Available online at www.sciencedirect.com
ScienceDirect
Journal of Computational and Applied Mathematics 203 (2007) 444–460
JOURNAL OF COMPUTATIONAL AND APPLIED MATHEMATICS
www.elsevier.com/locate/cam

Computation of dilute two-phase flow in a pump

S.B. Hazra[a,*], K. Steiner[b]

[a]Department of Mathematics, University of Trier, 54286 Trier, Germany
[b]Fraunhofer Institut für Techno- und Wirtschaftsmathematik (ITWM), D-67663 Kaiserslautern, Germany

Received 30 November 2004; received in revised form 17 August 2005

Abstract

This paper is a report on a joint project between academia and industry which is concerned with computation of dilute two-phase flow through a pump in turbulent condition. The flow field for the continuous phase is computed using the Reynolds averaged Navier–Stokes equations together with mixing length turbulence modeling. The dispersed phase is treated using the Lagrangian approach by tracking it's trajectory along which the information is passed. It is found that the bubbles and small solid particles flow out of the chamber (between the rotating impeller and the casing wall) with the conveying fluid. The solid particles of relatively bigger sizes accumulate at the low pressure zones near the cashing wall or the rotating shaft.
© 2006 Elsevier B.V. All rights reserved.

Keywords: Continuous phase; Dispersed phase; Navier–Stokes equations; Turbulence modeling; Lagrangian simulation; Steady viscous flow

1. Introduction

Flow of bubbles and solid particles is of fundamental importance in many natural, physical and industrial processes. In this study we simulate numerically such flow behavior inside a turbomachinery (in particular, inside a single stage centrifugal pump). During the pumping process multiphase flow (mixture of liquid, solid particles and/or gas bubbles) may occur through the chamber between the casing wall and the rotating impeller. Due to difference in densities of the bubbles or solid particles and the conveying fluid that is being pumped, the bubbles or solid particles move differently in the flow. Normally, the particles move towards the low-pressure zone in the chamber. The performance of the pump goes down with the increase of particles or gas bubbles in the chamber. The '*mechanical seals*' between rotating shaft and casing wall are wet by some fluid causing the friction to be reduced. If the bubbles accumulate around it, the seal becomes dry and this leads to it's destruction. Then the pump fails to function.

In all the phenomena and processes related to particles/bubbles, there is relative motion between particles on one hand, and surrounding fluid on the other. In many cases, transfer of mass and/or heat is also of importance. In our study, by the word 'particle' we mean a self-contained body with maximum dimensions about 0.5 μm to 1 cm, separated from the surrounding medium by a recognizable interface. The material forming the particle will be termed

* Corresponding author.
E-mail addresses: hazra@uni-trier.de (S.B. Hazra), steiner@itwm.fhg.de (K. Steiner).

0377-0427/$ - see front matter © 2006 Elsevier B.V. All rights reserved.
doi:10.1016/j.cam.2006.04.011

as ‘dispersed phase’. We refer to particles whose dispersed phase is composed of solid matters as ‘solid particles’ and those composed of gas as ‘bubbles’. These particles can be of different shapes and sizes.

Different models used to study such two-phase flows are given in [7,3,5,6] and in references therein. We compute here dilute liquid–particle flows considering one way coupling. In such flows, the flow of the dispersed phase is controlled by the surrounding conveying fluid, unlike the dense flows where it is controlled mainly by the particle–particle collisions. We neglect the effect of other forces except the drag force in this study. We use a Lagrangian description of the model to simulate numerically the dispersed phase. This model requires specification of the flow field of the conveying fluid, which is obtained numerically. The particle trajectory is computed by integrating the particle equations of motion as the particle proceeds through the chamber. For simplicity of description of bubbles in quasi-2D flow field, we consider the shape of the bubbles to be spherical-cap which is a valid approximation at high Reynolds number [8].

In the next section we describe the governing equations and initial/boundary conditions of the continuous phase. In Section 3 we briefly discuss about the mixing-length turbulence model that is used in our study. Section 4 presents the governing equations of the dispersed phase whose derivation (specially for gas bubbles) is presented in the Appendix (appears at the end of this paper). Section 5 presents a brief description of method of solution, the results and discussions. We make our conclusions in Section 6.

2. Governing equations and boundary conditions for the continuous phase

We study the flow between a rotating impeller and a casing wall of a (single stage) centrifugal pump. Such flow region can be modeled by the geometry between fixed and rotating disks (Fig. 1). The flow field can be described by the incompressible Navier–Stokes equations [14]. Here temperature variance is neglected and therefore the energy equation is not solved for temperature. We are interested in the steady-state solution of the continuous phase which does not depend on time. The dispersed phase is described by Lagrangian formulation, which means the path of the particles/bubbles is tracked by integrating the evolution equations in time.

Since the flow is turbulent for the Reynolds numbers (defined by $Re = r_{\max}^2\omega/\nu$, where ω is the angular velocity of the rotating shaft or the rotating impeller, $\nu = \mu/\rho$ is the kinematic viscosity) in the range used in our computations (is of the order of 10^6), we solve the Reynolds averaged Navier–Stokes equations together with Mixing Length turbulence modeling. Cylindrical polar coordinate system is considered to describe the problem mathematically. In case of rotational symmetry, $\vec{V}(r, z, \phi) = \vec{V}(r, z)$ only, where $\vec{V}$ is the velocity vector with components u, w and v in r (radial), z (axial) and ϕ (tangential) directions, respectively. In that case the Reynolds averaged Navier–Stokes

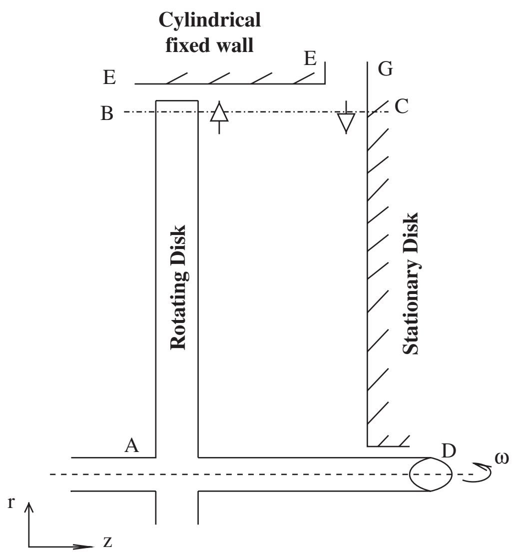

Fig. 1. The geometry of the computational domain.

Download English Version:

https://daneshyari.com/en/article/4642643

Download Persian Version:

https://daneshyari.com/article/4642643

Daneshyari.com