International Journal of Heat and Fluid Flow 42 (2013) 289–295

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Fluid Flow

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

A three dimensional model of a vane rheometer

Behzad Nazari [a,*], Ramin Heidari Moghaddam [b], Douglas Bousfield [a]

[a] Department of Chemical and Biological Engineering, University of Maine, Orono, ME 04460 USA
[b] Department of Chemical Engineering, Yasouj University, Yasouj 75914-353, Iran

ARTICLE INFO

Article history:
Received 4 November 2012
Received in revised form 19 February 2013
Accepted 21 February 2013
Available online 7 May 2013

Keywords:
Vane rheometer
Finite element
3-Dimensional flow
Newtonian fluid

ABSTRACT

Vane type geometries are often used in rheometers to avoid slippage between the sample and the fixtures. While yield stress and other rheological properties can be obtained with this geometry, a complete analysis of this complex flow field is lacking in the literature. In this work, a finite element method is used to calculate the isothermal flow parameters in a vane geometry. The method solves the mass and momentum continuity equations to obtain velocity, pressure and then stress fields. Using the total stress numerical data, we calculated the torque applied on solid surfaces. The validity of the computational model was established by comparing the results to experimental results of shaft torque at different angular velocities. The conditions where inertial terms become important and the linear relationship between torque and stress are quantified with dimensionless groups. The accuracy of a two dimensional analysis is compared to the three dimensional results.

1. Introduction

A vane rheometer consists of an impeller rotating in a baffle-cylinder geometry. This geometry is used to eliminate slip between the sample and the rheometer geometry. In the case of complicated fluids such as concentrated suspensions or polymer systems, it allows for an accurate and direct measurement of yield stress (Lecompte et al., 2012; Dong et al., 2009; Saaka et al., 2001; Wierenga et al., 1998). However, due to the complex flow geometry of vane, it is a significant challenge to relate the measured quantities such as shaft torque and angular velocity to other rheological properties such as shear thinning and thickening behaviors (Barnes and Nguyen, 2001; Barnes and Carnali, 1990). Several researchers (Yan and James, 1997; Keentok et al., 1985; Griffiths and Lane, 1990; Keshtkar et al., 2009), assuming no-secondary flow at low shear rates for the vane, have used the vane for measuring rheological properties, but it is clear that secondary flow regimes exist at high rotational rates. Even though the flow regimes in vane geometries have been studied by a number of researchers (Barnes and Nguyen, 2001; Martinez-Padilla and Quemada, 2007; Martinez-Padilla and Rivera-Vargas, 2006), due to the lack of an analytical model, the interpretation of the experimental results is not quantitative. Therefore, a numerical simulation that solves the velocity field is needed. Using numerical methods such as finite difference, finite element (FEM), and finite volume, conservation equations can be solved to obtain velocity, pressure, and other flow parameters. A number of researchers (Zhua et al., 2010; Sherwood, 2008) have studied a two-dimensional fluid flow in the vane geometry, but regarding end effects and vertical-direction flow, a three-dimensional model is needed. Baravian et al. (2002), Estellé et al. (2008), and Estellé and Lanos (2008) have used equivalent coaxial cylinder (Couette analogy) for calculating shear rate and stress from angular velocity and torque. Ovarlez et al. (2011) have studied the validity of Couette analogy using some magnetic resonance imaging measurements. Using velocity profiles, they suggested that there is a deviation from cylindrical symmetry. In this paper, a finite element model for the flow of a Newtonian fluid in vane geometry is proposed. The three-dimensional mass and momentum continuity equations are solved to obtain the velocity and pressure profiles throughout the geometry. By surface integrations of the total stress numerical data, the torques applied on the solid surfaces are predicted. The calculated torque data is compared to the experimental torque–angular velocity data obtained by the vane rheometer. A good comparison is obtained for two different fluids. We also studied the validity of Couette analogy in the present case.

2. Geometry and the fluid

The geometry of the vane consists of blades, shaft, cylinder, and baffles shown in Fig. 1. The fluids studied in this paper were standard Newtonian fluids (Brookfield) with viscosities 98 and 12,000 mPa s at 25 °C.

3. Theoretical background

The steady conservation of mass and momentum for any fluid is given in vector form as

* Corresponding author.
E-mail address: behzad.nazari@maine.edu (B. Nazari).

0142-727X/$ - see front matter © 2013 Elsevier Inc. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatfluidflow.2013.02.012

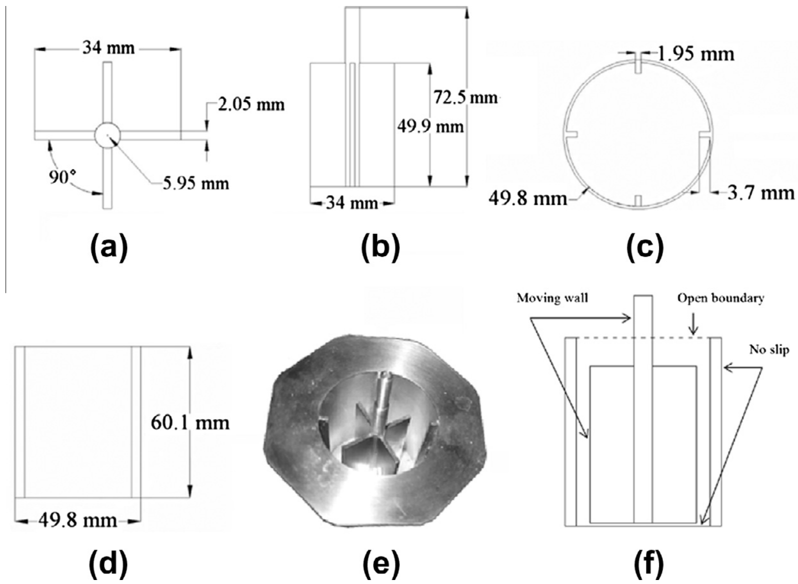

Fig. 1. The geometry: (a) vane top view, (b) vane front view, (c) container top view, (d) container front view, (e) the whole geometry, and (f) boundary condition set-up.

$$\nabla(\rho u) = 0 \tag{1}$$

$$\boldsymbol{\rho}(\boldsymbol{u}\nabla)\boldsymbol{u} = \nabla[-pI + \boldsymbol{\tau}] + \boldsymbol{F} \tag{2}$$

where $\boldsymbol{u}$ is the velocity vector, $\boldsymbol{\rho}$ is the fluid density, $\boldsymbol{p}$ is pressure, and $\boldsymbol{\tau}$ is the extra stress. $\boldsymbol{F}$ is the body force vector. To solve these equations, some constitutive relations are needed. A starting point is the expression for a Newtonian fluid given as

$$\boldsymbol{\tau} = \mu(\nabla\boldsymbol{u} + (\nabla\boldsymbol{u})^T) + \frac{2}{3}\mu(\nabla\cdot\boldsymbol{u})I \tag{3}$$

The dynamic viscosity $\boldsymbol{\mu}$ does not depend on the velocity field and is a material constant. Assuming incompressible flow, we can consider density ρ as a constant as well. With these two assumptions, the Navier–Stokes equations are formed as

$$\nabla u = 0 \tag{4}$$

$$\rho(u\nabla)u = \nabla[-pI + \mu(\nabla u + (\nabla u)^T)] + F \tag{5}$$

These equations are applicable for isothermal study of incompressible Newtonian fluid flow. The velocity components of the boundary surface $\boldsymbol{u_x}$ and $\boldsymbol{u_y}$ for each point on the impeller blades or shaft relate to the angular rotation rate $\boldsymbol{\Omega}$ as

$$u_x = -r\Omega\sin\theta = \Omega y \tag{6}$$

$$u_y = r\Omega\cos\theta = \Omega x \tag{7}$$

Which $\boldsymbol{x}$, $\boldsymbol{y}$, and $\boldsymbol{z}$ are position variables and $\boldsymbol{r}$ is lever arm (the distance of each point from the shaft axis). For baffles and cylinder walls, we assume no-slip boundary conditions. At the free surface of the fluid, based on the low viscosity of air compared to the liquid fluid viscosity, we neglect shear stress at liquid–gas interphase:

$$[-pI + \mu(\nabla\boldsymbol{u} + (\nabla\boldsymbol{u})^T)]n = -f_0\boldsymbol{n} \tag{8}$$

where $\boldsymbol{n}$ is the normal vector of the surface and $\boldsymbol{f_0}$ is the force applied to unit area of the surface (atmospheric pressure, 101 kPa). In Fig. 1f, a schematic of the boundary conditions used in this study is shown. We also set up a gravity induced body force $\boldsymbol{F}$ with the value 9800 N/m^3 in $-\boldsymbol{z}$ direction. The pressure condition at the surface and the body force do not influence the results.A commercial finite element package is used to solve the equations (COMSOL Multiphysics 4.1). Free mesh methods are used to discretize the volume. The mesh elements are mainly tetrahedral and triangular as well. However, top open boundary surface includes quadrilateral elements too. In Fig. 2, we have shown the mesh elements. There are 254,167 elements with a mesh average size of 0.00212 m in the model mesh.

3.1. Calculating torque from stress profile

The program outputs the velocity and pressure fields, but to calculate the torque, the total stresses acting on the walls or the stress acting on the impeller needs to be calculated. The geometry was drawn in a way that the shaft axis is in $\boldsymbol{z}$-direction and the coordinates origin is at the center of container base. The blades, shaft, and the cylinder will be divided into four quarters as follows as shown in Fig. 3. The torques are calculated for the blade in two positions, as shown in Fig. 3 and when the blade match up with the baffles.

The lever arm $\boldsymbol{R}$ and the angle relative to the shaft at any $\boldsymbol{x}$ and $\boldsymbol{y}$ position is then

$$R = \sqrt{x^2 + y^2}, \quad \sin\theta = \frac{y}{r}, \quad \textit{and} \ \cos\theta = \frac{x}{r} \tag{9}$$

For a specific surface element, a force balance gives, as demonstrated in Fig. 3, the component of the total stress as it affects the surface force and the torque. The force times the lever arm gives rise to a torque.Which total stress is defined as

$$\pi = pI + \tau \tag{10}$$

Regardless to the sign of $\boldsymbol{\pi_x}$ and $\boldsymbol{\pi_y}$, we have:

$$|dT_x| = |\pi_x * dA * \sin\theta * r| = |\pi_x \cdot y \cdot dA| \tag{11}$$

$$|dT_y| = |\pi_y * dA * \cos\theta * r| = |\pi_y \cdot x \cdot dA| \tag{12}$$

To calculate the total torque, we just need to know $|\boldsymbol{T_x}|$, $|\boldsymbol{T_y}|$, and signs of them in each quarter. The sign of the torques depends on the signs of $\boldsymbol{x}$, $\boldsymbol{y}$ (or $\boldsymbol{\cos\theta}$ and $\boldsymbol{\sin\theta}$, respectively), and total stresses. First, depending on the fact that in which quarter they are, we make $\boldsymbol{x}$ and $\boldsymbol{y}$ positive as follows:

Quarter 1	$+x$	$+y$
Quarter 2	$-x$	$+y$
Quarter 3	$-x$	$-y$
Quarter 4	$+x$	$-y$