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 preseparator, 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...
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 \( h = 30 \text{ mm} \), the widths of the gaps above and below the vanes and the width of the inflow and outflow boundaries are \( S = 15 \text{ mm} \). The distance from the rotation axis to the impeller is \( R_c = 300 \text{ mm} \), the distance from the axis to the outer radius of the vanes is \( R_s = 395 \text{ mm} \) and the distance from the axis to the stationary casing is \( R_o = 410 \text{ mm} \). In all calculations the thickness of the plates has been neglected. The angular velocity is always taken as \( 3000 \text{ rpm} \) or \( \Omega = 314.2 \text{ rad/s} \).

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, \( v_\theta \) 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
radial direction, 30 in tangential direction and 60 in axial direction, and a finer grid with \(88 \times 40 \times 80\) points. Since the differences between the results on both grids are on the order of 3%, we concluded that these grids are sufficiently accurate to fit our purpose.

Due to the high angular velocity of the vanes the flow in the impeller will be turbulent. It is assumed that there is a state of stationary turbulent flow and a turbulence model is used to solve the Reynolds-averaged Navier–Stokes equations. In CFX4 several turbulence models are available. We restrict ourselves to the standard \(k–\varepsilon\) model [5] and the differential Reynolds-stress model (RSM) [6]. The latter model tends to suffer from convergence problems. A remedy which sometimes helps is to start the calculation from a converged result of a simulation with the \(k–\varepsilon\) model. In the model the properties of air at a temperature of 473 K, which is realistic for the RPS in operation for cleaning of exhaust gases in biomass combustion, is used.

This leads to a density of \(\rho = 0.74 \text{ kg/m}^3\) and dynamic viscosity of \(\eta = 2.58 \times 10^{-5} \text{ Pa s}\). In the inflow boundary the velocity of the gas is prescribed. The axial velocity is taken as \(-4.9 \text{ m/s}\) which corresponds to a leak flow of \(Q_l = 500 \text{ m}^3/\text{h}\), the radial velocity is set equal to zero and the tangential velocity equals \(59.6 \text{ m/s}\), which follows from the design program for the total configuration.

3. One-dimensional model

In this section we consider the equations describing a turbulent flow in a small area between a rotating and a stationary plate. The distance between the two plates is denoted by \(d\). Apart from the turbulent velocity induced by the rotating plate, there is a mean flow in the radial direction with flow rate \(Q\) driven by a pressure difference. The main assumptions of the one-dimensional theory are that the velocity components and pressure are functions of the radial coordinate only and that due to the high angular velocity of the rotating plate the tangential velocity component is large compared to the other two components. These assumptions imply that the continuity equation can be reduced to:

\[
\frac{d(rv_r)}{dr} = 0, \tag{1}
\]

or

\[
v_r = \frac{Q}{2\pi rd}. \tag{2}
\]

The momentum equation in tangential direction can be written as:

\[
\rho v_r \left( \frac{dv_r}{dr} + \frac{v_\theta}{r} \right) = \frac{\tau_{w,\theta}}{d}, \tag{3}
\]

where \(\tau_{w,\theta}\) is the tangential component of the shear stress. This shear stress component is given by [7]:

\[
\tau_{w,\theta} = -\frac{1}{2} \rho \left( C_{f,rot} (v_\theta - \Omega r) \sqrt{(v_\theta - \Omega r)^2 + v_r^2} + C_{f,stat} v_\theta \sqrt{v_\theta^2 + v_r^2} \right), \tag{4}
\]

where \(C_{f,rot}\) and \(C_{f,stat}\) are the friction coefficients for the rotating and stationary plates. If these friction coefficients and the radial flow rate are known, (3) results in a first-order differential equation for \(v_\theta(r)\), which can be solved numerically if a boundary condition in the upstream boundary is known.

The momentum equation in radial direction reads:

\[
\rho \left( \frac{dv_r}{dr} - \frac{v_\theta^2}{r} \right) = -\frac{dp}{dr} + \frac{\tau_{w,r}}{d}, \tag{5}
\]

where \(\tau_{w,r}\) is the radial component of the shear stress, which can be written as:

\[
\tau_{w,r} = -\frac{1}{2} \rho \left( C_{f,rot} v_r \sqrt{(v_\theta - \Omega r)^2 + v_r^2} + C_{f,stat} v_r \sqrt{v_\theta^2 + v_r^2} \right). \tag{6}
\]

If \(v_r\) is known through the radial flow rate and \(v_\theta\) from the solution of (3), (5) can be used to obtain the pressure. However, due to the fact that \(v_r \ll v_\theta\), and due to the small friction coefficients, (5) can in good approximation be simplified to

\[
\frac{dp}{dr} = \frac{v_\theta^2}{r}. \tag{7}
\]

In the next section the simulation results will be presented and compared with this one-dimensional model.

4. Results

4.1. Convergence

On the standard grid a solution has been calculated for both the \(k–\varepsilon\) model and the Reynolds-stress model. The latter calculation has been started from the converged solution with the \(k–\varepsilon\) turbulence model. The convergence history for both calculations is plotted in Fig. 3. In the figure the residuals of the mass and tangential velocity component have been plotted as a function of the number of iterations. The first 3800 iterations are performed with the \(k–\varepsilon\) model. The last 3500 with the Reynolds-stress model. It can be seen that both models converge quite well, but for the Reynolds-stress model this is only possible if the calculation starts from the converged \(k–\varepsilon\) solution. The residuals of the other
quantities converge in a similar way. In the following, the results with the $k-e$ model are taken from the solution after 3800 iterations, the results with the Reynolds-stress model are taken from the solution after 3500 additional iterations.

In the discussion of the results and the formulation of the one-dimensional analytical model we distinguish between the various parts of the configuration: the upper part between and above the impeller vanes, the lower part below the bottom plate, and the transition part between the vanes and the outer casing. In the upper part the mean radial velocity component is positive, in the lower part it is negative and in the transition part the radial velocity changes sign.

4.2. Upper part

We start with the presentation of the numerical results in the upper part between and above the impeller vanes. First, we consider pointwise results in the plane at half the height of the impeller. In Fig. 4 the tangential velocity component is shown for the $k-e$ model and the RSM. In Fig. 5 the pressure is shown for both models. We see from these results that indeed the tangential velocity and the pressure hardly depend on the tangential coordinate, which is consistent with the basic assumption of the one-dimensional analytical model.

This equally holds for the other velocity components. It has also been verified that the tangential velocity component and pressure hardly vary with the axial coordinate in the upper part of the impeller, apart from a thin boundary layer at the upper stationary casing in which the tangential velocity component rapidly decreases to zero. However, the radial velocity component does vary with the axial and tangential coordinate, but the radial velocity component is only approximately 4% of the tangential component. Furthermore, we see that the results of the two models are quite close in this part of the impeller.

Since the flow hardly depends on the tangential coordinate, in the sequel we will only consider results averaged over this direction. Furthermore, since the results of the two turbulence models are almost equal in this part of the computational domain we will restrict to the RSM results. In Fig. 6 contour plots of the average tangential velocity component are shown. Fig. 7 shows the average radial velocity component and Fig. 8 the average pressure. In these figures the position of the vanes is indicated by straight lines. In these figures we see that the pressure in the upper part hardly depends on the axial coordinate. The same holds for the tangential velocity component apart from the boundary layers. The radial velocity component, however, does depend on the axial coordinate in the upper part. We will return to that below.

In Fig. 9 the results for the tangential velocity in the upper part are plotted as a function of the radial coordinate. Here averaged results are shown over a plane perpendicular to the radial direction. We see again that the two numerical results are in good agreement. Moreover, after a small transient in which the boundary condition for the tangential
velocity plays a role, the tangential velocity component is in good agreement with solid body rotation:

\[ v_\theta = \Omega r. \] (8)

For this geometry, in which the height of the vanes is larger than the spacing between the vanes and the upper stationary casing, this is not surprising. A model based on (3) will only give results in agreement with the numerical simulation in case a very large value of \( C_{f,rot} \) is taken.

In Fig. 10 the results for the average pressure are shown. Again, the results of the two turbulence models are quite close. According to the one-dimensional model the pressure can be found from (7), where \( v_\theta \) is given by (8). The result is included in the figure and it can be seen that the agreement with the numerical results is good. The offset in the beginning is again caused by the boundary condition for the tangential velocity component at the inflow boundary.

Next, we turn to the radial velocity component in the upper part of the geometry. As stated before, and can be seen clearly in Fig. 7, the average radial velocity component does depend on the axial coordinate. Between the impeller vanes it is almost constant, but above the vanes it decreases and attains negative values. In Fig. 11 a vectorplot of the averaged radial and axial velocity components is given in the upper part of the geometry. It is clearly visible that a vortex is present, leading to a higher radial velocity between the vanes and a negative radial velocity above the vanes. In order to include this phenomenon in the one-dimensional model, we divide the upper part of the geometry into two parts. The first part is between the vanes where the radial velocity does not depend on the axial coordinate and the total radial flow rate equals \( Q_t + Q^* \), so that

\[ \nu_r = \frac{Q_t + Q^*}{2\pi r S}. \]

The second part is above the vanes, where the radial velocity component decreases to a negative value and the total radial flow rate equals \( -Q^* \). This flow rate can be found by combining (3) and (7). We solve (3) with the unknown \( Q^* \) in the area between the vanes and the upper
horizontal casing. Here we consider the fluid in solid body rotation in the area between the vanes as the rotating plate. The friction coefficients of both plates are taken equal to 0.008. Subsequently the pressure difference in this area between the radial positions $R_s$ and $R_c+S$ is calculated with (7). The correct value for $Q^*$ is the one for which this pressure difference equals the pressure difference over the same radial distance in the area between the vanes.

In Fig. 12 the tangential velocity component in the area above the vanes is plotted as a function of the radial coordinate. In the figure the numerical results are for the RSM and averaged over tangential and axial coordinate. After a transient close to the trailing edge of the impeller the agreement with the one-dimensional model is quite good. The radial velocity component is shown in Fig. 13. Considering the fact that the boundary between the two layers in the upper part of the geometry is in reality not so well defined the agreement between the numerical results and the one-dimensional model is not so bad.

4.3. Transition part

In the transition part between the impeller and the outer casing the radial velocity reverses from positive to negative. In the previous section we formulated a one-dimensional model for the upper part, which makes it possible to calculate the tangential velocity and pressure at the trailing edge of the impeller. In the next section a model for the lower part will be formulated in order to find the pressure drop over the lower part of the geometry. As in the part above the vanes this model will be based on (3) and (7). Hence, the pressure drop will depend on the tangential velocity in the lower part, and more specifically on the upstream boundary condition for the tangential velocity. The model for the transition
The tangential velocity component changes in the transition part due to viscous losses over the stationary casing. This can be described by an integral angular momentum balance over this transition part:

$$R_s \int_{A_{\text{inlet}}} v_n v_r dA = R_s \int_{A_{\text{inlet}}} v_n v_{\theta} dA - \frac{1}{2} C_s \int_{A_{\text{casing}}} r v_{\theta}^2 dA.$$  \hspace{1cm} (9)

Here, $A_{\text{inlet}}$ and $A_{\text{outlet}}$ are the areas at $r=R_s$ above and below the impeller bottom plate, $A_{\text{casing}}$ is the area of the stationary casing between $r=R_s$ and $r=R_0$ and $C_s$ is a friction coefficient. The normal velocity component in the direction of the mean flow is denoted by $v_n$. 

For the integral over the outlet we use the one-dimensional model in which the tangential velocity component does not depend on the axial coordinate. Hence,

$$\int_{A_{\text{outlet}}} v_n v_{\theta} dA = Q v_{\theta,b},$$

where $v_{\theta,b}$ is the tangential velocity component at the inflow boundary of the lower part.

The integral over the inlet is more complicated. As before, we divide this area in two parts. The first part is between the vanes, where $v_{\theta}$ and $v_r$ are constant. The tangential component is given by solid body rotation: $v_{\theta} = \Omega R_s$ and the radial component is determined by the flow rate: $v_r = (Q_1 + Q^*)/(2\pi R_s h)$. For the second part we investigate the numerical results for the radial and tangential velocity components plotted in Figs. 14 and 15. We see that the radial velocity component can well be described by a linear function of the axial coordinate which satisfies the

![Fig. 12. Average tangential velocity as a function of radial coordinate in the part above the vanes; solid: RSM; +: one-dimensional model.](image1)

![Fig. 13. Average radial velocity as a function of radial coordinate in the part above the vanes; solid: RSM; +: one-dimensional model.](image2)

![Fig. 14. Average radial velocity as a function of axial coordinate above the trailing edge of the impeller; solid: RSM; +: one-dimensional model.](image3)

![Fig. 15. Average tangential velocity as a function of axial coordinate above the trailing edge of the impeller; solid: RSM; +: one-dimensional model.](image4)
total flow rate over this part, being equal to \(-Q^*\). The tangential velocity component is well described by a 1/7th power law \[8\]. With these models for \(v_r\) and \(v_h\) the integral over the inlet can easily be evaluated.

Finally, we consider the integral over the casing. We approximate it by taking for the tangential velocity the average of its value just above the trailing edge and at the outlet: 
\[v_h = \frac{\sqrt{2}}{3}(\Omega R_s + v_{h,b}).\]
We also average the radial coordinate: 
\[r = \frac{1}{2}(R_s + R_0).\]
For the friction coefficient a value of \(C_s = 0.055\) gives good results and the surface area of the casing can be calculated from the geometry: 
\[A = \frac{2}{2}(h + 2S) + 2\pi(R_s^2 - R_0^2).\]

For the parameter values mentioned in Section 2, this leads to a value of the tangential velocity component below the trailing edge of the impeller of \(v_{h,b} = 104\) m/s. This value corresponds well with the numerical results of the RSM, where a value of 105 m/s was found. The \(k-\varepsilon\) model, however, yields a value of 115 m/s, which implies that in this model the friction loss at the casing wall is underestimated. This difference between the two models leads to large differences in the tangential velocity component in the lower part of the geometry, as can be seen from a comparison of Fig. 6 for RSM with Fig. 16 for the \(k-\varepsilon\) model. Since the pressure drop in the lower part is directly related to the tangential velocity component the pressure difference over the total impeller is also substantially different for both models. In literature it has been reported before that the RSM performs better than the \(k-\varepsilon\) model for rotating flows \[9\].

### 4.4. Lower part

In the lower part of the geometry the pressure is reduced again due to the centrifugal term in the Navier–Stokes equation. In Fig. 17 the tangential velocity component for the RSM, averaged over the tangential and axial direction, is plotted as a function of the radial coordinate. Included is the
result of the one-dimensional model based on (3) with friction coefficients $C_{\text{rot}} = C_{\text{stat}} = 0.007$. It can be seen that the agreement is good. Note, that also for the $k-\epsilon$ model the agreement between the numerical result and the one-dimensional model is good, if the numerical result for $v_{\theta,b}$ is taken as a boundary condition. The corresponding pressure, together with the result based on (7), is plotted in Fig. 18. Since the model agrees well with the numerical results for the tangential velocity component, the agreement between the results for the pressure was to be expected.

4.5. Design considerations

Using the one-dimensional model the pressure difference over the impeller can be calculated as a function of the leak flow $Q_l$. The result is shown in Fig. 19 by the solid line. During operation of the RPS the pressure drop over the filter element will increase due to contamination of the filter channels. The leak flow over the impeller can be found from the figure as the value where the pressure difference over the impeller matches the pressure drop over the filter. We see from the figure that this implies that the leak flow will decrease in time. At a certain moment the pressure drop over the filter is so large that it cannot be matched anymore by the pressure difference over the impeller. At that moment the leak flow will change sign and contaminated gas will flow to the clean side of the RPS. In practice the filter channels should be cleaned before this can happen.

In the figure also numerical results are included at various values of the leak flow. The general trend of the curve is well predicted by the theory, but especially at the higher flow rate the difference between the numerical result and the one-dimensional theory is quite large. Inspection of the numerical results showed that this is mainly caused by a too low value of the tangential velocity component at the beginning of the lower part of the impeller in the one-dimensional model. This leads to a too low pressure drop in the lower part at hence to a too high pressure difference over the impeller. The too low tangential velocity component is caused by a too high friction loss in the transition part and this can be overcome by adjusting the value of the friction coefficient $C_{\text{v}}$. In the figure results of the one-dimensional model with $C_{\text{v}} = 0.04$ are indicated by the dashed line. For this value of $C_{\text{v}}$ the overall agreement is better, but the discrepancy at lower leak flow increases. An important quantity in the design of the impeller is the power needed. The power can be estimated by:

$$P = \rho (Q_l + Q^*) \left( v_{\theta,\text{out}}^2 - v_{\theta,\text{in}}^2 \right),$$

where $v_{\theta,\text{out}}$ is the tangential velocity component at the edge of the bottom plate and $v_{\theta,\text{in}}$ at the inflow boundary. With the use of the one-dimensional model we find that $P = \rho (Q_l + Q^*) \Omega^2 (R_1^2 - R_2^2)$. In Fig. 20 the power needed for the impeller is plotted as a function of the leak flow. 

![Fig. 20. Power needed for the impeller as a function of the leak flow.](image)

caused by the vortex in the upper part of the impeller leads to a substantial increase in power needed for the operation of the impeller.

5. Conclusions

Numerical calculations have been performed to study the flow in an impeller mounted on an RPS and to find a suitable analytical model for the relation between the pressure difference over the impeller and the flow. Two turbulence models have been tested: the standard $k-\epsilon$ method and the differential Reynolds-stress model. It appeared that the results of the latter model agree better with analytical models in the region around the tip of the impeller bottom plate.

The numerical results are used to formulate a one-dimensional model for the flow in the impeller. Although this model was based on numerical results at one flow rate, the results of the model are in reasonable agreement with numerical results in a range of flow rates relevant for the RPS.

In the region between the vanes of the impeller the tangential velocity follows solid body rotation $v_{\theta} = \Omega r$ and the pressure from the centrifugal term in the radial Navier–Stokes equation (7). This region is further characterised by a large vortex, which leads to a larger radial flow between the vanes and a negative radial flow above the vanes. The strength of this vortex can be found from an integral tangential momentum balance in the region above the vanes, requiring that the pressure differences between and above the vanes are equal.

In the region below the impeller bottom plate the tangential velocity component follows from the tangential Navier–Stokes equation (3) with a suitable value of the friction coefficient. The boundary condition for the tangential velocity at the beginning of the lower part can be found from the momentum loss in the transition part between the upper and lower part.
The one-dimensional model is sufficiently simple to integrate it in a design programme for the whole RPS configuration. In this way an RPS including the sealing impeller can easily be designed according to various specifications.

Acknowledgement

This work was supported by the European Union under contract number ERK6-CT-1999-0003, “Aerosols in fixed-bed biomass combustion”. The authors wish to thank our partners for the fruitful exchange of ideas.

References


