 3D flow in single lobe progressing cavity pumps (Paladino et al., 2008; Lima et al., 2009; Paladino et al., 2009; Pessoa, 2009; Almeida, 2010), which includes the relative motion between the rotor and stator using an Element-based Finite Volume Method (Baliga and Patankar, 1980; Raw, 1985). Full 3D transient velocity fields are obtained from the model together with pressure fields. Therefore, all flow variables, including the flow rate, hydraulic torque, and efficiency, can be evaluated. Turbulence effects are properly treated through the use of advanced turbulence models. The model can be extended for heat transfer calculations and multiphase flows, which are common in artificial lift applications as long as adequate interfacial transfer relations are included. All available multiphase closure models in CFD packages can be used. The detailed understanding of the flow behavior within progressing cavity pumps is of fundamental importance for designing, optimizing and operating PCP artificial lift systems.

This model intends to be an additional tool for PCP systems design and operation and not a substitute for experimentation or pilot/in-field testing. Nevertheless, experimentation is expensive, and obtaining local measurements of pressure, velocity or temperature fields is difficult. In addition, real operational conditions, such as downhole pressures and temperatures are difficult (if not impossible) to reproduce in laboratory tests, but can be readily simulated through the computational model presented in this paper. Once the model is validated through experiments developed under realizable laboratory conditions, it can be used as a prediction upscale tool under real operational conditions.

* Corresponding author, currently at: SINMEC Lab, Federal University of Santa Catarina, Brazil. Tel.: +55 48 3721 9562.
E-mail address: eepaladino@gmail.com (E.E. Paladino).

0920-4105/$ – see front matter © 2011 Elsevier B.V. All rights reserved.
doi:10.1016/j.petrol.2011.05.008

To date, at least to the authors' knowledge, this model is the first successful attempt for implementing a CFD model for laminar/turbulent flow simulations in this type of pump. Difficulties in mesh generation within the fluid domain and the imposition of a moving mesh to define the rotor kinematics were identified in the literature as the main restrictive factors for CFD modeling of PCPs (Gamboa et al., 2002). Element distortion becomes a serious problem in the discretization of sealing regions, where clearances between the rotor and stator are on the order of 10^{-4} or less of the global length. Then commercial mesh generators based on geometric interpolation, such as ANSYS-ICEM® (ANSYS Inc., 2010), have limited usability, even when employing structured hexahedral multi-block meshes and "smoothing" techniques, such as element orthogonalization based on Poisson equations. In the present work, a mesh generation algorithm, including the mesh motion defined by the pump kinematics, was developed and fully integrated through FORTRAN user routines with the ANSYS-CFX® commercial CFD package (ANSYS Inc., 2010). This algorithm and its implementation in the CFD package are described in detail in the next sections, and the algorithm can be considered as one of the main contributions of the present work.

The usual flow models based on the ideas of the PCP creator, Rene Moineau (Moineau, 1930), attempt to establish relations between differential pressure and flow rate by subtracting the counter-flow leaked through the seals from the displaced flow rate (Gamboa et al., 2003; Pessoa et al., 2009). Because the displaced flow rate depends only on the pump geometry and kinematics, fluid models usually attempt to calculate the leakage, or "slippage"[1], fluid properties and the differential pressure along the pump. As the differential pressure increases, so does the slippage, and the relation between the differential pressure across the pump and the net pumped volumetric flow can be calculated. Still, although several works related to the application of PCPs and control in artificial lift systems have been published, few references attempt to characterize the flow within PCPs.

Robello and Saveth (1998) developed optimal relationships between the pitch and the diameter of the stator in multi-lobe pumps to achieve a maximum flow rate. Their work focused on the geometrical parameters of the pump and their influence on the displaced flow rate but do not mention the slippage of the influence of the differential pressure or fluid properties on the pumped flow rate.

Gamboa (2000) attempted, albeit unsuccessfully, to develop the first 3D numerical study of the single-phase flow within a metallic stator PCP. His group continued to develop experimental studies, so that later, works of Martin et al. (1999), Suarez (2002), Olivet (2002), Olivet et al. (2002) and Gamboa et al. (2002) presented an extensive experimental study in PCPs for single- and two-phase flow conditions and obtained characteristic curves and transient pressure profiles within cavities in metallic (rigid) stator pumps; however, no flow model was developed.

Gamboa et al. (2002) presented some attempts of flow modeling within a simplified geometry of a PCP using Computational Fluid Dynamics with the intention of obtaining a better comprehension of the flow inside the pump, specifically in the sealing region, which governs the pump efficiency. Nevertheless, attempts for developing a three-dimensional model including the rotor motion failed due to the complexity of the geometry, the difficulties of the imposition of mesh motion and, in the view of the authors of the present work, the inadequateness or limitations of the numerical approach used to solve the governing equations.

Recognizing those limitations, Gamboa et al. (2003) presented a simplified model for single-phase flow by considering the possibility of variable gap due to elastomeric stator deformation. In this way, his model was able to reproduce the characteristic non-linear behavior of volumetric flow versus differential pressure in PCPs with elastomeric stators. The model is similar to those presented in other works based on the Moineau's approach, but, in this case, the slippage is calculated by considering two components, transverse and axial, and taking into account that sealing regions can be subjected to different pressure differences.

More recently, another interesting simplified model was presented by Andrade (2008) who solved the flow within a "developed" pump; i.e., the author solved for the flow between two non-parallel surfaces, whose local separation corresponds to the distance between the rotor and stator. Under the consideration that the distance between the rotor and stator (clearance) is much smaller than the rotor radius, the inertial terms and radial velocity component are neglected in the transport equations and the momentum equations can be analytically integrated, and the pressure is calculated from a Poisson equation that arises from a mass conservation equation. This approach presents good results for viscous fluids, but it is not suitable for low viscosity fluids, where inertial terms become important and the flow can eventually become turbulent.

Nevertheless, after an extensive literature review, no flow models that attempt to determine the solution for the full transient 3D Navier–Stokes equations within a PCP, including the relative motion between rotor and stator, as is proposed and developed in the present work, have been found.

1.1. PCP operation principle and performance curves

This section aims to briefly describe the operational principle of PCPs, to understand the shortcomings of the flow modeling and expected results. A detailed explanation of PCP design and operation is beyond the scope of this paper but can be found, for instance, in references (Revard, 1995; Cholet, 1997; Nelik and Brennan, 2005).

PCPs are positive displacement pumps, like screw pumps, but, in this case, the fluid displacement is promoted through the eccentric motion of a rotor (which is why these pumps are also called "eccentric screw pumps"). The cavities are isolated through seal lines, as shown in Fig. 1, and the fluid within them is displaced from low to high pressure regions. As the total differential pressure increases, the pump length, and thus the number of cavities, need to be increased to reduce the differential pressure between adjacent cavities, which reduces the slippage. (See Section 2.2 for the definitions of the variables.)

The sealing effect is obtained in a fine hydrodynamic, or elastomeric, region between the cavities. Depending on the pump type, a clearance or interference can be present between the rotor and stator, as shown in Fig. 1b and c, respectively, which depict a transverse section of the fluid region in a PCP. For the case of interference between the rotor and stator, a non-rigid (elastomeric) stator has to be used to allow for the deformation imposed by the rotor motion. In this situation, the sealing effect is naturally achieved by this interference. However, for flow modeling purposes, it can be considered that, during regular operation, a fine liquid film is present between the rotor and stator. This hypothesis is important in terms of flow modeling, because a single connected domain must be considered for the whole fluid domain to avoid isolated fluid regions.

As previously stated, the volumetric flow rate in a PCP can be calculated as the volume displaced in each revolution multiplied by the rotation, which is sometimes called the theoretical flow rate and does not depend on Δp, and subtracting the leaked flow, or "slippage", across the seal regions. Several factors affect the slippage, including the clearance between the rotor and stator, the fluid viscosity and the pressure drop between the cavities (which depends on the pressure drop along the whole pump).

Fig. 2 shows schematically the typical expected shape of performance curves for elastomeric and metallic stators. For the case of elastomeric stator, the pump operates with interference

[1] This term is commonly used in PCP and screw pump terminology to describe the process wherein a fluid is displaced axially through the pump, and the counterflow "slips" over the displaced flow.