Computational Modeling of the Three-Dimensional Flow in a Metallic Stator Progressing Cavity Pump

Emilio E. Paladino; João A. Lima and Rairam F. Almeida

UFRN / DEM / PPGEM, Brazil

Benno W. Assmann

PETROBRAS / UN-RNCE, Brazil
Introduction

• Since Moineau (1930), several approaches for evaluate PCP performance have been employed.

• Robello & Saveth (1998), Olivet et al. (2002), Gamboa et al. (2002) and other works from these groups, constitute main references for this research field, theoretical or experimental

• Experiments are expensive and normally provide only global values

• Simplified approaches have been proposed, but could be inaccurate or fail in complex situations
Introduction

• Simplified models
  – Quick responses to changes in operational variables or geometrical parameters
  – Only global parameters could be obtained (like )
  – Unusable for complex situations as multiphase flow (most common operation situation)
  – Need for guessed parameters

• Three dimensional computational model
  – Detailed knowledge of the flow field for geometry and operation optimization (too costly if done experimentally)
Introduction

• Three dimensional computational model (cont.)
  – Torque (and so power) can be calculated from model, including hydraulic losses
  – Correct representation of the flow field (Hagen-Poiseulle flow hypothesis can be too strong in some cases)
  – Multiphase flow can be simulated, having adequate models for interface morphology and momentum transfer
Preliminary Analysis

- General aproach in simplified models:
- Hypotheses
  - $\dot{V}_{Pumped} = \dot{V}_{Displaced} - \dot{V}_{Slip}$
  - Hagen-Pouisille flow in constant area channel for slip calculation

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

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

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

• Assuming that the volumetric flow can be calculated as:

\[ \dot{V}_{\text{Pumped}} = \dot{V}_{\text{Displaced}} - \dot{V}_{\text{Slip}} \]

  \begin{align*}
  \text{Depends on Geometric Parameters and RPM} & & \text{Depends on Geometry, } \Delta p \text{ and Fluid Properties} \\
  \Rightarrow \eta_{\text{Vol}} &= 1 - \frac{\dot{V}_S}{\dot{V}_D}
  \end{align*}

• Some conclusions can be obtained
  
  – For laminar flow the slip depends linearly on \( \Delta p \text{ and } \mu \)
  
  – The clearance \( (w) \) appears elevated to the cube, which means it has a strong influence on the volumetric efficiency
  
  – The channel length \( L \) is estimated in this kind of approaches but can be related to the pump length and has also an inverse linear influence on volumetric efficiency
Computational Model

• Main challenges
  – Mesh generation
  – Mesh motion set-up

• Characteristics
  – Full 3D, transient detailed model
  – Complete Navier-Stokes equation solved within the fluid region (Allows for turbulence and multiphase modeling)
  – Element Based Finite Volume Method used for equation discretization
  – Fully coupled solver for pressure and velocity (mass and momentum equations)
Computational Model

- The mesh generation process was a great challenge because of the element aspect ratio in near clearance regions.
- Furthermore, mesh motion imposes huge element deformation along time.
- Imposing the mesh motion directly in CFD code resulted in element distortion due to numerical diffusivity on the mesh motion calculation.
Computational Model

• Mesh motion
  – Imposed into the whole mesh (at each node)
  – Meshes generated automatically for each timestep
  – FORTRAN CFX User routines needed
Results and Discussion

- Model Validation
  - Gamboa et al. (2002) experiments used to validate the model
  - Some confusion with used parameters
  - Experiments data:
    - Geometry
      - Pitch number
        - 0.187 mm
    - Fluid properties:
      - \( \mu = 1 \text{ cP} \)
      - \( \rho = 1000 \text{ kg/m}^3 \)
      - \( \mu = 42 \text{ cP} \)
      - \( \rho = 868 \text{ kg/m}^3 \)

<table>
<thead>
<tr>
<th>Stator Pitch</th>
<th>119.99 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clearance</td>
<td>0.187 mm</td>
</tr>
<tr>
<td>Pitch number</td>
<td>6(?)(5) (?)</td>
</tr>
<tr>
<td>Rotor Diameter</td>
<td>39.878 mm</td>
</tr>
</tbody>
</table>
Results and Discussion

- Volumetric flow vs. Pressure

Different clearances
Different number of pitches
Results and Discussion

- Volumetric flow vs. Pressure
  - Different RPM
Results and Discussion

• Volumetric flow vs. Pressure
  – Water flow (1cP)
Results and Discussion

- Volumetric flow vs. Pressure
  - Eddy viscosity turbulent model (turbulence modeling study running on)
Results and Discussion

- Pressure distribution

EnLithen Here
Results and Discussion

• Pressure distribution
Results and Discussion

- Velocity at clearance and pressure along the rotor
  - Strong transient flow in sealing regions
Conclusions

- A detailed CFD model for a rigid stator PCP, considering the rotor motion, was successfully implemented.
- Provides detailed information about the pump fluid dynamics inside the PCP,
- This allows to calculate performance and can be used for geometrical and operational optimization.
- Detailed pressure fields available allow the development of fluid-structure interaction models in order to study the elastomeric stator.
Further work

- Implementation of the Fluid-Structure Interaction into the model in order consider the elastomer stator
- Analysis of the influence of the turbulence modeling
- Multiphase flow modeling (Experimental support needed)
- Etc.