Fluid structure interaction in piston diaphragm pumps

Ir. Ralph van Rijswick\textsuperscript{1,2}, Prof. Dr. Ir. Cees van Rhee\textsuperscript{1}

\textsuperscript{1} Department of Dredging Engineering, Delft University of Technology, Mekelweg 2, 2628 CD Delft, The Netherlands
\texttt{R.J.A.vanRijswick@tudelft.nl}

\textsuperscript{2} Weir Minerals Netherlands b.v., Egtenrayseweg 9, 5928 PH Venlo, The Netherlands

ABSTRACT

Piston diaphragm pumps are used world-wide for the transport of aggressive and/or abrasive fluids in the chemical, mining and mineral processing industries. Figure 1 shows a cross section of a piston diaphragm pump as is used in the mining and mineral processing industries for the transport of mineral ores and tailings in the from of a solid water mixture, called a slurry.

![Cross section piston diaphragm pump](image)

The crankshaft of the pump is driven by an external electric motor via a gear reducer and generates a reciprocating motion of the piston. The piston displaces a hydraulic fluid, often a mineral oil, which then displaces a rubber diaphragm. The rubber diaphragm displaces the pumped fluid and with the help of self-acting non-return valves a positive displacement pump action is generated.

The rubber diaphragm is a key element within the pump and the reliability of the pump is for a large part dependent on the fatigue life of the diaphragm. A typical piston diaphragm pump is operated around 60 stokes per minute which results in approximately $30 \times 10^6$ load cycles of the diaphragm per year in a continuous duty service. The overall deformation of the diaphragm is driven by the volumetric displacement of the piston but the exact deformation is determined by the fluid loads acting on the diaphragm. In current industry practice the flow field in the pump chamber and the resulting diaphragm deformation is for a large part unknown. Diaphragm designs have evolved from trial-and-error while size and speed selection is mainly based on scaling of operational experiences. Only limited, structure only, Finite Element Analyses (FEA) have been performed using estimated and assumed fluid loads. For better understanding of the diaphragm deformation a PhD project has been initiated within the Dredging Engineering department of Delft University of Technology with the following objective:

Development of an experimentally validated numerical fluid structure interaction model for the prediction of operation condition induced diaphragm deformation and strains in piston diaphragm pumps
The numerical model to be developed should include 2 fluid domains, the propelling fluid and the pumped slurry, both of non-rectangular dimensions given by the chamber geometry which interact with a structural model for the flexible rubber diaphragm. In order to model the industry relevant application a 3D model is required. Common approaches in modeling Fluid Structure Interaction (FSI) problems are based on the Arbitrary-Langrangian-Eularian (ALE) formulation of the fluid flow problem [1]. In the ALE approach the fluid mesh is deformed during the simulation as the structural deformation displaces the fluid domain boundaries. The advantage is that an unstructured, body-fitted mesh can be used for the discretization of the fluid flow equations. However, the disadvantage is the large distortion of the fluid mesh when it is deformed. In this case the displacement of the diaphragm is of the same order as the dimensions of the fluid domain, actually the design objective is to use as much fluid displacement within a given pump chamber volume as possible. This would result in severe mesh distortion which most likely would require re-meshing during simulations. In order to eliminate these mesh distortion issues, a model based on the Immersed Boundary Method (IBM) has been set up. The model uses a uniformly distributed rectangular grid in which the chamber housing and diaphragm are immersed. The interaction of the diaphragm with the fluid flows is based on the feedback forcing IBM as originally developed by Peskin, [3]. A lagrangian mesh is used for the deformable structure which is immersed and simply advected by the fluid flow. The basic principle is to advect and subsequently calculate the nodal reaction forces of the immersed deformable structure every time step and to feed the nodal reaction forces back into the flow solver as body forces. As the structural nodes are not aligned with the fluid cell centers, a bell-shaped distribution function is used to distribute the nodal reaction force over several fluid cells. The presence of the solid pump chamber is modeled using a direct forcing IBM described by Fadlum [2]. In this case the body forces can be directly calculated from the Navier-Stokes equations by forcing the local fluid velocity to the velocity of the solid domain, which is zero in this case. The model is set up as basic as possible and characterized by the following:

- Incompressible Navier-Stokes solver assuming laminar flow
- Explicit first order Euler time integration
- Rectangular staggered grid
- Hybrid differencing [4] of advective terms which switches between upwind and central depending on local Peclet number
- Finite volume, cell-centered, multigrid Poisson solver [5]
- Immersed boundary method using direct forcing in a strain case approximation of the solid boundaries [2]
- Immersed boundary method using feedback forcing for modeling interaction with flexible diaphragm [3]
- Current 2D model set up from scratch, fully vectorized and implemented in Matlab

A screen shot of the current 2D model is shown in figure 2.

Figure 2: Screen shot of current 2D model in Matlab
The disregard of any turbulence effects and the stair case approximation of the solid boundary might seem a rather rough and inaccurate approach. However, dimension analyses and numerical experiments have shown that the fluid flow in the pump chamber is inertia dominated by both unsteady as well as convective acceleration, which makes accurate modeling of viscous losses, turbulent mixing and wall shear stresses less important.

The modeling of the structural deformation of the diaphragm is relatively straightforward in 2D. The diaphragm is modeled using a string of markers which are immersed in and simply advected by the fluid flow. The markers are interconnected using linear springs and each triplet of markers acts as a hinge with a torsional spring at the central marker of each triplet. Every time step the reaction forces on the markers are calculated form the deformation of the linear and torsional springs which are then fed back into the fluid flow as body forces using a bell-shaped distribution function. As the reaction forces can be calculated in the deformed state of the structure, geometric non-linearity is automatically included in the model. This becomes much more complicated when using 3D solid or shell finite element formulations as is required for the final 3D model. As both the flow solver as well as the structural model are fully explicit, a relatively simple program results which is easy to vectorize and lends itself for future parallelization. Only the Poisson solver is implicit for which a relatively efficient multi-grid solver has been implemented.

Currently a 2D model has been set up which is used as a development platform and is used for basic parameter studies of which some results will be shown. Current activities focus on both extension of the model to 3 dimensions as well as experimental verification for which a test rig has been designed and commissioned. The extension of the flow solver to 3D is relatively straightforward but the 3D formulation of the diaphragm deformation is more involved. Here rotation-free 3D shell finite element formulations are required for proper interaction with the fluid flow as nodal forces can be supported more easily than nodal moments in the fluid with body forces.

REFERENCES


