CFD simulation of gas flow and particle movement in ESPs

Niels F. Nielsen and Leif Lind
FLS Airtech A/S
DK-2500 Valby
Denmark

Abstract

A first step in order to model the electro-hydrodynamic flow inside a full Electrostatic Precipitator (ESPs) is to consider the gas distribution separately. Flow management by gas distribution screens are needed in the inlet and outlet funnels and perhaps also between sections of different electrical fields. Currently, the traditional approach of cut-and-try in the design phase and physical scale model or field tests in the order executing phase is being replaced by Computational Fluid Dynamics (CFD) at FLS Airtech.

The distribution of three-dimensional flow and pressure are simulated by a commercial CFD code giving accurate and highly detailed information over the calculated domain. Furthermore, the code includes a specially designed FLS Airtech module for modelling the effect of the gas distribution screens.

The paper presents simulations of full ESPs including inlet funnel with gas distribution screens, precipitation sections with collector curtains and anti-sneakage baffles, hoppers with partition plates and outlet funnel with gas distribution screen.

Attention is paid to different inlet/outlet funnels, advantages of screens between sections of different electrical fields and on anti-sneakage baffles. Further, numerical simulations versus full scale and model scale measurements are discussed. The study of different inlet types demonstrates the flexibility of the FLS Airtech ESP design making it easy to implement depending on layout limitations or process conditions. Moreover, calculations with small particles indicate that sneakage is an important issue and implementation of anti-sneakage baffles can reduce emission significantly.

1. Introduction

Many approaches have been made in order to model ElectroStatic Precipitators (ESPs) (Egli et al., 1996, Houlgreave et al., 1996, Lawless, 1996, Medlin et al., 1998, Meroth et al., 1996, Schmid & Buggisch, 1998, Schmitz et al., 2001, Zamany, 1992). The literature naturally concentrates on electro-hydrodynamic effects with different model complexity and studies of different parameters present in an ESP simulation. However, so far almost all models are limited to consider only a small part of the ESP and assume that these conditions prevail throughout the complete ESP. On the other hand concentrating on the gas flow distribution modelling of a full ESP is possible. The gas distribution is of special interest because it is one of the parameters playing a role for optimal ESP operation. Furthermore, clients may require documentation or evaluation (uniform, skew or special distribution) of the gas distribution and the traditional approach of cut-and-try in the design phase and physical model testing and field tests in the order executing phase is being replaced by Computational Fluid Dynamics (CFD) simulations.

Regarding the ESP design the wish for compact installations demands use of wide angle cones, which must be fitted with proper vanes and screens in order to fulfil the demands from the client for a suitable gas distribution rather than a distribution with large regions of flow reversal. Many possibilities exists to prevent flow reversal caused by wide angle cones, e.g. swirl flow generators (Goenka et al., 1990), splitters (Kline, 1959), wall suction (Yang & El-Nasar, 1975), wall shape modification (Carlson et al., 1957) etc., but the most common way is installation of screens in order to obtain a desirable velocity distributions. Hence, screens must be considered as an integral part of a wide angle cone design for an ESP installation and modelling of the wide angle inlet section is a prerequisite for modelling the gas distribution in an ESP.

Minute modelling of the exact gas distribution screen geometry is, even with large and fast computers, difficult and very time consuming (Arrondel et al., 2001). In standard commercial CFD codes the gas distribution screens are simulated by distribution of one-dimensional discrete drag forces. Further, a general extension of the standard model has been introduced (Nielsen et al., 2001) by including both drag and lift forces as discrete source terms for the momentum balance equations. In this model the drag and lift coefficients depends on flow angle of attack and screen design, which is included in the model via a database library with empirical coefficients.

The present paper presents numerical simulations of the gas flow of a full ESP including different inlet funnels with gas distribution screens, precipitation sections with collector curtains, anti-sneakage baffles, in one case with a gas distribution screen placed between two fields, hoppers with partition plates and outlet funnel with gas distribution screen. Further, particle movement has been considered with special attention to sneakage problems. Finally, numerical simulations versus full scale and model scale measurements are discussed.

2. ESP geometry, flow conditions and modelling strategy

The present study is based on application of the commercial CFD code STAR-CD (STAR-CD manual, 2001). Modelling fluid flow via CFD requires specification of the geometry through definition of a computational grid, the necessary models to present the physics of the problem, numerical solution strategy and specification of boundary conditions. Special treatment has been necessary in order to model the gas distribution screens and in the present case the source term model by (Nielsen et al., 2001) has been used. Furthermore, three-dimensional particle movement based on the calculated flow field have been considered.

2.1. ESP geometry and flow conditions

The modelled geometry (computational mesh) and by that the full ESP flow simulation model includes any detail that significantly affects the gas flow field. This includes guide vanes, gas distribution screens, collector curtains, anti-sneakage baffles, dividing plates in hoppers, and other flow obstructions. The different computational mesh used consists of between 1,500,000 and 2,500,000 computational cells depending on the actual case. The large number of computational cells results in very high resolution of the flow (or pressure) field in an ESP.

The mean gas velocity through the ESP is given in each separate case below but is approximately 1.0 m/s in the ESP gas passages (between collector curtains). At the inlet of the computational mesh a uniform velocity profile is specified. This is a common approach and in the case of ESP gas flow modelling it is expected to be a good approximation based on the fact that the downstream positioned gas distribution screens will control the flow development.

2.2. Modelling strategy

This section describes the background for the numerical analysis including model description and modelling procedure. Only the main principles are described and for a more detailed discussion of the screen model representing the gas distribution screens are referred to (Akoh, 1998) and for CFD in general to (Ferziger & Peric, 1996, STAR-CD manual, 2001, Wilcox, 1993).

The flue gas flow is modelled as steady, incompressible, and isothermal with fluid properties identical to a flue gas at the given ESP temperature. Turbulence is modelled by the standard k-ε model (Launder & Spalding, 1974), which is widely used for industrial internal flows and comprises differential transport equations for the turbulent kinetic energy k and its dissipation rate ε. The high Reynolds number form is used in conjunction with the so-called “law of the wall” representation of the boundary layer flow. This choice is based on the best compromise between accuracy and boundary layer resolution.

The Reynolds number between collector curtains of ESP gas flow studies is in the range of 8000 to 20000 ($Re = UD\rho/\mu$). Any changes in inlet temperature, volume flow rate etc. will affect the Reynolds number. But at the Reynolds number values present in the ESP gas passages the gas distribution will not be affected. Even at say 1/2 the design volume flow for the medium and high Reynolds number cases the gas distribution normalized with the mean velocity will be identical with the gas distribution at design conditions.

Collector curtains within the ESP sections are modelled as baffle-computational-cells (solid baffles) that are effectively zero-thickness, two-dimensional cells that otherwise act as solid cells and are placed at a distance equal to the actual spacing between collector curtains. Solid baffle computational cells which in general can be placed between two fluid cells are also used to model guide vanes, anti-sneakage baffles in the ESP top and bottom, center column, dividing plates in the hoppers, and other flow obstructions.

The inlet and outlet gas distribution screens are modelled by so-called source terms (Nielsen et al., 2001) that implement the forces from the screen acting on the flow without modelling the geometrical details of the screen. This effectively reduces the computational effort required. The model is three-dimensional and includes different body forces in all three momentum equations (x, y, z-directions) and empirical drag coefficients via a database. The database depends highly on the geometry of the screen and the angles of attach and are in the present case especially designed for FLS Airtech gas distribution screens. These screens are constructed as a number of vertical U-members (U cavity facing the gas flow direction) with throttle (cover) plates of different sizes depending on the opening area (porosity) of the screen in-between. Guiding plates may replace the throttle plates in a given number of places if a change of flow direction is what is wanted. In general this design is very flexible and depending on the needs a screen can easily be changed to different opening areas at separate places of the single screen by replacement of throttle plates or guiding vanes.

The boundary condition at the outlet specifies that the exit mass flow is fixed from overall continuity considerations, and at walls and at solid baffles “no slip conditions” are specified.

Overall it is believed that the numerical settings and models applied by the FLS Airtech ESP gas flow simulation model are the best possible in account of accuracy, methodology, and computer time.

2.3. Model approximations

The main purpose of the present study is the gas and dust distribution and the electrical field, discharge electrodes, and mechanical operations like rapping etc. are not considered.

The solid baffle model of the collector curtains does not accurately model the corrugated FLS Airtech collecting plate geometries but reflects the basic physics of the gas flow in the precipitator sections by taking into account the flow development between the plates and by that the resistance present. This approach may result in a large overall number of computational cells but the number can be limited by only using local mesh refinement between the plates (STAR-CD manual, 2001).

The study is limited to time independent (steady state) results. Clearly, complex cone flows suggest the need for unsteady calculations, however, investigations comparing steady state and unsteady results indicated that steady state calculations, although not fully converged (weak transient behaviour), could be performed with high accuracy (Nielsen, 2000).

2.4. Particle movement

The three-dimensional particle movement equations for a number of test particles have been solved by STAR-CD on the basis of ESP geometry and the calculated three-dimensional time mean gas velocity field. The basic assumptions of the particle model are that the flow is treated as a dilute suspension and that electrical effects are not included. The former assumption means that the influence of the particles on the gas flow is omitted and uncoupled calculations are carried out. First, the flow field is calculated as a pure gas flow. Second, a limited number of test particles are introduced in the solution of the gas flow and particle trajectories are calculated under the influence of particle inertia, gravitational forces and frictional forces resulting from gas flow relative to particle movement. The particle model assumptions are:

- Spherical particles without rotation but with friction resistance depending on the local slip velocity between flue gas and particle according to extended Stokes law,
- Influence of gas temperature is included in the friction law by gas density and viscosity,
- Influence of gravitational forces is included,
- Turbulent dissipation is not included,
- Particle-particle collision is not included,
- Collision with walls is expected to be sticky. Particles adheres to the wall,
- Particles are started with zero initial velocity.

3. Results and discussions

Four different cases have been considered, that of two full scale ESPs with different FLS Airtech inlet types determined by layout conditions (section 3.1), that of improving the emission by a full screen between different electrical fields (section 3.2), that of improving the emission by implementation of anti-sneakage baffles (section 3.3), and finally discussions of CFD results versus model and full scale measurements (section 3.4).

3.1. Case 1 – ESP gas distribution – Influence of inlet types

The gas velocity distribution within an ESP plays a role in overall performance. If local velocities are too high the residence time for sub-micron particles may be too short, and if velocities are very high there is a risk of scouring and re-dispersion of already precipitated particles. If local velocities are too low, particle space charge may be low too increasing local current densities thus leaving less current to other regions of the field. Low velocities may also be influenced by a high turbulence level giving rise to increased penetration. For these reasons proper design of flow control devices are critical within ESPs. Typical control devices are gas distribution screens, splitter plates and guide vanes. Until approximately 1995 physical model testing was the preferred tool to analyse ESP gas flow. Since that time CFD has proven successful and to day both model testing and CFD analysis are utilised for ESP gas flow tests.

Figures 1 and 2 shows two CFD models of two different ESPs. Figure 1 is a cement plant kiln/raw mill ESP while Figure 2 is a cement plant clinker cooler ESP. The models are fully three-dimensional and represents full-scale layout at the actual operating temperature. Hence no scale correction factors are needed in order to obtain accurate flow pictures. The most important internal geometrical details are included. Specifically the geometry of the kiln/raw mill ESP represents full scale dimensions of a 14.5 m high and 17.8 m wide FLS Airtech ESP with a short manifold with two sharp bends including guide vanes, two central type inlets including three gas distribution screens, four 4.5 m long precipitator sections (400mm channel width), anti-sneakage baffles, pyramid hopper system with dividing plates, and a single wide central type outlet. The mean velocity in the kiln ESP is 1.1 m/s. The geometry of the clinker cooler ESP represents full scale dimensions of a 15 m high and 15.5 m wide FLS Airtech ESP with one wide S-type inlet including guide vanes and two gas distribution screens, three 4.5 m long precipitator sections (400mm channel width), anti-sneakage baffles, pyramid hopper system with dividing plates, and two central type outlets. The mean velocity in the precipitator section is 1.0 m/s.

The inlet types of the kiln and clinker cooler ESP were determined by layout conditions. In the kiln ESP case a very short distance between the upstream Gas Conditioning Tower (GCT) and the ESP was necessary. A very short manifold connecting the GCT outlet with the two ESP inlet funnels was designed and the resulting gas distribution in the ESP was tested by CFD calculations. In the clinker cooler ESP case the FLS Airtech design is S-type inlet due to layout demands. Often ESPs for clinker coolers are placed beside the cooler and the raw gas duct may easily be connected to the ESP side. In the present case the inlet was very wide and the number of guide vanes and gas distribution screen configuration was optimised by CFD calculations.

Figures 3a and 3b shows velocity distribution in the precipitator of the kiln/raw mill ESP as axial iso-contour velocities in different cross sections (Figure 3a) and as velocity vectors at the selected transversal position 1/8 the ESP width from the wall (Figure 3b). The distribution before field 1 (inlet distribution) is quite uniform which the screens of the inlet funnel determines. The distribution after field 1 is a little skew with bottom peak velocities and depends partly on the inlet distribution and partly on the geometry of the hoppers. After field 2 and field 3 the effect of the resistance from the collecting plates has smoothed out the velocity distribution and the axial velocity is fairly uniform. The distribution after field 4 (outlet distribution) yields top peak velocities, which is determined by the screen of the outlet funnel. Note that this is more clearly seen on Figure 3a. The gas distribution in the ESP results in quite strong vortices in the first and last hopper but with the strongest vortex in the first hopper as expected. The simulation shows that indeed an acceptable gas distribution is obtained in the kiln/raw mill ESP downstream the special designed manifold.

Figures 4a and 4b shows velocity distribution in the precipitator of the clinker cooler ESP as axial iso-contour velocities in different cross sections (Figure 4a) and as velocity vectors at the selected transversal position 1/8 the ESP width from the wall (Figure 4b). The distribution before field 1 (inlet distribution) is, although with some small peaks, quite uniform which is determined by the screens of the inlet funnel. The distributions after field 1, 2 and 3 (outlet distribution), and in the hoppers have similar trends as the kiln/raw mill. The present simulation shows that 15 vanes are needed in the clinker cooler ESP S-type inlet in order to obtain an acceptable gas distribution before the two screens and by that before the field 1 of the precipitator.

It is noted, that the possibilities of changing the velocity distribution downstream the precipitator is highly related to the vertical direction and minor to the spanwise direction due to the vertical collecting plates ranging from the top beams to the dividing plates in the hopper. The only possibility of changing the spanwise distribution is between the three fields.

The gas distribution of both the kiln/raw mill and clinker cooler cases differs from the uniform distribution by changing the distribution from bottom peak velocities at the inlet field to a distribution with top peak velocities at the outlet field (uniform distribution in the middle).

3.2. Case 2 – Effect of gas distribution on emission - Screens between fields

It is well known and generally accepted that the dust distribution in ESPs is skew with bottom peak load. The skew distribution is more pronounced at the outlet field of the ESP and sneakage may be a problem. Sneakage problems are a result of the gas distribution and in due cause the standard FLS Airtech vertical outlet gas distribution has a maximum at the top in order to prevent sneakage at the bottom. A top peak velocity distribution at the ESP outlet can also be obtained by installing a full gas distribution screen between the last and second last electrical fields. The opening area of the screen must low at the bottom and high at the top in order to decrease sneakage of especially small size particles.

Particle movement as described in section 2.4 has been investigated by numerical calculations in a typical ESP design. Three groups of particles have been considered, that of particles below approximately 5 μm (small size particles), that of particles between approximately 5 μm and 25 μm (medium size particles), and that of particles above approximately 25 μm (large particles). The geometry of Case 2 is identical to the kiln/raw mill ESP of Case 1 but split up in two new designs and recalculated flow fields. One without anti-sneakage baffles and without a full gas distribution screen denoted Design A and one without anti-sneakage baffles but with a full gas distribution screen (upper 2/3 of screen height 60% open and lower 1/3 30% open) between the third and fourth electrical field denoted Design B. Particles are started at the inlet of the ESP and studied for constant particle density (ρ_p=2000 kg/m^3) and initial velocity ($v_{p,i}$=0 m/s).

Figure 5 shows particle movement, presented as particle trajectories, for small, medium and large size particles of Design A. Note that particles are defined as collected if they move inside field 2, 3 or 4. The study shows that many small as well as medium size particles are not collected because they move outside the electrical field. Large particles outside the electrical field are to some extent collected mechanically. Figure 6 shows particle movement of Design B for small size particles only. Medium and large size particles are of less interest to discuss because they are usually always collected either mechanically or electrically. A large amount of the small size particles present at the outlet field are collected in Design B (compare with upper Figure 5). But it is noted that the full screen must be closed at the top in order to prevent sneakage in this area. Hence if a full gas distribution screen that is 30% open at the bottom, 60% open at the middle, and closed at the top is placed between the last and second last electrical field sneakage problem becomes almost negligible which results in lower emission. It is noted that the “full screen” solution is especially useful in upgrading cases where the outlet funnel is without a gas distribution screen.

3.3. Case 3 – Effect of gas distribution on emission - Anti-sneakage baffles

An approach to prevent sneakage of small as well as medium and large particles in general in an ESP is introduction of anti-sneakage baffles above and below the electrical fields. As mentioned sneakage problems are a result of the gas flow pattern. Medium and large particles will not follow the flow pattern and are not expected to be a problem in view of optimal ESP operation because these are always collected. Furthermore, in absence of the electrical field investigation of these particles is trivial. Small particles, on the other hand, say with a diameter less than 5μm, will follow the flow pattern and may move to the electrical field or may pass outside the field depending on the anti-sneakage baffle design.

The geometry of the Case 3 is identical to the kiln/raw mill ESP of Case 1 but two different designs are compared. One without anti-sneakage baffles (Design A from Case 2 above) and one with anti-sneakage baffles at the top and bottom of the ESP (Design C, same flow field as Case 1). As in Case 2 particles are started at the inlet and studied for constant particle density (ρ_p=2000 kg/m^3) and initial velocity ($v_{p,i}$=0 m/s).

The degree of sneakage is defined as the ratio of particles passing outside the electrical fields to the total number of test particles started at the inlet. As in Case 2 please note that particles are stopped if they move inside field 2, 3 or 4 (defined as collected). Figure 7 shows particle trajectories for small size particles of Design C. In the case without anti-sneakage baffles (Design A, upper Figure 5) approximately 15 % of the released particles are passing outside the electrical fields. In the case with anti-sneakage baffles (Design C, Figure 7) below 1 % of the released particles are passing outside the electrical fields. Hence implementing anti-sneakage baffles can largely reduce sneakage problems resulting in lower emission.

Furthermore, calculations have also shown that installation of anti-sneakage baffles results in lower gas velocities in the area at the bottom of the collector curtains compared with cases without anti-sneakage baffles. This effect will in general decrease sneakage.

3.4. Case 4 – Discussion of CFD versus model and full-scale measurements

CFD studies usually have in the range of 1,000,000 to 3,000,000 control volumes (computational cells) that results in a highly detailed analysis of the flow within an ESP since the velocity and pressure are known at every cell. This high amount of data is impossible to obtain in full scale or even in a physical scale model. Furthermore, CFD studies resolve the gas flow distribution in three velocity components. Clearly the advantage of CFD studies versus measurements is the high resolution and the detailed description of the flow. CFD studies of turbulent gas flow in an ESP is a cost effective investigation tool that can partially replace physical model testing in the laboratory. This result in shorter delivery time from problem formulation to the result and at the same time eliminates the need for cost consuming model construction. Modification work only last a few minutes and a new case, or several new cases, can run during the night lasting approximately 8 to 12 hours for each calculation depending on the number of computational cells and the workstation used. Hence, a very short investigation time has been obtained which is very helpful especially in trouble shooting cases.

Modelling gas distribution screens placed in electrostatic precipitator inlet and outlet funnels is an important issue. The impact of the gas distribution screens on the flow depends on local flow properties, which necessitates exact geometric modelling of the screen in order to ensure trustworthy results. However, exact geometric representation is far beyond the capabilities of to-days computers. Instead source terms are added to the governing equations at the position of the screens. The magnitude of these source terms needs to be based on empirical relations determined experimentally or by local geometric modelling where single screen elements are modelled geometrically accurate. Studies have shown (Nielsen et al., 2001) that standard commercial models are not accurate. Instead empirical force coefficients in all three main flow directions are needed

For several years full-scale measurements have been carried out in ESPs. The measurements have taken place at the inlet (before field 1), between fields, and at the outlet (after last field) and are usually due to clients request compared with international accepted standards (ICAC, 1997). CFD calculations have shown that in many cases the velocity distribution before field 1, although gas distribution screens are present, to a large extend is prescribed by the angle of the wall of the inlet funnel. Hence especially in the cross section before field 1 flow angularity is an important issue because a not vanishing transversal velocity component is present. A vane anemometer is in general utilized for full-scale measurements. The vane anemometer does not provide accurate mean values of a particular velocity component but will detect the speed. Furthermore, the turbulence level is high before field 1 also resulting in inaccurate vane anemometer results. CFD studies can present the speed as well as all three-velocity components. The axial velocity should be the velocity component relevant for ESP gas distribution studies. The ICAC-EP-7 mentions, “Common velocity anemometers only measure the magnitude of the principal velocity component, and not the direction, or magnitude, of the true velocity vector”. And then streamers and smoke are recommended finding eddies and recirculating zones, but nothing about the direction of the velocity vector. The object ICAC-EP-7 with respect to determination of the gas distribution is solely to outline techniques and recommended procedures for model testing. But nothing about CFD is included.

In physical model scale tests velocities are typically measured with pitot-static tubes or hot wire (or hot film) anemometers. But more advanced equipment as Laser Doppler Anemometry (LDA) and Particle Image Velocimetry (PIV) systems may also be used to some extent. These systems give accurate values of velocity components as well as accurate values of turbulence. It is well known that the turbulence level is not well predicted by the standard k,ε turbulence model commonly used for CFD calculations of industrial flows. Hence if accurate turbulence levels are of interest model scale test are more accurate than CFD calculations.

In ESP gas flow CFD studies the most critical point is, as mentioned above, modelling of the diffuser flow present in the inlet funnel with gas distribution screens. Further the hopper flow with flow reversal may to some extent be critical. Physical model scale tests plays an important role regarding validation of the actual CFD calculation. The advantage of using a numerical approach is the detailed description of the flow and the possibility to investigate different designs without necessarily building them. On the other hand, depending on the actual flow problem, the limitations arising from modelling assumptions e.g. turbulence modelling makes it necessary to verify the ability of the numerical code to solve the actual flow field. However, only a limited number of measurements are necessary.

4. Conclusions

The present study has focused on modelling different full-scale ESPs. The gas flow distribution as well as particle transport has been investigated and discussed in terms of different designs. With increasing environmental requirements to reduce emission sneakage problems have become an important issue. This subject has been investigated by numerical modeling and discussed for different layouts. Finally the advances and limitations of numerical simulations as well as model and full-scale measurements have been discussed.

A number of parameters influence the efficiency of an ESP and by that the emission after the ESP. One parameter is the gas distribution, which is a subject of continuous discussion. But it is clear that a given ESP is designed with respect to emission according to a given gas distribution. The present gas distribution shows a rather uniform transversal distribution throughout the ESP while the vertical inlet distribution has a maximum at the bottom and the vertical outlet distribution has a maximum at the top.

This shape is in agreement with the standard FLS Airtech gas distribution that has proven to be efficient.

The study indicates that installation of a full screen between the last and second last field and especially installation of anti-sneakage baffles at the top and bottom of the ESP can large reduce sneakage problems and by that emission.

When using numerical simulation as an engineering tool modeling methods and smaller structural elements as for example the gas distribution screens should be considered carefully. Further comparison between numerical results and especially full-scale measurements should be evaluated. Anyway the present study has demonstrated that a numerical simulation of turbulent gas flow in an ESP is an effective investigation tool that can replace physical model testing in the laboratory, resulting in shorter delivery time from problem formulation to the result. Also, the effect of merely shaping velocity distribution, if this is what is wanted, can quickly be tested.

The study has been limited to separate investigation of gas distribution in an ESP but is a step forward towards a full ESP numerical simulation, which is presently under development including electrostatic forces, models for different electrodes, different collecting plates and re-entrainment from the collecting plates.

REFERENCES

Akoh, E., 1998, STAR-CD Source term model for lift and drag forces, DMI report: EFP-98/DMI-98860, Danish Maritime Institute, Lyngby, Denmark.

Arrondel, V., Bacchiega, G., Gallimberti, I., ESP Modelling: From University Studies to Industrial Application, Proceedings of 8th. International Conference on Electrostatic Precipitation, May. 14-17, 2001, Birmingham, Alabama, USA, Session A4-2.

Carlson, J.J., Johnston, J.P. & Sagi, C.J., 1967, Effects of Wall Shape on Flow Regimes and Performance in Straight, Two-dimensional Diffusers, Journal of Basic Eng., pp.151-160.

Egli, W. Kogelschatz, U. & Persson, T., Three-Dimensional Corona Current Distribution in Complex ESP Configurations, Proceedings of 6th. International Conference on Electrostatic Precipitation, June 18-21, 1996, Budapest, Hungary, pp.166-171.

Ferziger, J. H. & Peric,M., 1996, Computational Methods for Fluid Dynamics, Springer.

Goenka, L.N., Panton, R.L. & Bogard, D.G., 1990, Pressure and Three-component Velocity Measurements on a Diffuser that Generates Longitudinal Vortices, Journal of Fluids Eng., vol. 112, pp.281-288.

Houlgreave, J.A., Bromley, K.S. & Fothergill, J.C., A Finite Element Method for Modelling 3D Field and Current Distribution in Electrostatic Precipitators with Electrodes of Any Shape, Proceedings of 6th. International Conference on Electrostatic Precipitation, June 18-21, 1996, Budapest, Hungary, pp.154-159.

Institute of Clean Air Companies, Electrostatis Precipitator Gas Flow Model Studies, Publication EP-7, January, 1997.

Kline, S.J., 1959, On the Nature of Stall, Journal of Basic Eng., vol.81(D), pp.305-320.

Launder, B.E., and Spalding, D.B., 1974, The numerical computation of turbulent flow, Comp. Meth. in Appl. Mech. & Eng., 3, p. 269.

Lawless, P.A., 1996, ESPVI 4.0 Electrostatic Precipitator V-I Performance Model - User's Manual. Center for Aerosol Technology, Research Triangle Institute, North Carolina, USA.

Medlin, A.J., Fletcher, C.A.J. & Morrow, R., Electrohydrodynamic Modelling of Fine Particle Collection in Electrostatic Precipitators, Proceedings of 7th. International Conference on Electrostatic Precipitation, Sept. 20-25, 1998, Kyongju, Korea, pp.665-672.

Meroth, A.M., Nicolaus, S., Levin, P.L. & Schwab, A.J., Effective Solution of 3D Charge Coupled Problems in Electrostatic Precipitators, Proceedings of 6th. International Conference on Electrostatic Precipitation, June 18-21, 1996, Budapest, Hungary, pp.137-143.

Nielsen, N.F., 2000, Preliminary test simulations of modelling screens in electrostatic precipitators, FLS miljø report: Doc. 00-01208, FLS miljø, Valby, Denmark.

Nielsen, N.F., Lind, L., Akoh, E. & Hvid, S.L., Numerical modelling of gas distribution in Electrostatic Precipitators, Proceedings of 8th. International Conference on Electrostatic Precipitation, May. 14-17, 2001, Birmingham, Alabama, USA, Session A2-2.

Schmid, H.-J. & Buggisch, H., On the Modelling of Particle Transport in Electrostatic Pricipitators, Proceedings of 7th. International Conference on Electrostatic Precipitation, Sept. 20-25, 1998, Kyongju, Korea, pp.673-687.

Schmitz, W., Gibson, D. & Pretorius, L., Computational Fluid Dynamics Modelling of Collection Dynamics, Proceedings of 8th. International Conference on Electrostatic Precipitation, May. 14-17, 2001, Birmingham, Alabama, USA, Session A2-1.

STAR-CD manual, version 3.15, 2001, Computational Dynamics Ltd., London, UK.

Wilcox, D.C. 1993, Turbulence Modeling for CFD, DCW Industries, Inc., California, USA.

Yang, T. & El-Nasar, A.M., 1975, Slot Suction Requirements for Two-dimensional Griffith Diffusers, Journal of Fluid Eng., vol.97, pp.258-260.

Zamany, J., 1992, Modelling of particle transport in Commercial Electrostatic Precipitators, Ph.D Thesis, ATV EF 316, Technical University of Denmark and FLS miljø a/s.

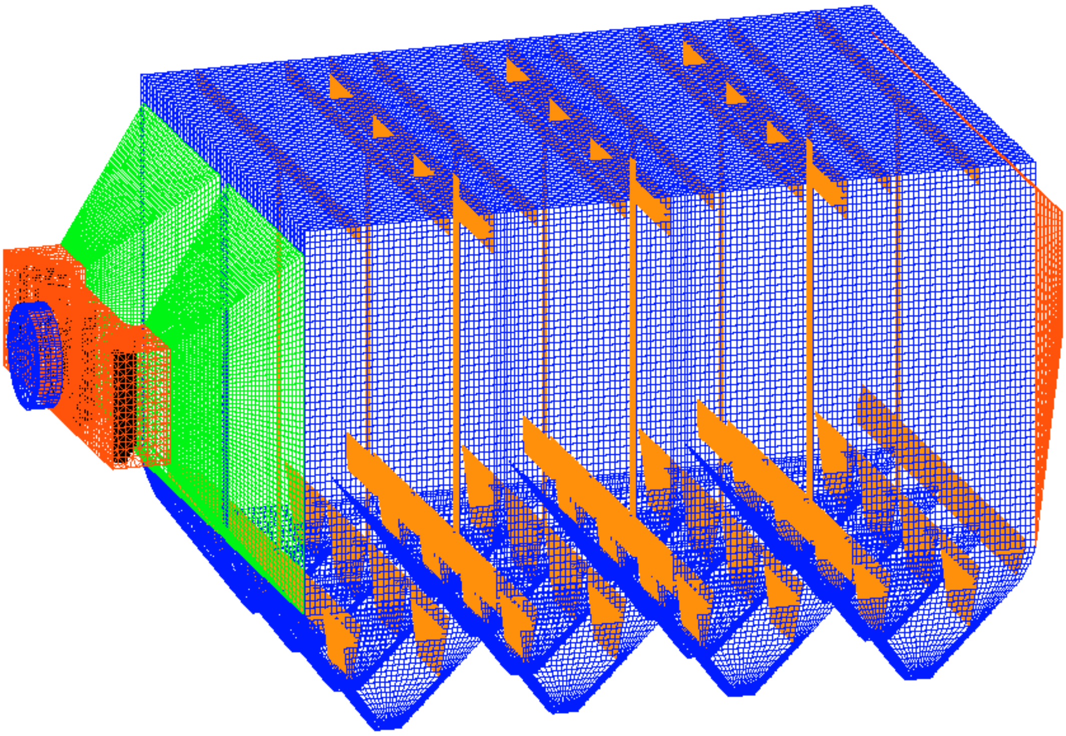

Figure 1. Computational mesh of cement plant kiln/raw mill ESP. The computational mesh consists of approximately 2,200,000 computational cells and includes inlet manifold with guide vanes, two inlet funnels each with three gas distribution screens, ESP section with collector curtains, anti-sneakage baffles, center column, hoppers with dividing walls, and a single wide outlet funnel with one gas distribution screen. The mesh is shown without gas distribution screens and collector curtains for clarity. Flow is from left to right.

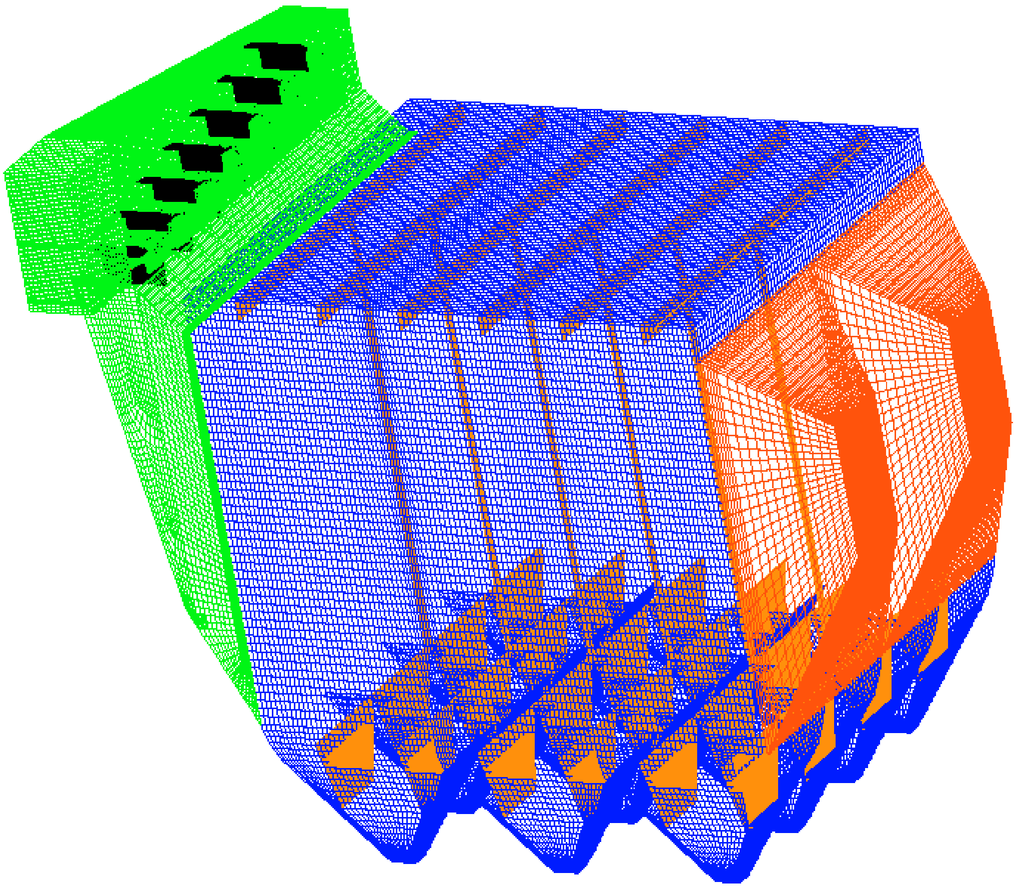

Figure 2. Computational mesh of cement plant clinker cooler ESP. The computational mesh consists of approximately 1,550,000 computational cells and includes inlet funnel with guide vanes and two gas distribution screens, ESP section with collecting curtains, anti-sneakage baffles, center column, hoppers with dividing walls, and outlet funnel with one gas distribution screen. The mesh is shown without gas distribution screens and collector curtains for clarity. Flow is from left to right.

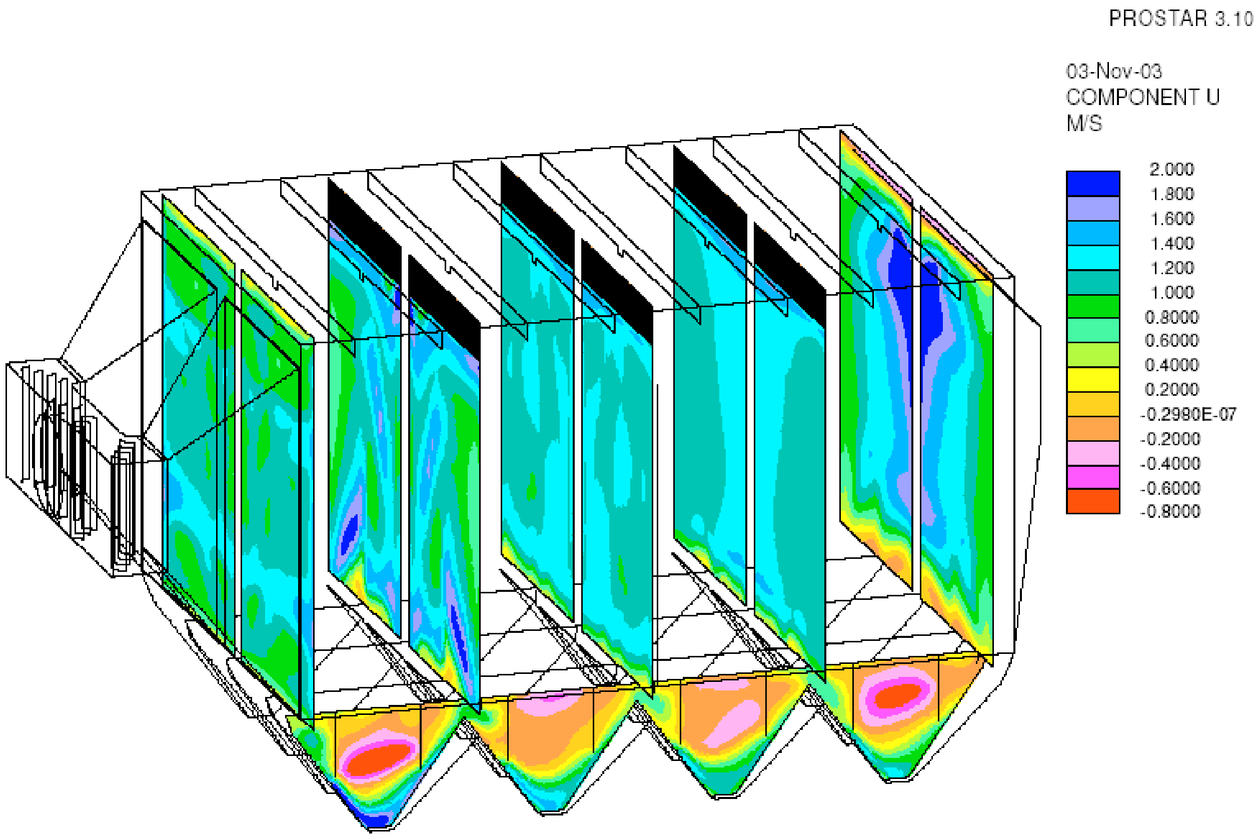

Figure 3a. Axial velocity distribution in the precipitator section shown before field 1 (inlet), after field 1, after field 2, after field 3, after field 4 (outlet), and in the hoppers. The kiln/raw mill ESP inlet funnels are each fitted with three gas distribution screens and the outlet funnel with one screen. Note that the mean axial velocity in the precipitator section is 1.1 m/s.

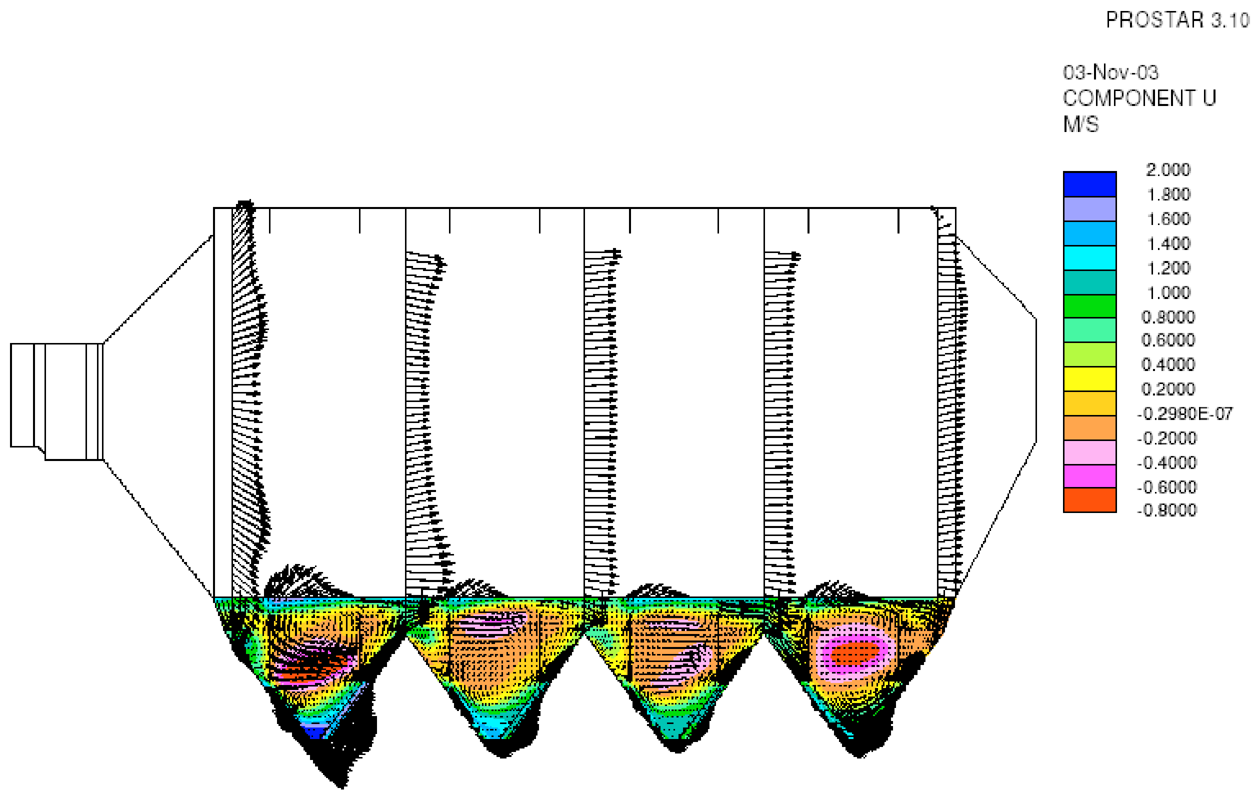

Figure 3b. Vertical velocity distribution (velocity vectors) in different cross sections of the precipitator section and in the hoppers. Transversal position equals 1/8 the ESP width from the wall. Distributions in ESP are shown before field 1 (inlet), after field 1, after field 2, after field 3, and after field 4 (outlet). Distribution in hoppers is shown as axial iso-contours as well as velocity vectors.

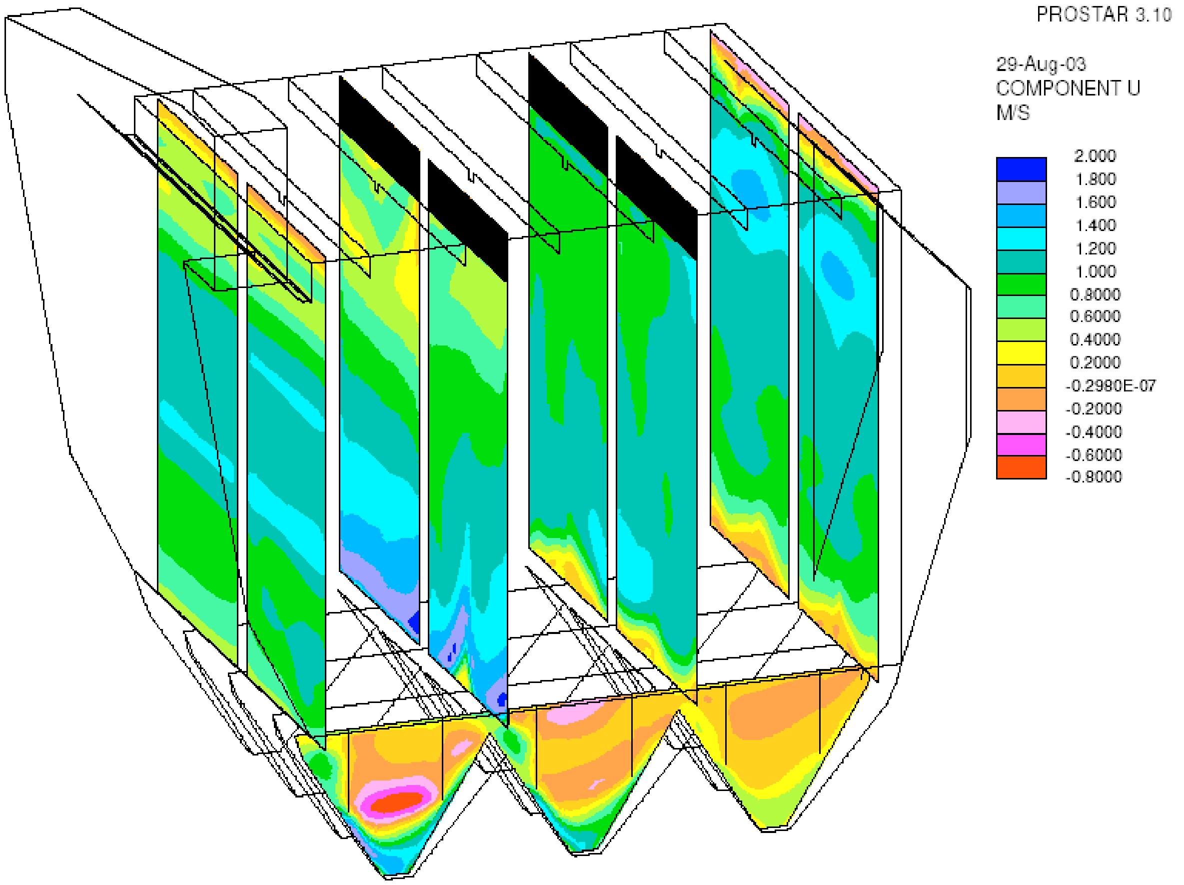

Figure 4a. Axial velocity distribution in the precipitator section shown before field 1 (inlet), after field 1, after field 2, after field 3 (outlet), and in the hoppers. The clinker cooler ESP inlet funnel is fitted with guide vanes and two gas distribution screens and each of the outlet funnels with one screen. Note that the mean axial velocity in the precipitator section is 1.0 m/s.

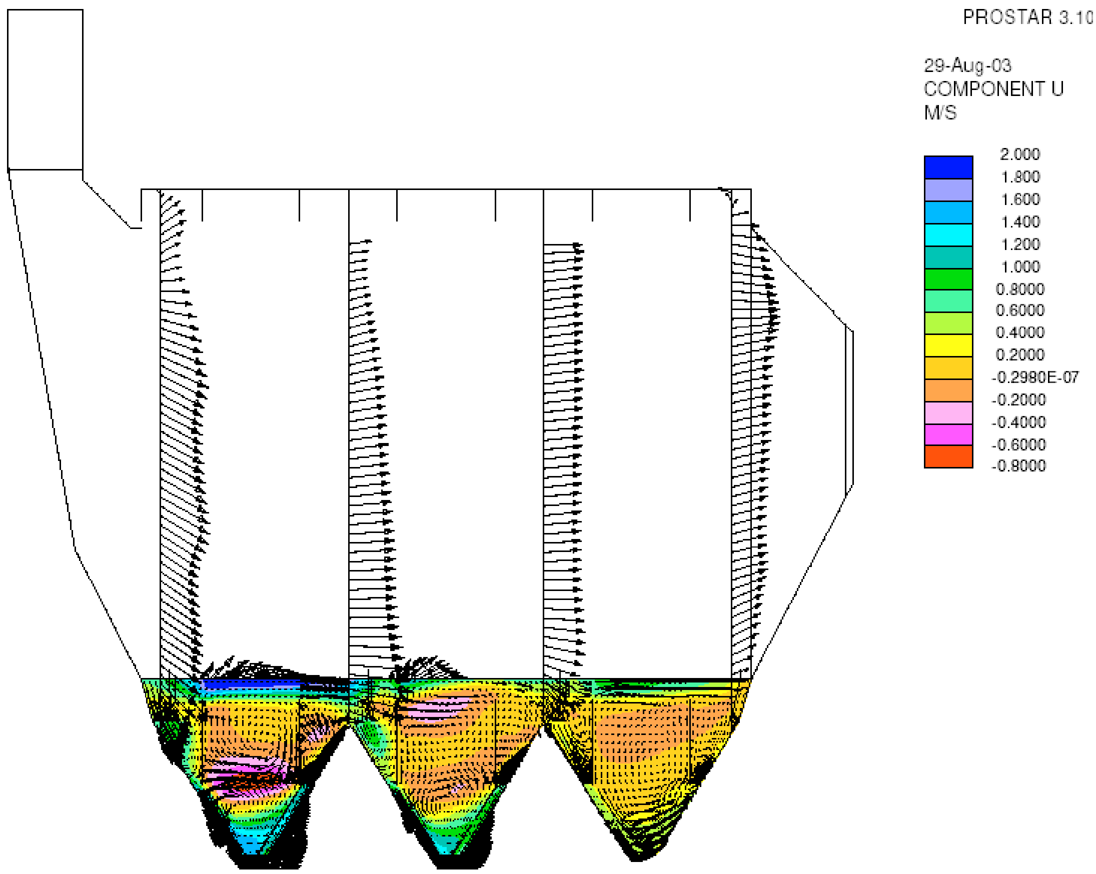

Figure 4b. Vertical velocity distribution (velocity vectors) in different cross sections of the precipitator section and in the hoppers. Transversal position equals 1/8 the ESP width from the wall. Distributions in ESP are shown before field 1 (inlet), after field 1, after field 2, and after field 3 (outlet). Distribution in hoppers is shown as axial iso-contours as well as velocity vectors.

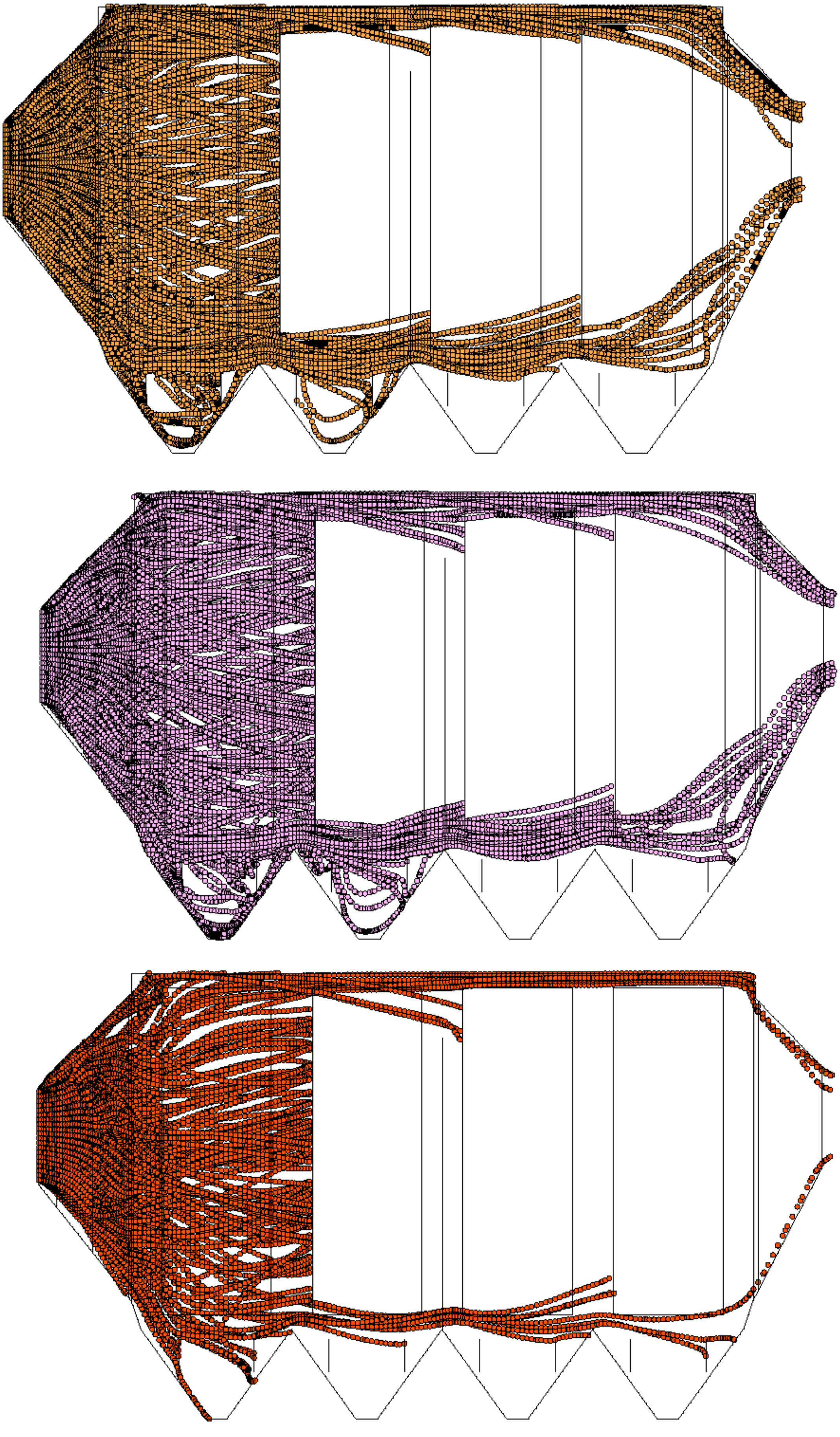

Figure 5. Particle movement of small size particles (d_p<5μm, upper Figure), medium size particles (5μm<d_p<25μm, middle Figure), and large size particles (d_p>25μm, lower Figure) for case without anti-sneakage baffles and without any screens between electrical fields (Design A). Flow is from left to right.

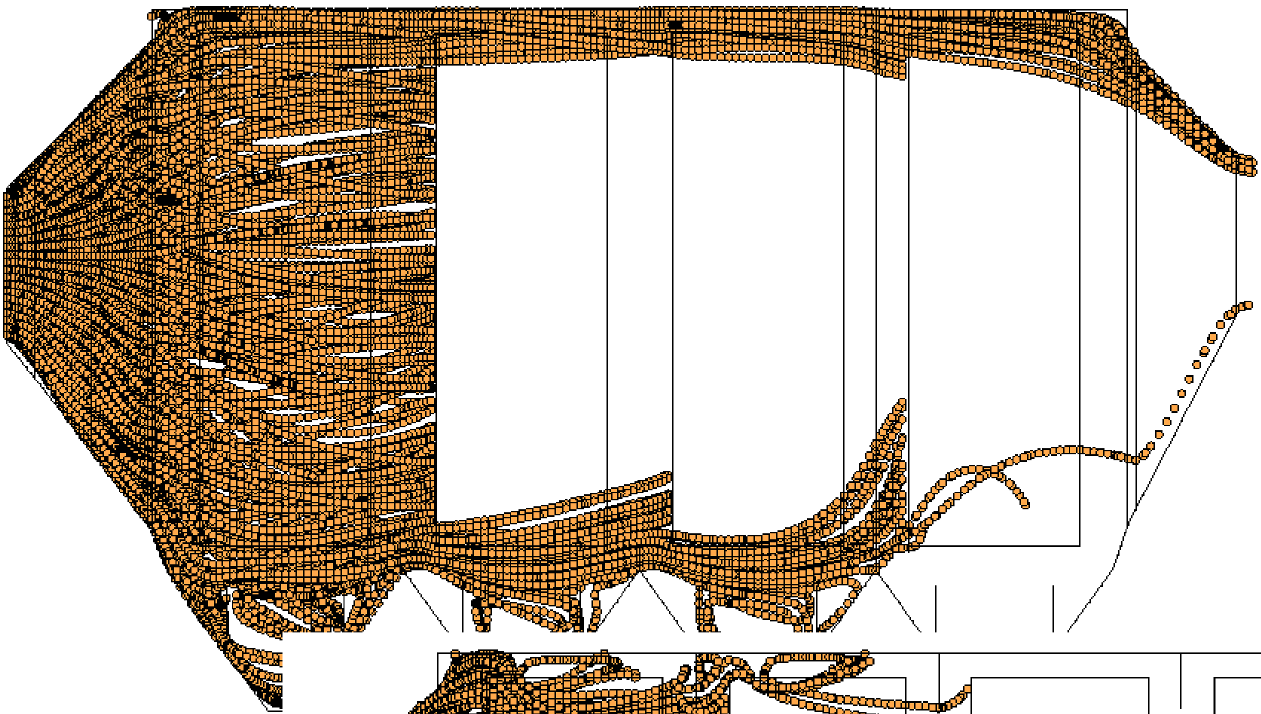

Figure 6. Particle movement of small size particles (d_p<5μm) for case with a full screen placed between last and second last electrical field (Design B). Flow is from left to right.

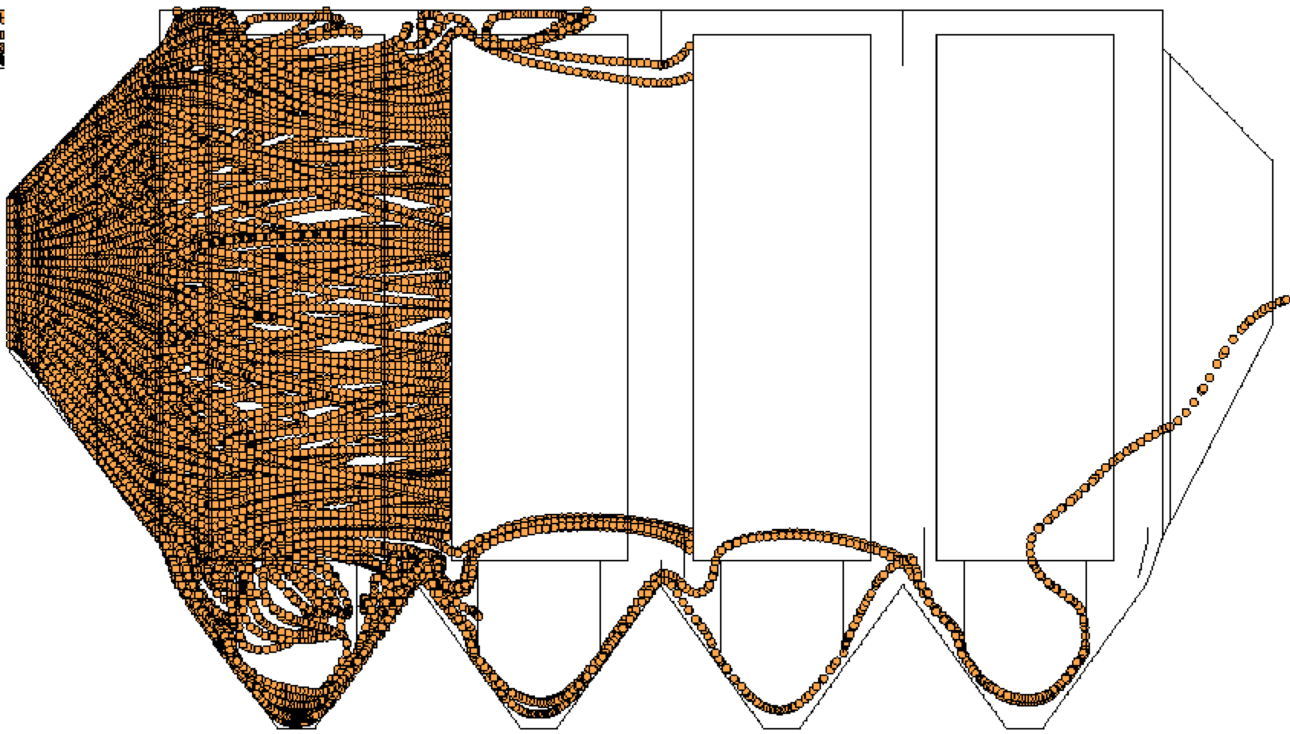

Figure 7. Particle movement of small size particles (d_p<5μm) for case with anti-sneakage baffles placed at top and bottom of ESP (Design C). Flow is from left to right.