Proceedings, ICESP VII
Sep. 20-25, 1998, Kyongju, Korea

ESP PERFORMANCE PREDICTION AND FLOW OPTIMISATION USING CFD MODELLING

W Schmitz, D Gibson
Technology Research and Investigations
Eskom
P.O. Box 40175
Cleveland 2022
South Africa

L Pretorius
Department of Mechanical and Manufacturing Engineering
Rand Afrikaans University
P.O. Box 524
Aucklandpark 2006
South Africa

ABSTRACT

Computational fluid dynamics CFD has become widely accepted by industry as an accurate predictive tool to optimise aerodynamic flows.

Eskom (South Africa) over the past five years have extensively applied CFD and, by cross referencing all simulated results to measured data, have gained credibility in the application of this technology. CFD has been applied to simulate and optimise the flow and particle distribution within electrostatic precipitators. Past experience in this field has shown that the use of CFD can produce erroneous results specifically in predicting inlet flow distributions.

As one of the largest utilities in the world Eskom has access to different designs of ESPs and extensive test work, i.e. flow distribution tests, has built up an extensive library of data which has been used to correlate CFD results with measured data.. During the past years flow distribution measurements have been carried out on all of Eskom plant including those where modifications were implemented regarding skew flow. In addition, CFD has been extensively used on non Eskom plant.

This paper details some of the development work conducted by Eskom whereby which errors have become understood and rectified. Hence accurate predictions of the flow field within an ESP have been performed on seven different plants.

Current work is focused on incorporating all ESP collection dynamics (electrostatic field and forces, dust collection and re-entrainment) into a full CFD model.

INTRODUCTION

Eskom's dust collection plant consists mostly of ESPs (i.e. 73% of the total). With the ongoing reduction of allowable emissions, as stipulated by the Chief Air Pollution Control Officer (CAPCO) in South Africa, Eskom is seeking cost effective ways of reducing emissions further. In some cases the plant has been retrofitted with fabric filter systems or is in the process of being retrofitted. It is perceived that by optimising the operating conditions of Eskom's precipitators a considerable performance increase can be achieved.

Research has been initiated to achieve the capability of modelling all dynamic aspects of ESP performance using Computational Fluid Dynamics (CFD). A full modelling capability would create the opportunity to investigate the different influencing factors which govern the dust collection efficiency and lead to improved ESP performance. The dust collection within an ESP is influenced by a number of factors, namely:

Resistivity, ash quality, ash distribution, gas composition, gas temperature, gas distribution, duct geometry, inlet geometry, electrostatic forces, electrode geometry, collection plate geometry, rapping etc.

Literature describes many approaches to model ESP performance. Models have increased in their complexity, incorporating more and more parameters and effects which govern the performance of an ESP (Lawless, 1994). The literature bears enough information that can be used to model Electro-hydrodynamic effects in an ESP (Zamany, 1992). However, none of the researchers have applied the technology to a full ESP and combined the most suitable models into an economic, but realistic CFD model. Literature has stimulated encouragement for the development of an improved, second generation, ESP performance model. The improved model should be based on CFD technology and should incorporate Electro-hydrodynamic effects as well as re-entrainment and dust collection mechanisms.

Insufficient knowledge is available to correctly model re-entrainment effects in a CFD environment. The literature also lacks in the presentation of work relating to the modelling of ash build up on plates and the effects of rapping and ash dislodging from collection plates. None of the reported models take any maldistribution of particles or velocity into account throughout the ESP. Most models only consider a small portion of the ESP for correct modelling and assume that these conditions prevail throughout the complete ESP. Although these models are useful for the parameter study of influencing parameters and the subsequent performance optimisation, they are not sufficient to model the ESP as a whole.

As one of the largest utilities in the world Eskom has access to different designs of ESPs and extensive test work, i.e. flow distribution tests, has built up an extensive library of data which has been used to correlate CFD results with measured data.. During the past years flow distribution measurements have been carried out on all of Eskom plant including those where modifications were implemented regarding skew flow. In addition, CFD has been extensively used on non Eskom plant.

RESEARCH PROGRAM

From work carried out on plant and in the laboratory as well as CFD modelling it is perceived that the gas flow distribution throughout the ESP is of considerable importance. On a micro scale the flow development between the collection plates and the particle movement influences the collection dynamics. Therefore, the first part of the research program concentrated on the flow distribution and development within an ESP. The second part of the research concentrates on the movement of particles within the flow field from the inlet to the outlet of the ESP. The implementation of two phase flow modelling incorporating electrostatic forces, the effects of re-entrainment, the effects of different electrode and collection plate designs as well as rapping effects is envisaged in the near future. The work described in the following concerns mainly the development of flow modelling techniques undertaken during the course of 1997. This work forms part of the development of a more complete CFD model for ESPs.

Duct Geometry

Elevation differences between the air heater outlet and the ESP inlet often leads to an inlet duct geometry, which develops highly non uniform flow into the inlet of an ESP.

Modelling the ducts between the air heater and the ESP is to be carried out with sufficient care in order to establish the inlet conditions correctly. In cases of a temperature maldistribution at the outlet of the air heater the temperature stratification generally persists throughout the inlet duct and can even be observed within the casing of the ESP. The collection efficiency within the ESP can be adversely affected due to the temperature variation between the collection plates. CFD can be applied to either correct the temperature distribution or to take the temperature stratification into account in a complete performance model.

The inlet duct geometry also affects the particle distribution approaching the inlet of a precipitator. It is therefore imperative to model the paths of the different size dust particles. Turning vanes are often present in the ducts in order to reduce pressure losses and to control the approach velocity distribution into the inlet cone. The insertion of turning vanes into the flow domain is carried out by placing correctly shaped non-porous baffles into the flow domain. Turning vanes play a major role in the redistribution of dust within the duct, often leading to concentrated dust streams at the inlet of the ESP.

Modelling of Flow Distribution Devices

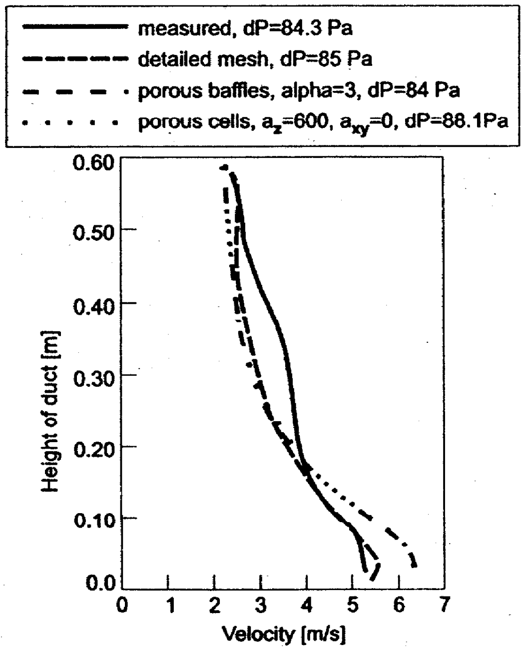
The inlet of the ESP consists of a diverging cone section in which flow devices are placed, manipulating the distribution of the flow to the desired inlet profile into the first field. Perforated screens, lattice screens, u-channels and rounded bars are typical insertions used for flow redistribution. The correct modelling of these devices is an important feature of an accurate ESP model. Extensive research work using laboratory tests, site tests and suitable sub-models has enabled Eskom to predict any type of flow distribution devices with sufficient accuracy.

Most distribution screen designs encountered in ESPs were tested in the laboratory for their pressure loss characteristics for angular and none angular flow. This data was then incorporated into the CFD model.

Perforated plates

Typically three vertical screens are placed in the inlet cone. In order to predict the flow pattern throughout the entire ESP it is imperative that the effects of the baffles on the flow distribution are correctly predicted in the CFD model.

In the past these screens were mainly modelled by placing a porous baffle in the position of the screens (Schmitz, 1996). This method met with various degrees of success. It was fairly accurate where the flow was perpendicular to the screen but was suspect when the flow approached the screen at angles considerably less than 90°.



Figures 1: FAR 44.4% (Schmitz, 1997)

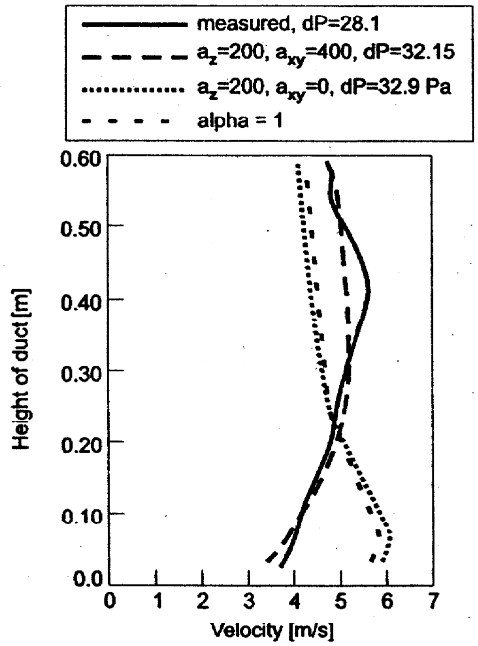


Figure 2: FAR 60% (Schmitz, 1997)

In these diverging sections however, large parts of the flow approach the screens at an angle and this can result in inaccurate flow distribution predictions. It was found that special adaptations had to be made if screens were to be modelled using layers of porous baffles. Since porous baffles only allow the definition of resistance parameters acting on the flow component perpendicular to the plate, considerable errors are encountered when placed in the three dimensional flow field of an inlet cone, Fig.2.

For the 44.4% free area ratio a detailed model of the screen showed good comparison between measured and predicted results, Fig.1.

The use of porous media to model perforated plates is better suited to model in areas of diverging or converging flow fields i.e. inlet and outlet cones. Porous media allows the definition of resistance parameters independently in all three space dimensions. Laboratory tests on a two dimensional, half cone model (divergence to the bottom), showed that for free area ratios (FAR) of fifty and below modelling should be carried out by using porous media. A comparison of both methods is shown in Fig.1-2. The α values indicated in the legend of the graphs are resistance values used in the CFD model and are chosen according to the screen resistance. In case of the porous baffles the subscript z indicates the resistance in direction of the main flow, whereas the xy are the resistance values orthogonal to the z direction.

Comparing the results for the lattice screen with a free area ratio (FAR) of 44.4% it can be seen that the detailed mesh best predicts the measured profile. For the detailed mesh a cell density was chosen which exactly reflects the geometry of the lattice screen. The "exact" model predicts the velocity profile as well as the pressure loss with good accuracy, Fig.1. The top and bottom of the graph represent the centre line and the diverged part of the test cone. Deviations in the measured and predicted velocity could be caused by inaccuracies in measurements resulting from the physical model tests i.e. Pitot traverse. By applying the respective resistance values to the porous baffle and porous media the predictions lead to a slight over prediction of the peak at the bottom of the duct, both profiles being almost identical. The predicted pressure drop however is close to the measured pressure loss.

Tests carried out on a screen with a FAR of 60% produced a measured velocity profile with higher velocities towards the top of the duct. For resistance values relating to the pressure loss produced by such a screen it can be seen that the porous media and the porous baffle both predict the wrong velocity distribution, Fig.2. As for the porous baffles no adaptation is possible to match the measured results without sacrificing the accuracy in predicting the pressure loss. An adaptation to a more realistic velocity distribution can be achieved with use of a porous media. Adaptation is achieved by applying suitable alpha values in the x and y direction. For a 60% free area ratio. Values for α_x and α_y of approximately twice the α_z value gives the closest fit, Fig.2. It would appear that the ratio of α_z to α_x and α_y is dependant on the free area ratio of the screen.

Lattice screens

The square lattice screen can be defined as a screen constructed from strips of flat plate, arranged horizontally and vertically. The density or free area of the screen is determined by the gaps between the strips which form relatively large openings compared to perforated plates. In a given flow domain these screens can be modelled by choosing a grid definition suitable to the width of the metal strips.

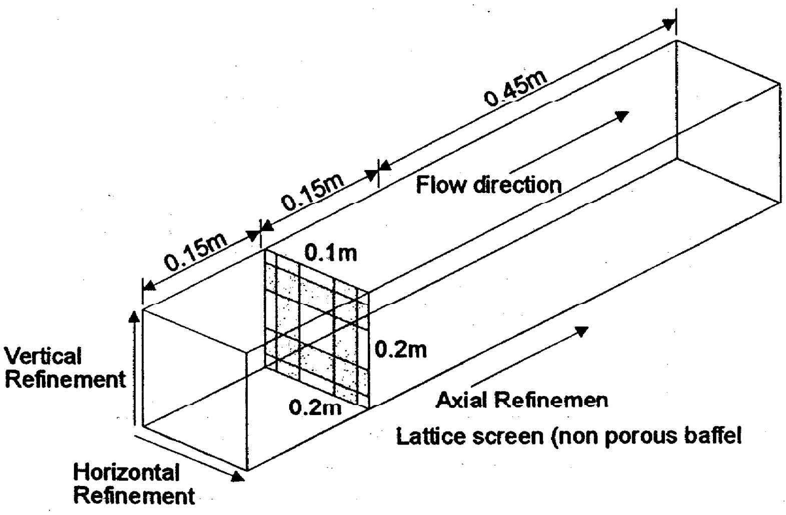


Figure 3: Sub-model for screen lattice, i.e. the strips indicate blocked off cells in the domain

In the sub-model the lattice screen was simulated by blanking off rows and columns of cells using non porous baffles. Although the model seems to predict the flow distribution in the inlet cones reasonably well, it was suspected that the pressure drop was not predicted with sufficient accuracy. In order to optimise the required cell density in the vicinity of the lattice the effect of mesh refinement on the pressure differential across a flat square lattice screen was investigated. A CFD submodel was used to predict the pressure loss across a typical lattice screen for different degrees of mesh refinement. Fig.3 shows the square lattice screen used for the purpose of this study. The square lattice screen consisted of a 10mm wide flat strip and a 20mm hole size, i.e. a FAR. of 44.44%. The accuracy of the pressure differential, as calculated by the CFD software, is dependent on the degree of refinement of the numerical mesh in close vicinity to the screen. The higher the degree of refinement which was applied to the model, the more accurate the flow around and through the screen was simulated, producing more accurate correlation to the actual flow conditions. It is however necessary to determine the minimum degree of refinement required to achieve acceptable correlation between a numerical simulation and actual conditions.

For further confirmation three literature sources were used to find the correct pressure differential across the screen (Miller, 1984) **ΔP = 166.9Pa**, (Blevins, 1984) **ΔP = 162.6Pa**, (Idelchik, 1986) **ΔP = 167.5Pa**).

It can be seen that results from all three literature sources correlate well and it can therefore be stated that the true pressure differential across the screen should be in the range of 162.6Pa to 167.5Pa at an approach velocity of 7.4m/s and an air density of 1kg/m^3.

For the purpose of this study, the vertical and horizontal refinement was always similar and was therefore equally refined for each different test series. The horizontal and vertical refinement will be referred to as the **parallel refinement**. Five series of tests were conducted where the parallel refinement was constant for each series of tests while the axial refinement was increased for each simulation. Axial refinement meaning the number of cells in the vicinity before and after the screen in flow direction.

All curves show a minimum of predicted pressure loss at low axial refinement and subsequently increase in predicted pressure drop with increasing axial refinement. Results for series 2, 3 and 4 converge towards the true pressure loss of approximately 165Pa as the axial refinement is increased. This indicates that the benefit of parallel refinement is limited. Results from series 0 and series 1 tend to over predict the pressure loss for high axial refinement values. However, most of the results show an under prediction of the pressure loss. From a cell density point of view series 0 and series 1 are the most economic configurations, together with low axial refinement. According to Fig.4, screen models would under predict the pressure loss by about 20 to 60 percent. In order to predict pressure losses more accurately, considerably finer meshes would have to be used in the vicinity of the screen. However, this would mean that the model would become uneconomic. In most cases the accuracy of the pressure loss prediction is secondary compared to the flow distribution.

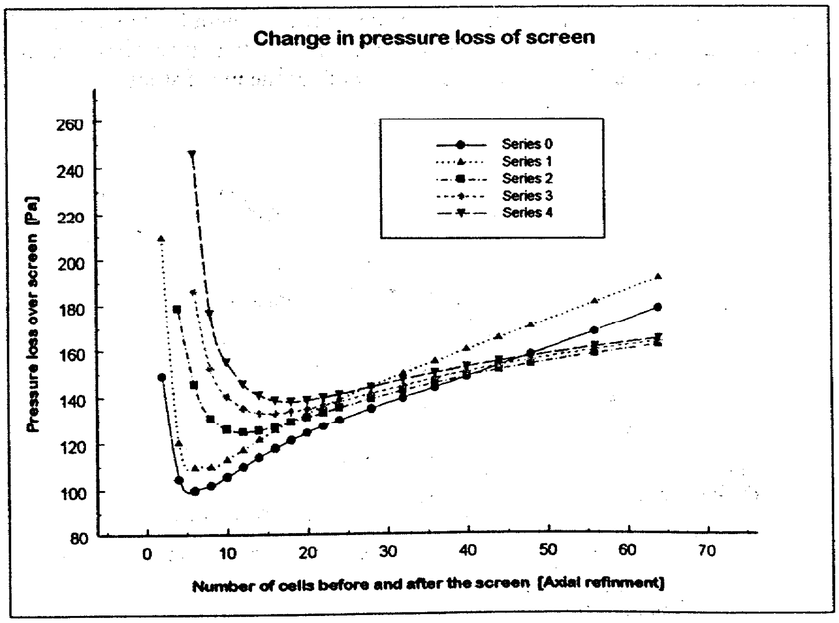


Figure 4: Pressure drop results for an approach velocity of 7.4m/s and a density of 1kg/m^3

Round bars and U-channels

The redistribution of flow within the conical section of precipitators is often carried out by the use of vertical, round bars or U-channels. Depending on the skew of the inlet flow profile into the cone a single or double row is installed. In the vicinity of the bars or channels the flow experiences acceleration through the narrow gaps between the evenly spaced obstructions. This causes the flow to distribute more evenly in the cross sectional area. Although in a CFD model of an ESP porous media can approximate screens, this would only be conducted when the model is limited to a certain number of flow cells due to computational limits posed by the hardware and software available. A more accurate way to model round bars would be by inserting solids into the computational mesh in place of the bars or to cut out parts of the domain according to the physical arrangement. U-channels would be modelled by placing non porous baffles into the flow domain which would approximate the shape and number of the channels. In both cases it has been found that the gap, i.e. horizontal distance between the obstructions, should consist of not less than two flow cells of free flow width in order to avoid errors in pressure loss prediction as described above.

Modelling of Collection Plates and Discharge Electrodes

The collection plates within the casing of a precipitator act as flow straighteners, which inhibit large scale turbulent mixing. On a small scale however, the shape of the plates and the presence of discharge electrodes can cause turbulence in the vicinity of the plate and around the discharge electrodes. Although the velocities in the casing are generally expected to be below 2m/s, the experienced resistance influences the distribution of the gas within the casing. This is often observed when flow distribution measurements before and after the first field are compared. Although the flow distribution may not be uniform in front of the field, behind the field these discrepancies are mostly overcome.

In order to model these effects, methods were developed to achieve accurate flow predictions without having to use the large number of cells required for a detailed model. A detailed model of collection plates and discharge electrodes may make the computation not viable for practical and commercial use.

The geometry of the plates, the distance between them and the presence of discharge electrodes causes a resistance characteristic that discriminates between the main flow direction and the vertical direction between the fields. Generally the resistance in the main flow direction is higher than the vertical one. This characteristic has been established in a detailed sub-model for a number of different plate geometries. Typically a detailed grid of a single lane is created and flow solutions are generated for a range of velocities between 0.5m/s and 5m/s. The grid is extended by 1m before and after the plates in order to account for entry and exit effects. The horizontal flow path is modelled by a single cell layer consisting of more than 20000 cells. For the vertical resistance prediction a grid is generated that allows flow simulation in a single vertical plate channel over the full length of the collection plate. From pressure plots the vertical and horizontal resistance characteristics of the plate-wire configuration is established.

A simplified sub-model, later used in the complete ESP, is then created and tuned to produce similar flow resistance characteristics.

In the complete model an approximation of the plate resistance can be achieved by using a porous media, which stretches over the entire volume of each field. The correct flow resistance is produced by assigning α and β values (resistance coefficients found in commercial CFD codes) to the three space dimensions of the porous media. The disadvantage of the porous media approach is that it can adversely affect the results for particle tracking through the domain. To accommodate particle tracking, the plates can be represented as none porous baffles, placed at a distance according to the actual plate arrangement. To achieve the different resistance values in horizontal and vertical direction, additional baffles can be inserted orthogonal to the main flow direction. This approach however results in a much larger number of cells within the casing of the ESP model.

Two Phase Flow Modelling

The particle trajectory through the entire flow domain of the ESP can be predicted by uncoupled, two phase flow modelling. Gravity forces, particle momentum effects (dependant on particle mass), drag forces and centrifugal forces determine the actual flow path of dust in the flow field.

The incorporation of two phase flow modelling enables the particle behaviour within an ESP to be studied. Particle trajectories indicate the likely distribution of different size particles at the inlet of the ESP which can be used to determine the most suitable gas flow distribution at the inlet to the casing. It is also useful in the study of hopper reentrainment and gas bypassing the plates. A typical particle trajectory is shown in Fig.5. Note that the trajectories in Fig.5-6 do not incorporate any electrostatic forces and are merely a result of gravity, momentum and drag forces. Particle impacting on any internal structure and walls are bounced back into the flow according to standard bounce factors (Star-CD, 1997).

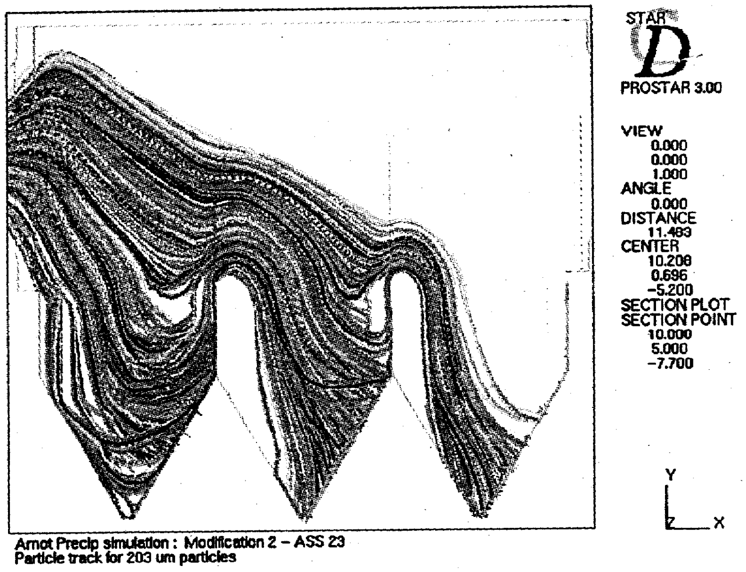


Figure 5: Particle tracks for 203μm particle size

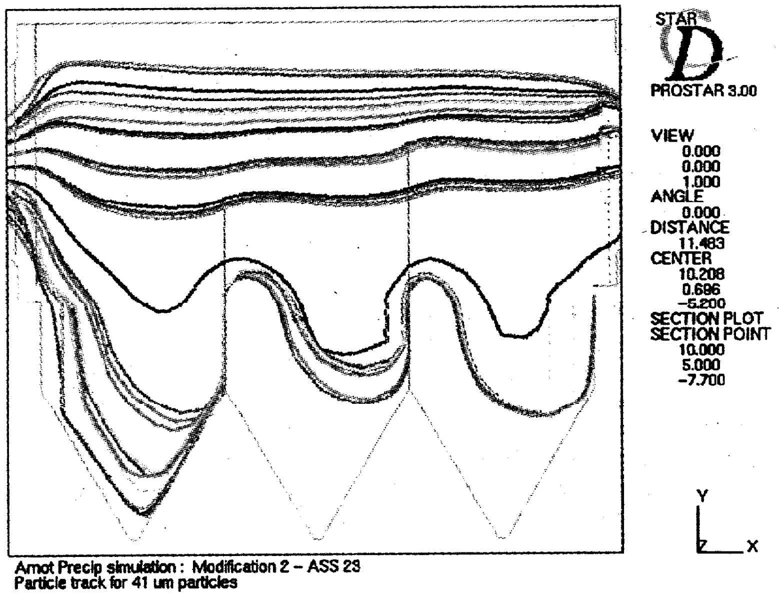


Figure 6: Particle tracks for 41μm particle size

Erosion Prediction

Two phase flow modelling allows the study of potential erosion in ducts, on vanes, internal structures, distribution screens and collection plates. The incorporation of an erosion model in some of the studies carried out by Eskom has led to the identification of high erosion areas within the ducts and of the baffles even before they occurred in the real plant. Results have been correlated to actual erosion patterns with excellent comparison. The erosion model considers the particle trajectory within the flow domain and various impact parameters on walls and internal structures. Impact velocity, angle of impact, particle size, number of impacts and erosion properties combined form the basis for the erosion prediction model.

CONCLUSIONS

From the work carried out it can be concluded that CFD can be used effectively to predict the flow within an ESP.

CFD can be used to investigate suitable flow modifications in order to improve ESP performance. With future implementation of collection dynamics into the CFD code it could be possible to model the performance of an ESP more accurately than currently practised. It will allow detailed studies of particle behaviour, which will create a better understanding of the effects of prevailing dynamics within an ESP. It is believed that these studies will create a wide range of opportunities to further improve the performance of ESPs. One of the advantages of CFD lies in the fact that modifications can be evaluated with confidence prior to installation.

CFD has played an important role in the implementation of the Skew Gas Flow Technology within Eskom.

REFERENCES

(Books)
Blevins, R.D., 1984, Applied Fluid Dynamics Handbook, New York, Van Nostrand Reinhold Company Inc.
Idelchick, I.E., 1986, Handbook of Hydraulic Resistance, 2nd Edition translated by Erwin Fried, Washington.
Miller, D.S., 1984, Internal Flow Systems, BHRA Fluid Engineering Series, Vol. 5, 1978.
Schmitz, W., et al, 1998, Modelling of Screens within Industrial Flow Applications, 2nd South African Conference on Applied Mechanics.

(Symposium proceedings)
Lawless, P.A., et al, 1994, ESPM: an advanced electrostatic precipitator model, 1994 IEEE.

(Reports)
Schmitz, W., 1996, Eskom internal report.
Schmitz, W., 1997, Eskom internal report.
Zamany, J., 1992, Modelling of Particulate Transport in Commercial Electrostatic Precipitators, PhD Thesis