Available online at www.sciencedirect.com
ScienceDirect
International Journal of Engineering Science 44 (2006) 915–926
International Journal of Engineering Science
www.elsevier.com/locate/ijengsci

Unsteady shear flow of fluids with pressure-dependent viscosity

Mehrdad Massoudi *, Tran X. Phuoc

US Department of Energy, National Energy Technology Laboratory, P.O. Box 10940, Pittsburgh, PA 15236, USA

Received 15 February 2006; received in revised form 16 May 2006; accepted 18 May 2006

Abstract

We study the unsteady shear flow of fluids with pressure-dependent viscosity, situated between two parallel horizontal plates with the upper plate moving while the flow is subjected to an oscillating pressure gradient. The dimensionless form of the momentum equation is solved numerically using a central difference approximation for the spatial derivative terms and a forward difference approximation for the time derivative term. In addition to providing the velocity profiles at the midsection between the two plates, the values of shear stress at the lower (stationary) plate for various values of the dimensionless numbers are also plotted.

Keywords: Non-Newtonian fluids; Unsteady shear flow; Pulsating pressure; Pressure-dependent viscosity; High pressure combustion; Coal slurries

1. Introduction

Pressure plays an important role in the design, operation, maintenance, control, and efficiency of power plants. For example, in high-intensity combustors, swirl is commonly used to improve flame stability, to produce higher rates of entrainment of the ambient fluid and fast mixing. In these practical flows, pulverized coal and air are injected tangentially toward the wall of the combustor to cause cyclonic action. Particles have a wide range of sizes and densities. The removal of solids from these high-pressure high-temperature environments is a serious practical issue. It has been shown that cyclone cleaning systems offer a viable solution to some of these difficulties (see [10]). Furthermore, in pressurized circulating fluidized bed boilers, for example, the effect of high pressure on char combustion has been shown to be significant (see [13]). High pressure combustion of co-firing coal and biomass in CO_2/O_2 environments is another desirable process because emissions of CO_2, SO_2, and NO_x in the exhaust gases are reduced significantly. This combustion technique, therefore, offers a unique opportunity to reduce CO_2 emissions from coal combustion that does not require CO_2 separation, storage, transport and disposal which are very expensive and technically difficult. In addition, the mechanism by which NO_x and SO_x emissions can be reduced has not been well established especially at high

* Corresponding author. Tel./fax: +1 412 386 4975.
E-mail address: Massoudi@netl.doe.gov (M. Massoudi).

0020-7225/$ - see front matter © 2006 Published by Elsevier Ltd.
doi:10.1016/j.ijengsci.2006.05.010

pressures. Even though the data from large-scale testing are more practical, these tests are expensive, and the instrumentation is too difficult and complicated to yield reliable results. In addition, due to the large amount of fuel used in these experiments and the difficulty in replacing devices such as fuel injection system, fuel delivery system, etc., large-scale testing for different conditions on injection rates, injection locations, fuel compositions etc., are very limited.

In recent years, with faster and more efficient computers and more accurate numerical schemes, many CFD (computational fluid dynamics) codes have been developed to study multiphase flows. These codes, in many instances, have been used as verification tools or a means to evaluate the existing experiments. In some instances, the true potential of these codes, which is their predictive capabilities, has also been utilized. An important issue is the physical models which are embedded in these codes. As more sophisticated and more complex materials, such as fibers, composites, nanoparticles, etc., are being used, the need to improve and derive more accurate constitutive relations also increases. In mechanics, especially in fluid mechanics, exact solutions to the governing equations of motion are important for many reasons. In addition to providing and enhancing the field with aesthetic beauty of closed form solutions, exact (analytical) solutions also serve as standards or measures whereby the computational/numerical solutions (for complicated geometries and flow conditions) can be tested for accuracy and effectiveness. As few as these exact solutions are for the steady flows of the Navier–Stokes fluid (see [36]), there are even fewer exact solutions for the unsteady flows of these fluids (see [6–8]). Unsteady flows of fluids occur in nature and in many industrial applications; the unsteadiness can be due to a variety of reasons: the unsteady motion of the boundaries, application of body forces, impulsive motion or sudden acceleration of boundaries or the body, fluctuating nature of the flow as in turbulence, application of unsteady forces (or stresses) or displacement at the boundaries. Another class of interesting problems is the flow of blood in (tapered) arteries (see [15,16,22,19]) whereby the changes in the geometry of the artery, modeled as an elastic flexible tube, can also cause unsteady flow.

The difficulties in obtaining exact solutions for the unsteady flows of non-Newtonian fluids or complex fluids modeled as mixtures are exacerbated due to the non-linearities in the constitutive relations and the interactions among the phases. Ting [33], using complex analysis methods, studied unsteady flows of second grade fluids. Rajagopal [24] established exact solutions for a class of unsteady unidirectional flows of an incompressible second grade fluid, while ignoring the inertial effects. Later Bandelli and Rajagopal [3] proved a general theorem on start-up flows for second grade fluids (see [2] for the inclusion of thermal effects in these studies, Bandelli et al. [4] for some unsteady flows of second grade fluids, and Baris [5] for the unsteady solutions of a binary mixture of Newtonian fluids). One of the main reasons for the success of the Navier–Stokes constitutive relation, in addition to its elegance and compactness, is that the stress tensor **T** is explicitly described in terms of **D** (the symmetric part of the velocity gradient) and other kinematical variables. This means that once the solutions to the problem, whether using numerical techniques or analytical methods represented in terms of velocity and pressure fields, are obtained, the stress at any point can be determined uniquely from the constitutive relation. This success is also shared by the power-law models and the fluids of differential type (for example, the grade fluids), and a host of other engineering models. Furthermore, from a computational point of view, it is much easier and less cumbersome to solve the equations for the explicit models. There are, however, cases [such as Oldroyd type fluids and other rate-dependent models] whereby it is not possible to express **T** explicitly in terms of **D** and other kinematical variables. For such cases, one must resort to implicit theories, for example, of the type (see [27])

$$\mathbf{f}(\mathbf{T}, \mathbf{D}, \theta) = \mathbf{0} \tag{1}$$

where θ is the temperature. In recent years, Rajagopal [25–27] has provided a rigorous methodology where constitutive relations for a class of fluids whose viscosity depends on pressure and shear rate can be obtained. These types of fluids are encountered, for example, in lubrication industry where the fluid is under high pressure [31], and in fact this phenomenon was recognized by Stokes as early as 1845 [30]. With all these studies, very little is known or presented in the literature about the unsteady flows of fluids with pressure-dependent viscosities.

In this paper, the unsteady shear flow of fluids with pressure-dependent viscosity, situated between two parallel horizontal plates with the upper plate moving while the flow is subjected to an oscillating pressure gradient is studied (see Fig. 1). The dimensionless form of the momentum equation is solved numerically using a