Hydrodynamic Modeling of a Rotating Cone Pump Using COMSOL Multiphysics™

Anoop Uchagawkar, Mikhail Vasilev and Patrick L. Mills*
Texas A&M University-Kingsville, Department of Chemical & Natural Gas Engineering
*Corresponding author: Texas A&M University-Kingsville, Department of Chemical & Natural Gas Engineering, EC 303D, MSC 193, Kingsville, TX-78363-8202, USA. Email: Patrick.Mills@tamuk.edu

Abstract: COMSOL Multiphysics™ provides a powerful modeling platform for analysis of complex fluid dynamical systems. The objective of this paper is to develop a model for the fluid dynamics of a rotating cone pump based on the 3-D transient Navier-Stokes equations for an assumed geometry. The unsteady-state, incompressible fluid flow behavior in the annular region between the rotating cone and a stationary enclosure is analyzed under both laminar and turbulent flow conditions. COMSOL simulations have been performed to determine the fluid velocity and pressure profiles for various geometrical parameters and process conditions. The combined effects of angular rotational speed (Ω), cone height (H), and the ratio of outer to inner radius (κ) on the volumetric flowrate (Q) on pump performance have been systematically studied.

Keywords: rotating cone pump, laminar, turbulent, time-dependent, CFD.

1. Introduction

Velocity profiles in liquid films flowing over rotating conical surfaces are of considerable interest in advanced process manufacturing industries. The efficiency of important process equipment, such as spinning cone columns, fluid degassers, centrifugal disc atomizers, centrifugal film evaporators, and rotating packed-bed reactors, to name a few examples, is greatly influenced by the nature of the fluid velocity distributions. Conical pumps offer simple alternatives to conventional centrifugal pumps yet the analysis of pump performance has received minimal attention in the literature. Robust models that allow prediction of pump performance would be useful for guiding new designs for various applications versus using less efficient empirical trial-and-error approaches.

The analysis of fluid flow in a conical pump has some features that are similar to the flow between rotating coaxial cylinders. The flow between two coaxial cylinders in which the inner one rotating and the outer one is stationary, which is commonly referred to as Taylor–Couette flow (TCF), is of notable interest for various applications that occur in chemical engineering as well as in fundamental fluid mechanics. Special attention has been given to the analysis of Taylor vortices in basic flow and dynamic helical vortices in other flow regimes. It has been found the vortices occur in the direction toward the smaller radius in the case of basic flow. Conversely, helical vortices occur while toward the larger radius in other flow regimes. Moreover, unsteady helical vortices can coexist with unstable steady Taylor vortices [1].

The objective of this work is to develop a COMSOL Multiphysics™ model for the fluid dynamics in both the laminar and turbulent flow regimes using the 3-D transient Navier-Stokes Equations for an assumed conical pump geometry. Primary interests are focused on the flow field behavior, such as the evolutionary pattern of fluid streamlines, velocity profiles and pressure profiles with respect to Reynolds number, as well as their dependence on the pump operational and geometric parameters, such as the rotational speed (Ω), cone semi-angle (**α**), and ratio of outer to inner radii (κ) on pump performance.

2. Design of Rotating Cone Pump

A typical rotating cone geometry is shown Figure 2. It consists of a vertical, rotating inner cone with an upper radius R_0 = 15 mm, a rotating inner solid cone with a radius that is 70 to 90% of R_0, and a stationary outer cone. The semi-cone angle (α) was varied from 15^o to 45^o as part of a parametric study, and the gap between the rotating cone and stationary cone was maintained constant. At the exit side of the pump, the radii aspect ratio (κ) was varied from 0.7 to 0.9. The inlet (bottom end) is maintained at atmospheric pressure. The fluid column height (H) was varied from 10 to 35 mm. The working fluid is water. The fluid temperature is constant at 26^{o}C and the fluid viscosity is 1 cp.

Initial simulations were performed with no rotation at different flowrates to estimate the velocity profiles and flow regime. Thereafter, the inner cone was rotated using different (Ω) speeds from 3000 to 12000 rpm to evaluate the pump performance using different geometrical and process conditions.

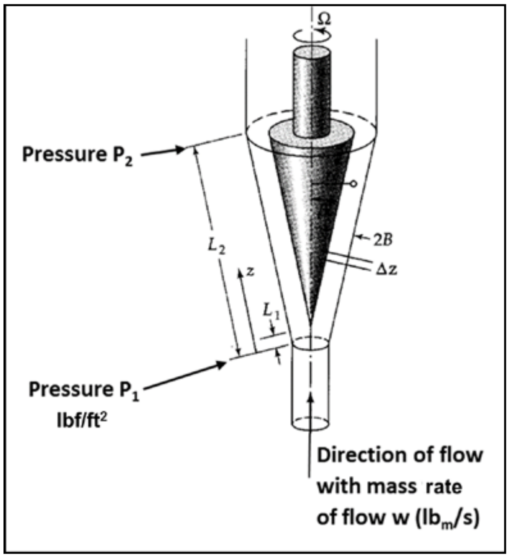

Figure 1. Schematic of the Rotating Cone Pump.

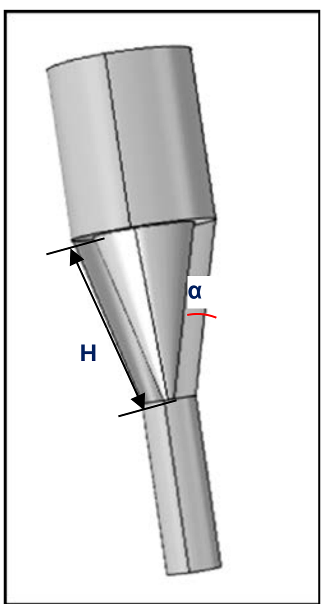

Figure 2. Geometry of the Rotating Cone Pump created in COMSOL.

3. Methodology

3.1 Model Assumption

The current model is based upon homogeneous single-phase fluid flow. Both approximate 1-D and more rigorous 3-D models were used. The model equations for the 1-D steady-state problem were obtained by first assuming $v_r = v_z = 0$ and $v_\theta = v_\theta(r)$, and then simplifying the equation of continuity and momentum transport equations in polar coordinates. These results are not shown here [1], but are based upon laminar flow. They were used to obtain initial estimates of the velocity profiles for the COMSOL 3-D solution.

The steady-state or stationary solutions for the fluid velocity profiles in 3-D were obtained by integration of the time-dependent solution. The flow domain or control volume for the 3-D model was restricted to the region between the rotating inner cone and the stationary outer enclosure. When this domain is discretized using the finite element method, the components of the time-dependent fluid velocity vector are **U** = [U_x U_y U_z]. The fluid is assumed to be water which is both incompressible and exhibits Newtonian behavior. The fluid properties density (ρ) and viscosity (μ) are assumed to be constant. Both laminar-flow regime and turbulent flow regime were analyzed. For the latter, the (k-ε) model was used.

3.2 Model equations

Conservation of mass and momentum principles are applied to the system which leads to the equation of continuity and momentum transport equations in vector form, which are summarized below (Bird *et al*., 2007). These are also similar to forms used in COMSOL Multiphysics.

Equation of Continuity

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{u}) = 0$$

Momentum (Navier-Stokes) equation

$$\frac{\partial}{\partial t}\rho \boldsymbol{u} = -[\nabla \cdot \rho \boldsymbol{u} \boldsymbol{u}] - \nabla p - [\nabla \cdot \overline{\overline{\boldsymbol{\tau}}}] + \rho \mathrm{g} = 0$$

Turbulent Flow (k-ε Model)

The k-ε equations are derived from the application of a rigorous statistical technique (Renormalization Group Method) to the instantaneous Navier-Stokes equations. The k-ε model focuses on the mechanisms that affect the turbulent kinetic energy (per unit mass) k.

<u>Equation of Continuity</u>

$$\frac{\partial \rho}{\partial t} + \rho \nabla \cdot \boldsymbol{u} = 0$$

<u>Momentum Transport Equation</u>

$$\rho \frac{\partial \boldsymbol{u}}{\partial t} + \rho(\boldsymbol{u} \cdot \nabla)\boldsymbol{u} = \nabla \cdot \left[-p\boldsymbol{I} + (\mu + \mu_T)\left(\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T\right) - \frac{2}{3}\rho k \boldsymbol{I} \right] + \boldsymbol{F}$$

<u>k-ε Model Equations</u>

Turbulent kinetic energy, k, is given by:

$$\rho \frac{\partial k}{\partial t} + \rho(\boldsymbol{u} \cdot \nabla)k = \nabla \cdot \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \nabla k \right] + P_k - \rho \epsilon$$

Turbulent dissipation, ε, is given by:

$$\rho \frac{\partial \epsilon}{\partial t} + \rho(\boldsymbol{u} \cdot \nabla)\epsilon = \nabla \cdot \left[\left(\mu + \frac{\mu_T}{\sigma_\epsilon} \right) \nabla \epsilon \right] + C_{c1} \frac{\epsilon}{k} P_k - C_{c2} \rho \frac{\epsilon^2}{k}$$

Turbulent viscosity is modelled by:

$$\mu_T = \rho C_\mu \frac{k^2}{\epsilon}$$

Production of turbulent kinetic energy defined as:

$$P_k = \mu_T\left[\nabla \boldsymbol{u} : \left(\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T\right)\right] - \frac{2}{3}\rho k \nabla \cdot \boldsymbol{u}$$

In the above equations, **u** is the flow velocity, p is the pressure, μ_T is the turbulent viscocity, k is the turbulent kinetic energy, ε is the turbulent dissipation energy, β is the coeffcient of thermal expansion, P_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, and σ_k, σ_ε, C_{c1}, $C_{c2,}$ C_μ are model constants determined by an experiment.

4. Use of COMSOL Multiphysics

COMSOL Multiphysics 5.0 is used to numerically solve the Navier-Stokes equations. This CFD package uses the finite volume method and supports structured, mapped and unstructured grids. It enables the use of different discretization schemes and solution algorithms, together with various types of boundary conditions.

An unstructured grid with triangular elements was used. The mesh size was varied by the means of adjusting different meshing parameters, including the maximum element size, minimum element size, maximum element growth rate, curvature factor and resolution of narrow regions. The grid had refined boundary layer regions along the walls to capture flow patterns in close vicinity to the wall.

To conserve computational time and to assess the effect of the number of elements on the error, meshes having a different number of elements (48,000, 92,000 and 124,000) were tested: Figure 3 shows that the pressure head computed from these three different meshes is essentially the same so that the results are not grid-dependent.

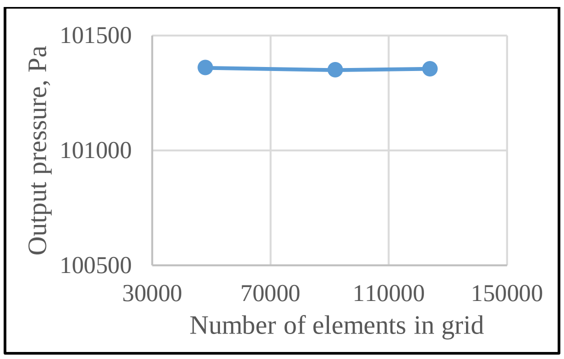

Figure 3. Dependence of the calculated fluid exit pressure on the number of elements.

5. Results and discussion

In this section, both the steady-state and transient performance of the rotating cone pump for different geometries is presented. Figures 4 and 5 show velocity magnitude distributions along the plane x = 0 and streamlines for cone semi-angles of 15^o and 45^o, respectively. It can be seen that the velocity magnitude does not exceed 0.1 which corresponds to a Reynold number = 1.5. In this range of Reynolds numbers and pump dimensions, centrifugal forces are not the main ones that are driving the flow. Instead, the flow is mainly driven by viscous forces [3]. It is also seen that the velocity patterns change from being mainly in the axial direction at the inlet to the angular direction in the angled region. After exiting the angled region of the cone, the velocity undergoes a transition from an angular-controlled pattern to a combination of angular and axial behavior.

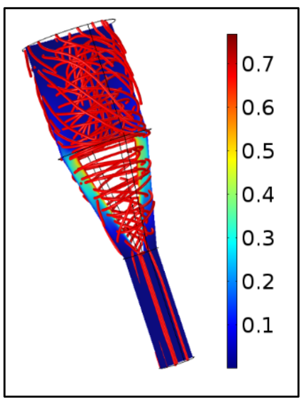

Figure 4. Velocity profiles for a semi angle of 12°, Q = 1 ml/s

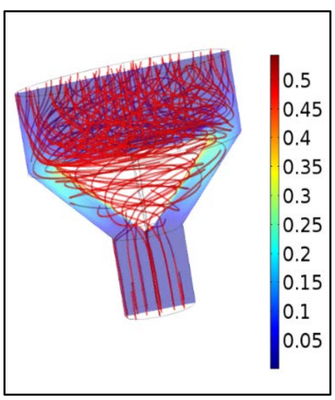

Figure 5. Velocity profiles for a semi angle of 45°, Q = 1 ml/s

The fluid pressure profiles are shown in Figures 6 and 7, which correspond to the velocity profiles shown above in Figures 4 and 5, respectively. The pressure profiles show that the hydraulic head of the pump is a weak function of the cone semi-angle and height of the rotating cone. On the contrary, as might be expected, the frequency of rotation has a pronounced effect on the pump head. However, even for a maximum rotational speed of 12,000 RPM, the hydrodynamic head does not exceed 135 Pa.

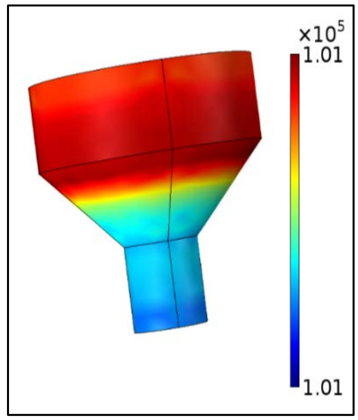

Figure 6. Pressure field for a semi angle of 45°, Q = 1 ml/s

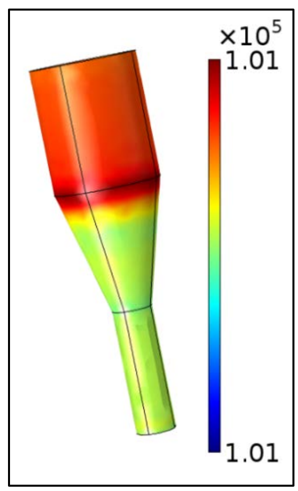

Figure 7. Pressure field for a semi angle of 12°, Q = 1 ml/s

The head curve has been acquired for different values of angular speed. For that purpose, model has been run repeatedly for each volumetric flow rate – angular speed pair and the calculation goal was outlet pressure. The results are shown in Figure 8. The outlet pressure decreases almost linearly with increasing volumetric flowrate. The pump head is nearly proportional to the square of rotational speed.

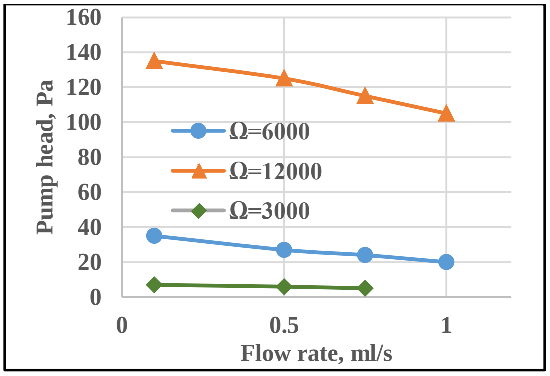

Figure 8. Pump head for different RPM values

An pproximate solution for rotating cone pump performance is available in theopen literature [1]. This solution uses the equations for slit flow to the rotating cone problem. However, it still utilizes all of the asserted approximations, *e.g.*, the flow is laminar with curvature and entrance effects being eglected.

Mass flowrate for the slit approximation model

$$w = \frac{4\pi B^3 \rho \sin\alpha}{3\mu \ln({}^{L_2}/_{L_1})}\left[(p_1 - p_2) + (\frac{1}{8}\rho\Omega^2 sin^2\alpha)\cdot(L_2^2 - L_1^2)\right]$$

In the above equaiton, L_1,L_2 are the heights corresponding to pressures p_1 and p_2 respectively.

This above formula implies linear dependency of volumetric flowrate (assuming constant density) on pressure drop. Comparison of predicted values and those computed acquired from CFD simulations are shown in Figure 9. The approximation overpredicts the pump head by a factor of two. One of the sources of this error may be attributed to the observation that the centrifugal force is not the primary one which generates the flowto in this particular scale of flow geometry.

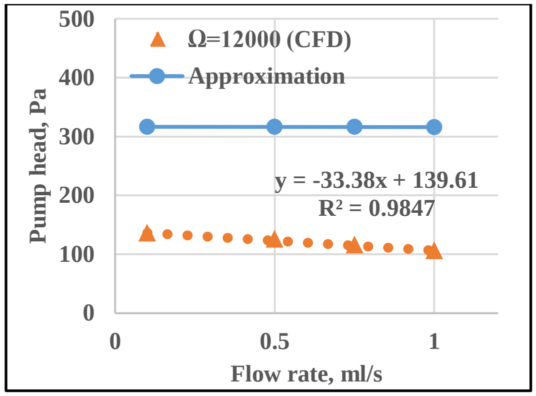

Figure 9. Comparison between the predicted versus calculated head curve for Ω = 12000 RPM

The fluid viscosity, along with fluid density will also have an effect on the pressure head generated by the pump. Two head curves for different liquids at same operating conditions are compared in Figure 10. Water, being more dense (ρ = 999.66 kg/m^3) and more viscous (μ = 1.01 cP), than diethyl ether (ρ = 713.58 kg/ m^3, μ = 0.24 cP), creates bigger pressure difference in all range of flow rates. This result is opposite of what is expected from conventional centrifugal pump. This can be explained by the predominance of viscous forces acting on fluid in micro-sized pumps. However, trends for both liquids are similar and in agreement with ones for centrifugal pumps.

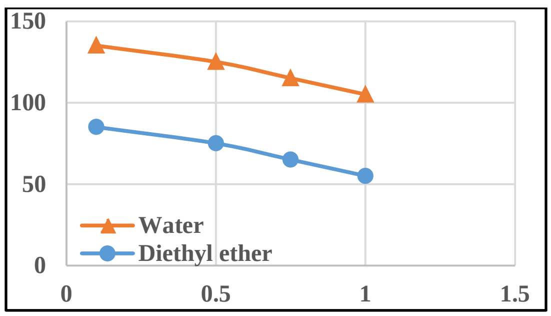

Figure 10. Head curve for diethyl ether and water at Ω = 12000 RPM

6. Conclusions

This paper analyzed the performance of a rotating cone micro pump. Solutions for the Navier-Stokes equations have been obtained for different geometries and boundary conditions.

The rotating cone pump is capable of creating comparatively large throughputs; however it cannot create heads comparable to conventional centrifugal pumps of different designs. For applications where small pressure heads are satisfactory, both the simplicity and performance of the pump will make it a good choice.

7. Nomenclature

B - Slit width, mm
H - Height of the cone, mm
P_1 - pressure at inlet, pa
P_2 - pressure at outlet, pa
Ri - Radius of the inner cone, mm
Ro - Radius of the outer cone, mm
k - Turbulent kinetic energy
T - Temperature of fluid, ^{o}C
u - Flow velocity, m/s

Greek denotes

Ω - Rotational speed of cone, mm
α - Semi-cone angle, ^{o}C
μ - Viscosity of fluid, cP
μ_T - Turbulent viscosity
ρ - Density of fluid, g/cm^3
ε - Turbulent dissipation energy
P_k - Turbulent kinetic energy due to mean velocity gradients

8. References

1. Bird, R.B, Stewart, W.E & Lightfoot, E.N. *Transport Phenomena*. (2nd ed.), John Wiley & Sons, Inc, New York (2007)
2. Xiaofei Xu, Pu Wena, Lanxi Xu, Dapeng Cao, “Occurrence of Taylor vortices in the flow between two rotating conical cylinders,” *Commun Nonlinear Sci Numer Simulat* **15** 1228–1239 (2010)
3. Gad-el-Hak, M., “The Fluids Mechanics of Microdevices—The Freeman Scholar Lecture,” *ASME J. Fluids Eng.*, **121**, 5–33 (1999)
4. Dijk *et al*. “Hydrodynamics of liquid flow in a rotating cone,” *International Journal of Numerical Methods for Heat & Fluid Flow*, **11(5)**, 386-401. (2001).
5. Bataineh, K.M. and Taamneh, Y, “Novel rotating cone viscous micro pump.” Int. J. Engineering Systems Modelling and Simulation, Vol. 5, No. 4, pp.188–196. 2013.