Available online at www.sciencedirect.com

SCIENCE DIRECT®

Powder Technology 154 (2005) 73 – 82

POWDER TECHNOLOGY

www.elsevier.com/locate/powtec

Numerical study of the rotational phase separator sealing impeller

J.G.M. Kuerten*, H.P. van Kemenade, J.J.H. Brouwers

Department of Mechanical Engineering, Technische Universiteit Eindhoven, P.O. Box 513, NL-5600 MB Eindhoven, The Netherlands

Available online 31 May 2005

Abstract

The rotational phase separator is a patented technique for separating heavy or light phases and particulate matter from fluids by centrifugation. This technique is used in areas ranging from the treatment in fluids in industrial processes to the filtering of air to protect men from respiratory allergic reactions. In a particular design to separate particles from gas flows the rotating filter element is combined with a cyclone type housing. For this application a sealing impeller is required to prevent uncleaned gas flowing alongside the filter element to the outflow. Since the design of the impeller is critical to the performance of the apparatus, the flow phenomena involved are studied numerically. The results are used to formulate a one-dimensional model for easy use during the design process.

Keywords: Particle separation; Design; Computational fluid dynamics

1. Introduction

The rotating particle separator is a novel and patented technique initially intended for separating solid and/or liquid particles of 0.1 μm and larger from gasses [1,2]. The core component is the rotating filter element (see Fig. 1) which consists of a multitude of axially oriented channels which rotate as a whole around a common axis. Particles or droplets in the fluid, which flows in a laminar motion through the channels, are centrifuged to the outer walls of each individual channel while the purified fluid leaves at the exit. The rotating particle separator is well developed for cleaning gasses in both domestic and industrial applications [3,4]. A typical design, intended for use in biomass gas cleaning systems, is depicted in Fig. 2. The filter element is mounted on a cyclone pre-separator, where the largest particles are separated. Due to the centrifugal force the particles, which are heavier than the gas in which they are entrained, are forced in the direction of the outer walls of the channels. When a particle reaches a channel wall, it will remain there due to attractive Van der Waals or surface forces. Only particles exceeding a certain diameter reach the wall, smaller particles remain in the gas flow. Due to viscosity there is a pressure drop over the channels, which increases with increasing contamination of the channels with particles. On the other hand there is a spacing between the rotating filter wall and the stationary cyclone casing. Therefore, a leak flow of uncleaned gas will occur from the cyclone along the filter to the outflow. In order to prevent this, an impeller can be used in the spacing between the filter wall and the stationary casing. In the ideal case a small leak flow of clean gas from the area above the filter back to the cyclone results in a pressure difference over the impeller which just equals the pressure drop over the filter channels. In order to design a well-operating impeller a one-dimensional model for the flow in the impeller, which is able to predict the pressure difference over the impeller as a function of the leak flow, should be formulated. Therefore, in this paper we present a numerical model of the flow in the impeller and use the results to formulate a simple one-dimensional analytical theory for this flow.

In the next section the numerical model is described. Subsequently, the basic equations for the one-dimensional

* Corresponding author. Tel.: +31 40 247 2362; fax: +31 40 2475399.
E-mail address: j.g.m.kuerten@tue.nl (J.G.M. Kuerten).

0032-5910/$ - see front matter © 2005 Elsevier B.V. All rights reserved.
doi:10.1016/j.powtec.2005.04.039

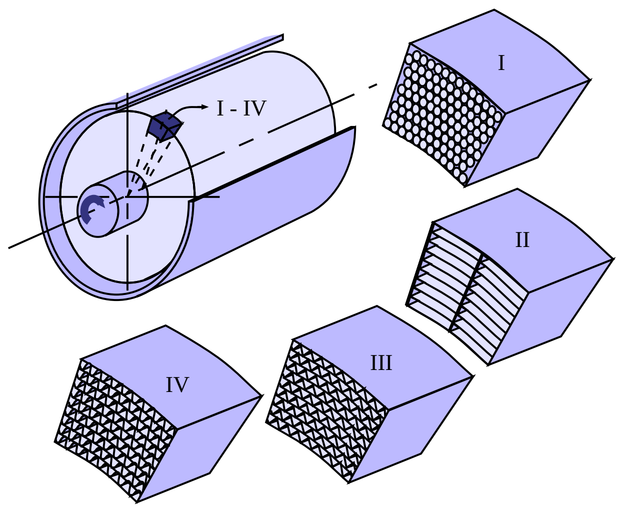

Fig. 1. The rotating filter element.

analytical model is given. In Section 4 results are presented and used to formulate the one-dimensional theory. Finally, in Section 5 the conclusions are stated.

2. Numerical model

The impeller consists of a number of radial vanes mounted on a circular bottom plate. The bottom plate is connected to the outer wall of the filter element and rotating with the same angular velocity. Therefore, it can in good approximation be assumed that the flow is periodic in the tangential (angular) direction with the periodicity equal to the number of vanes. Consequently, it is sufficient to model the area between two vanes. In axial direction only the area between the two horizontal plates of the stationary casing is taken into account. The spacing between the upper horizontal plate and the filter element functions as the inflow boundary, where the velocity of the leak flow has to be prescribed. The spacing between the lower horizontal plate of the stationary casing and the filter element is the outflow boundary. The three walls of the stationary casing which bound the flow domain are treated as solid walls on which the gas velocity has to be equal to zero. The vanes, the impeller bottom plate and the filter element wall are treated as walls with fixed angular velocity. Finally, the boundaries of the model in the planes of the vanes are treated as periodic boundaries. The dimensions of the geometry modelled here are given in Fig. 2. The number of vanes equals $n=40$, the height of the vanes equals $h=30$ mm, the widths of the gaps above and below the vanes and the width of the inflow and outflow boundaries are $S=15$ mm. The distance from the rotation axis to the impeller is $R_c=300$ mm, the distance from the axis to the outer radius of the vanes is $R_s=395$ mm and the distance from the axis to the stationary casing is $R_0=410$ mm. In all calculations the thickness of the plates has been neglected. The angular velocity is always taken as 3000 rpm or $\Omega=314.2$ rad/s.

The numerical calculations have been performed with the CFD software package CFX4, which is a structured flow solver based on a finite volume method. Due to the cylindrical shape of the geometry cylindrical coordinates and velocity components are used. We will denote the velocity components by v_r in radial direction, v_θ in tangential direction and v_z in axial direction. A uniform grid in the three directions has been used. We performed calculations on two grids. a coarser grid with 66 points in

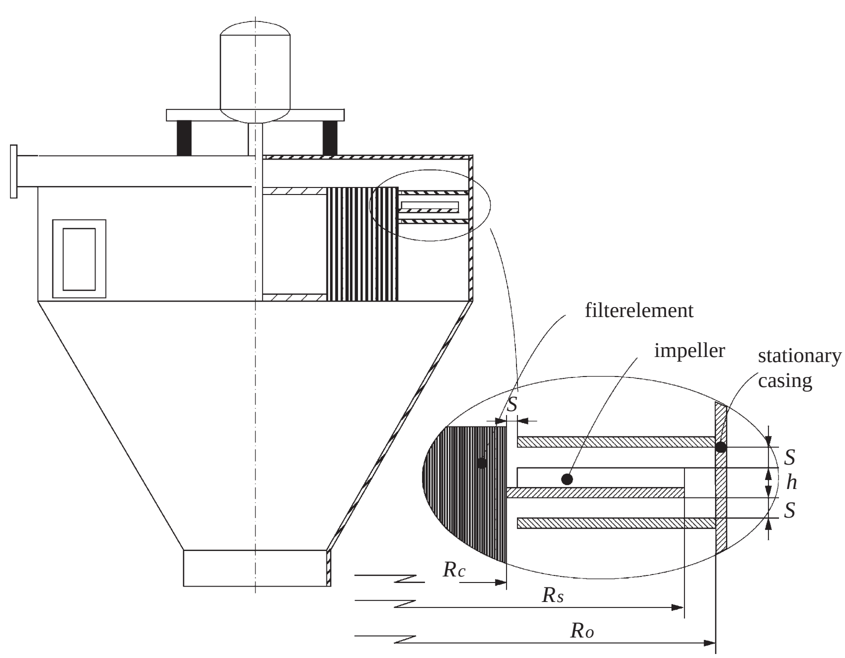

Fig. 2. Sketch of RPS and impeller.