**Paper on the 7th Workshop on Two–phase Flow Predictions,
April 11–14, 1994, LSTM Erlangen–Nürnberg, F.R.G.**

# Numerical Two-Phase Flow Predictions of Fine Particle Classification based on the 'Coanda effect'

I. Schulze, Th. Frank and D. Petrak
Fachbereich Maschinenbau
Lehrstuhl für Technische Thermodynamik
und Wärmelehre
Universität Chemnitz-Zwickau
Projektgruppe Mehrphasenströmungen

**Abstract**

The aim of this paper is the investigation of the sharp separation of micron or submicron fine particles by air classification. The phenomenon of the Coanda effect enables the classification of ultra fine particles by centrifugal force. The flow type can be established in a special Coanda geometry using a free jet - the so called Coanda jet - and a control jet of lower velocity. The outcome should be a sharp classification for a defined particle cut-off size. The measurements of velocity distributions in the paper from Okuda and Yasukuni (1981) are used for the comparison of own numerical simulations with these experimental measurements. The Coanda effect can be calculated with a Navier - Stokes - Solver. The cut-off location for a 150.0 m/s Coanda jet velocity and particles of 1.0 $\mu m$ diameter is 0.02 m behind the Coanda cylinder for a Coanda radius of 0.03 m. The cut-off location is not sharp enough for particle diameters of 1.0 $\mu m$ and smaller if the Coanda jet velocity is lower than 150.0 m/s.

## 1. Introduction

The importance of classification of ultra fine particles is very significant for chemical, pharmaceutical and other industrie. The effort to get finer particles is necessary to manufacture products with higher quality. There are different mechanisms for air precipitation like air separators which separate the particles with centrifugal force or gravity in counter-current or crosswise-current technique and so on. Another way of particle classification is the application of the Coanda effect. The Coanda effect is the interaction of a nozzle jet with a convex surface. The fluid between jet and surface is accelerated through the impulse exchange with the jet. The wall prevents the instream of new fluid - a low pressure between wall and jet is the consequence. The jet leaves its stream direction because of that low pressure and follows the curvature of the convex wall surface. This effect gets the name Coanda from the 1932 and 1934 patents by H. Coanda. The purpose to separate sharply micron or submicron particles by air classification with application of the Coanda effect was object of some studies, i.e. Okuda, Yasukuni (1981) and Guitton, Yasukuni (1977). The results were mostly experimental, the particle trajectories were calculated analytical. The aim of this paper is the numerical solution of the Navier - Stokes equations for the fluid phase and the calculation of particle trajectories in a simple plane Coanda geometry (fig. 1). The particles are fine molten aluminia. The geometry consists of two halfcylinders with a radius of 0.03 and 0.12 meters.

## 2. Equations of fluid and particle motion

The turbulent two phase flow (gas-particle) is described under the assumption that the particulate phase is dilute and therefore inter-particle effects are neglected. The two-phase flow is statistical stationary, incompressible and isothermal. The gas phase is Newtonian and has constant physical properties. The fluid flow calculations are based on the time-averaged Navier - Stokes equations in connection with the standard $k-\varepsilon$ turbulence model (Launder and Spalding 1974). The general form of the twodimensional elliptic differential equation for the fluid phase can then be written as:

$$\frac{\partial}{\partial x}\left(\rho_F\, u_F\, \Phi\right)+\frac{\partial}{\partial y}\left(\rho_F\, v_F\, \Phi\right) = \frac{\partial}{\partial x}\left(\Gamma\frac{\partial \Phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma\frac{\partial \Phi}{\partial y}\right) + S_\Phi \tag{1}$$

where $\Phi$ stands for $u_F$, $v_F$, $k$ and $\varepsilon$. The terms $S_\Phi$ and $\Gamma$ are the source term and the effective diffusion coefficient respectively. The continuity equation is obtained by setting $\Phi = 1$ , $\Gamma = 0$ and $S_\Phi = 0$. The source terms for the standard $k-\varepsilon$ model, the effective viscosity, the diffusion coefficient $\Gamma$ and the standard $k-\varepsilon$ model coefficients are as follow:

$$\begin{aligned}
S_k &= P_k - \rho_F\varepsilon \\
S_\varepsilon &= C_{\varepsilon 1}\,\frac{\varepsilon}{k}\,P_k - C_{\varepsilon 2}\,\rho_F\,\frac{\varepsilon^2}{k} \\
\mu_{eff} &= \mu_0 + C_\mu\,\rho_F\,\frac{k^2}{\varepsilon} \quad , \quad \Gamma_\Phi = \frac{\mu_{eff}}{\sigma_\Phi} \\
C_\mu &= 0.09 \quad , \quad C_{\varepsilon 1} = 1.44 \\
C_{\varepsilon 2} &= 1.92 \quad , \quad \sigma_k = 1.0 \quad , \quad \sigma_\varepsilon = 1.33
\end{aligned} \tag{2}$$

where $P_k$ is the rate of production of turbulence. The influence of particle motion on fluid turbulence characteristics was neglected. The logarithmic law of the wall was assumed for the boundary conditions of the turbulence equations.

The particle phase was described by the Lagrangian approach where a large number of particles were followed in time along their trajectories through the flow domain. Each particle trajectory represents a number of real particles with same physical properties. The particle trajectories were determined by solving the ordinary differential equations for the particle location and velocity components. A large density ratio $\rho_p/\rho_F$ was given and so the force due to particle rotation, the pressure gradient in the flow, the added mass force and the Basset history force are negligible. For particle calculations in the Coanda jet there is no gravity necessary. The equation of particle motion can be written as:

$$\frac{d\,x_P}{dt} = u_P \quad , \quad \frac{d\,y_P}{dt} = v_P$$

$$\frac{d}{dt}\begin{bmatrix} u_P \\ v_P \end{bmatrix} = \frac{3}{4}\frac{\nu\,\rho_F}{\rho_P\,d_P^2}\,Re_P\,C_D(Re_P)\begin{bmatrix} u_F - u_P \\ v_F - v_P \end{bmatrix} \tag{3}$$

with:

$$Re_P = \frac{d_P\,v_{rel}}{\nu} \tag{4}$$

$$v_{rel} = \sqrt{(u_F - u_P)^2 + (v_F - v_P)^2} \tag{5}$$

where $d_P$ is the particle diameter; $u_P$, $v_P$ are the particle velocity components in Cartesian coordinate system $(x, y)$; $C_D$ is the drag coefficient; $\nu$ is the kinematic viscosity and $\rho$ is the density of fluid (F) and particle (P) respectively. The drag coefficient $C_D$ is a function of $Re_P$.

## 3. Numerical Programs

The basis for the numerical calculations of two-dimensional fluid flows is the multigrid Navier-Stokes-Solver FAN-2D developed by Lilek and Peric 1993 (University of Hamburg). The turbulent flow is simulated by the standard k - $\varepsilon$ - turbulence model (Launder and Spalding 1974) in connection with the logarithmic law of the wall as boundary condition. Calculations can be made with this program for time dependent planar or axisymmetric, laminar or turbulent, incompressible fluid flows of Newtonian fluid in domains of arbitrary geometry. The numerical solution method in this code is based on the finite volume discretization of the governing equations. The block structured characteristic of this program was not used for the Coanda calculations.
A computer program PartFlow was developed for the predictions of particle trajectories (Frank 1991). This program simulates the motion of the disperse phase in a given fluid flow and is based on the Lagrangian approach. The numerical system of PartFlow and FAN-2D (with own implementations) together is capable to calculate the phase interaction caused by the impulse exchange between the fluid and the disperse phase. Therefore the Particle - Source -in cell model (Crowe 1977) is used in PartFlow. Because of the low volume flow rate of the disperse phase in the Coanda flow the influence of particle motion on the fluid flow was not taken into account.

## 4. Geometry and Numerical Mesh

The test Coanda equipment was simplified for calculations of air flow. Therefore a two-dimensional channel of 0.06 m width was bent over a half cylinder with a radius of 0.03 m. The entry of the Coanda (main) jet is located beside the inner cylinder wall and had a width of only 2.0

mm. The full cross section on the other side of the channel is the outlet. The control jet as a second one beside the Coanda jet had a width of 0.03 m (see fig. 1).

First numerical simulations were made in a geometry of only this two halfcylinders. This geometry was changed because of numerical divergence in the outlet region (like by backward facing step simulations with a too short outlet region). For that reason the numerical mesh is now six times longer than the Coanda radius. The technical interest is only from the inlet line up to the separation line without the prolongation. The mesh was configured with 237 x 104 grid nodes.
This configuration was calculated for several Coanda jet velocities up to 200.0 m/s without any control jet. The result was a large recirculation zone (near the zoom window in fig. 1). The particles were captured in this recirculation zone and the classification was not possible.
A smaller control jet was described by Okuda and Yasukuni (1981) without informations about the velocity ratio of control to main jet. Some calculations with variations of velocity ratio and control jet width were made.

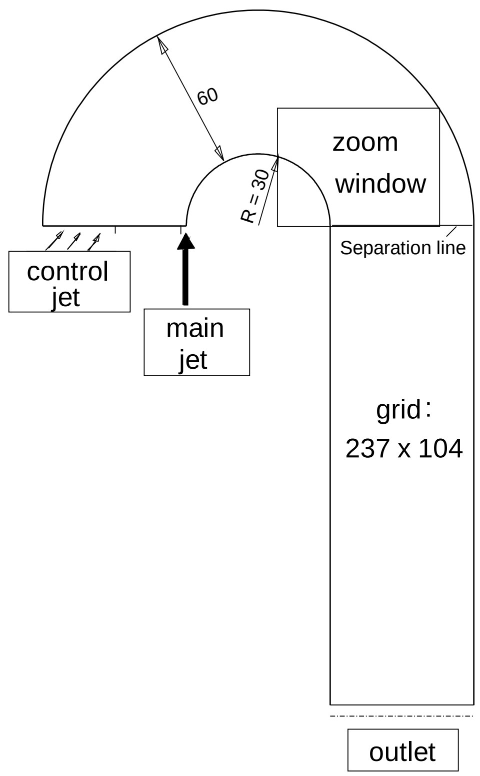

Fig.1 Coanda geometry

## 5. Results

The grid refinement in the Coanda jet region was necessary to get sufficient inlet grid points for the jet. Therefore the low distance between wall and the first numerical grid line can be a problem for the validity of the logarithmic law of the wall. The calculations have shown that the dimensionsless distance $y^+$ for such high velocities is between 4.0 and 20.0 in our calculations. The numerical convergence was quite good and the results calculated with the RNG - turbulence model are nearly the same.
The results of the control jet variations can be summarized as follows. A control jet width of 0.01 m produces a recirculation zone between main and control jet. The Coanda jet gets therefore a partly deflection with a trend to the control jet. Much more better results were realized with a control jet width of 0.03 m (half of the inlet cross section).
The numerical results for velocity ratios (control to main jet velocity) of 0.25 to 0.316 have shown that the influence of the control jet on the Coanda jet is less important in this range. All shown results are calculated with jet velocity ratio of 0.316 for a control jet width of 0.03 m.

The control jet is arranged on the left hand side of the instream cross section (fig. 1). The control jet velocity vectors had an inflow angle of 45° and the vector lengths were constant. The development of the Coanda jet velocity profile (fig. 2) agrees with the results of Fernholz (1966). This control jet produces a small base flow above the Coanda jet without any influence on the Coanda jet profile near the inner cylinder surface. All recirculation zones in the main flow region are disappeared. Only a very small recirculation zone exists between both jets which has no influence on the Coanda jet.

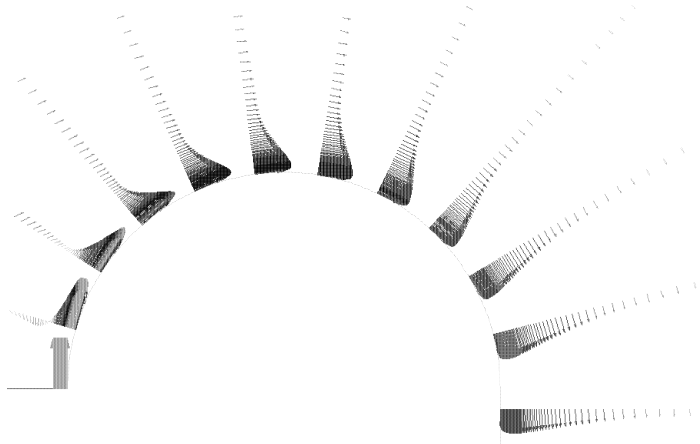

Fig. 2
Development of the coanda jet profile, main jet velocity = 200 m/s

The particle trajectories move along with the Coanda jet. The results of the numerical simulations were compared with the measurements of Okuda and Yasukuni (1981), especially the Coanda jet profiles.
The center line (line through velocity maxima of each profile, see fig. 2) was not the same in own calculations and in the measurements of comparison. The center line in the diagrams shown in the paper of Okuda and Yasukuni has a greater distance to the Coanda halfcylinder on the separation line. This difference may be explained by the absence of the control jet in the measurements of Okuda and Yasukuni. Our calculations with a control jet lead to a movement of the velocity maxima in the profiles after 90° away from the inner cylinder.
Furthermore these results are the starting point for the numerical calculations of the particle trajectories. A variation of different parameters was made during predictions. Particle trajectories were calculated for particle diameters of 0.5 to 5.0 $\mu m$ for a given fluid flow of Coanda jet velocities of 50.0 to 200.0 m/s.

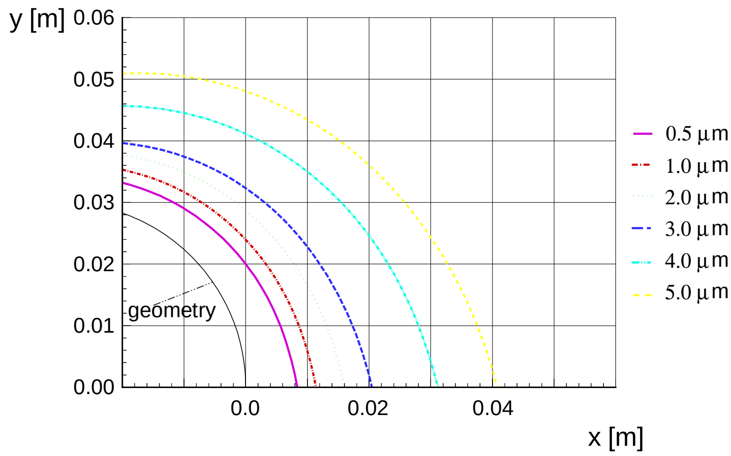


Fig. 3 Numerical results of particle trajectories, main jet velocity=50.0 m/s

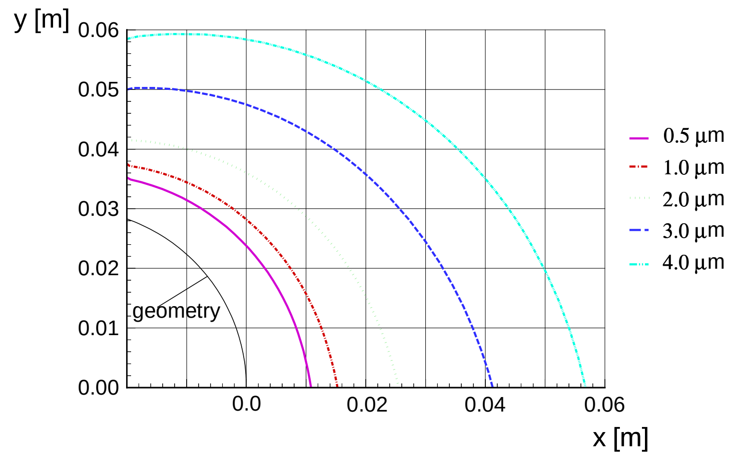


Fig. 4 Numerical results of particle trajectories, main jet velocity=200.0 m/s

All investigated particle diameters have their definite cut-off locations (see fig. 3 and 4) for a defined main jet intensity.

All particles are starting regular over the full main jet cross section of 2.0 mm and get nearly in one point of the separation line - the cut-off location. Only one particle trajectory for each particle diameter which starts in the center of the main jet inlet section was shown in fig. 3 and 4.

The separation results for main jet velocities of 50.0, 78.0, 150.0 and 200.0 m/s have a very good agreement with the results of Okuda and Yasukuni (1981). Fig. 3 and 4 show the cut-off locations in the zoom window (see fig. 1) for the calculated particle diameters for two main jet velocities. Particles of 5.0 $\mu m$ get wall interactions with the outer Coanda cylinder wall if the main jet velocity is 200.0 m/s (neglected here).

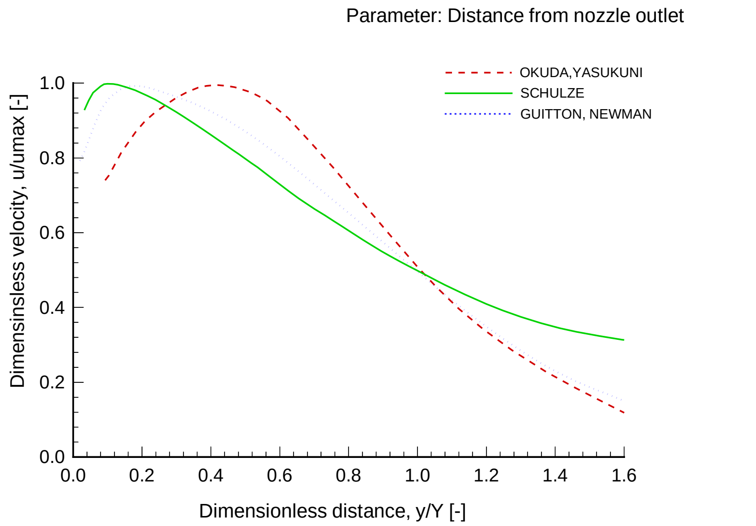


Fig. 5 Time mean velocity for the air in the main region of the jet, where Y is y-distance of $u_a = u_{max}/2$

The distance between cut-off location and the inner Coanda halfcylinder is greater for greater particles. The separation distance between particles of 1.0 and 2.0 $\mu m$ for a main jet velocity of 50.0 m/s is only 4.25 mm. For a main jet velocity of 200.0 m/s these particles were separated in a distance of 10.0 mm.

Fig. 5 shows the dimensionsless velocities in comparison to Guitton, Newman (1977) and Okuda, Yasukuni (1981). The difference after $y/Y = 1.0$ is caused by the control jet velocity of 31.6 % of the main jet velocity in our calculations. Reasons for differences in the maxima location can be different Coanda geometries, i.e. by Guitton and Newman the geometry is formed like a logarithmic profile. The cut-off locations for several main jet velocities are shown in fig. 6.

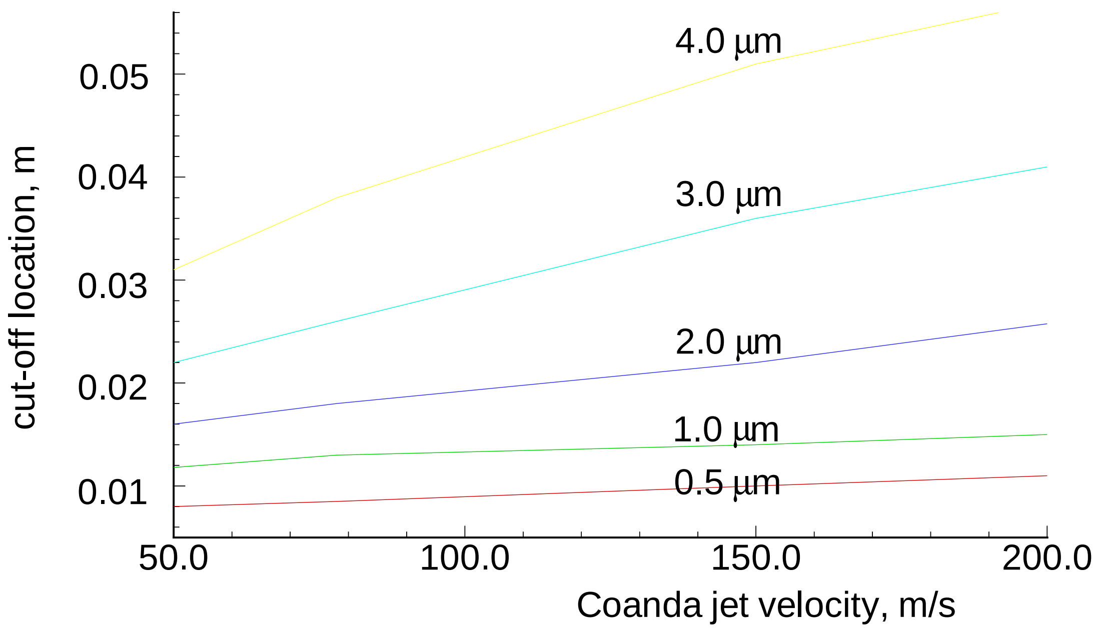


fig. 6 cut-off location

## 6. Conclusions

The calculations with PartFlow show that for main jet velocities of 150.0 m/s and higher the cut-off size of particles is $\geq 1.0$ $\mu m$ for a Coanda radius of 3.0 cm. The cut-off location for 1.0 $\mu m$ particles is lying 2.0 cm behind the inner Coanda cylinder on the separation line. For lower main jet velocities with regard to location and particle size the cut-off size is not sharp enough. High main jet velocities i.e. 150.0 m/s are necessary to separate particles of 1.0 $\mu m$ and finer. If the demand on the particle separation quality is less or greater particles should be separated lower main jet velocities are sufficient and energy can be saved.
A classification can be made for each micron or submicron particle size. Okuda and Yasukuni (1981) present that the classification is sharply for a main jet of 200.0 m/s. The particles up to 1.0 $\mu m$ can be separated 2.0 cm behind the bending edge. Therefore the agreement of numerical predictions and measurements is quite good.
The separation quality should be investigated later with own measurements. The influence of the Coanda radius on the cut-off location was not the object of this paper.

## References


Crowe, C.T.; Sharma, M.P. and Stock, D.E.: The Particle - Source - In - Cell (PSI-Cell) Model for Gas - Droplet Flows. Trans. of ASME, J. Fluids. Eng. (1977), Vol. 99, pp. 325 - 332

Durst, F.; Melojevic, D. and Schönung, B.: Eulerian and Lagrangian Predictions of Particulate Two - Phase Flows, Forschungsbericht SFB 80/T/222, 1983

Fernholz, H.-H.: Zur Umlenkung von Freistrahlen an konvex gekrümmten Wänden (Coanda - Effekt), Forschungsbericht Deutsche Luft- und Raumfahrt 66-21, 1966

Frank, Th.: Numerische Berechnung der feststoffbeladenen Gasströmung im horizontalen Kanal unter Berücksichtigung von Wandrauhigkeiten , Dissertation - Bergakademie Freiberg, 1991

Frank, Th. and Schulze, I.: Numerical Simulation of Gas-Droplet Flow around a Nozzle in a Cylindrical Chamber using a Lagrangian Model based on a Multigrid Navier - Stokes - Solver, to heldon International Symposium Methods for Multiphase Flows June 19-23, 1994 in Lake Tahoe, USA

Guitton, D.E. and Newman, B.G.: Self-preserving Turbulent Wall Jets over Convex Surfaces, J. of Fluid Mech., 1977, 3, 155-185

Hahnheiser, W.: Untersuchungen an einem Umlenkwindsichter unter besonderer Berücksichtigung des Coanda-Effektes, Dissertation - Technische Universität Clausthal, 1992

Launder, B.E. and Spalding, D.B.: The numerical computation of turbulent flows, Comp. Meth. Appl. Mech. and Eng., 1974, 3, 269-289

Lilek, Z. and Peric, M.: USER Manual for the FAN-2D, Version 1.0, Institut für Schiffbau der Universität Hamburg, 1993

Okuda, S. and Yasukuni, J.: Application of Fluidics Principle to fine Particle Classification, in: Proceedings of International Symposium on Powder technology, 1981

Patankar, S.V.: Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corporation, 1980