Improved Methodology for Accurate CFD and Physical Modeling of ESPs

A. Banka[1], B. Dumont[1], Dr. J. Franklin[1], G. Klemm[2], R. Mudry[1]

[1] *Airflow Sciences Corporation, Livonia, Michigan, USA*

[2] *Southern Company Services, Atlanta, Georgia, USA*

Corresponding author*: rmudry@airflowsciences.com*

Keywords: Electrostatic Precipitator, Flow Modeling, CFD, Physical Modeling, ICAC EP-7; Thin Resistances

Introduction

Flow modeling of Electrostatic Precipitator (ESP) systems to achieve uniform gas distribution has been conducted for decades[1,2]. Often the modeling is deemed successful if the ESP startup performance meets the emissions reduction guarantees. Closer inspection of available flow distribution data from models and actual plant measurements, however, tend to show mixed levels of agreement[3]. In many cases, the correlation between models and actual plant data is still suboptimal when looked at from a detailed statistical view. In particular, correlation between Computational Fluid Dynamics (CFD) and scaled physical models is often not as strong as desired to determine which model provides a better result.

To help resolve the discrepancies between modeling methods, detailed research has been performed on a key geometry element of the ESP flow system design: the perforated plate. Also called distribution grids, screens, or simply perf plates, these devices provide flow resistance and back pressure to flatten out the velocity profile. Correct simulation of these perforated plates and their flow resistance is critical to achieving accurate model results and obtaining the end goal of a uniform velocity distribution in the ESP.

The research indicates potential flaws in current modeling methods for both CFD and scaled physical models that affect accuracy. As pollution reduction requirements become more stringent and plant operating conditions more varied, the need for more accurate flow model results increases. Using the current research, an improved modeling methodology for perforated plate simulation is suggested based on the findings. This methodology is intended to provide more accurate flow models, leading to enhanced ESP designs and improved operation.

ESP Flow Modeling Challenges

The methods used for physical and CFD modeling of ESPs are quite well-documented in industry literature[3,4,5]. Like all engineering design, these techniques have their own technical limitations from a general sense[6,7]. There are numerous articles written about "best practices" for maximum accuracy. For CFD, accuracy considerations include the mesh resolution, cell topology and flow alignment, solver numerics, selection of turbulence model, steady versus transient solution, and geometry details simulated. For physical modeling, best practice considerations include measurement instrument selection and proper usage, data collection methods, correct fluid dynamic similarity, and the geometry details simulated.

Besides the technical influences, an ESP modeling effort must also consider non-technical factors, such as schedule and budget. These commercial factors, of course, tend to work counter to the technical factors. The end goal is thus to obtain a sufficiently accurate flow model result allowing the ESP to be optimally designed within an affordable and tractable timeline.

Identifying the Problem

The ESP inlet ductwork generally has high velocity flow to convey the particulate, and then the cross section expands by a factor of ten or more in order to reduce gas flow velocities through the ESP collection region. These large expansions contain various flow control devices such as vanes, baffles, and one or more perforated plates.

In assessing correlation between CFD models and physical models, compared to both plant data and compared to each other, it has been observed by the authors that ESP inlet geometry plays a key role. Correlation appears much better for "square-style" ESP inlets (Figure 1). The gas flow travels from above or below the ESP, then encounters a 90 degree elbow where it both expands and turns towards the ESP. The elbow generally has a series of turning vanes, also called ladder vanes, and one or more perforated plates to condition the flow prior to entering the ESP collection region.

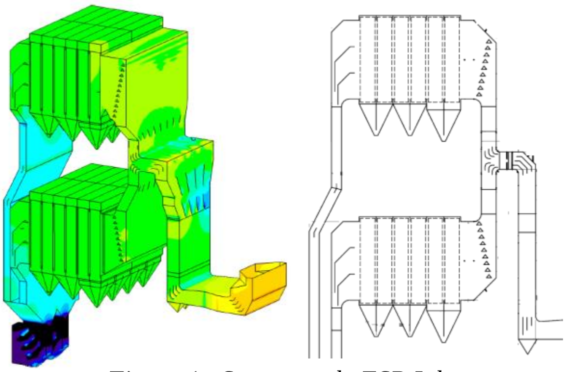


Figure 1. Square-style ESP Inlet

On the contrary, model predictions at the ESP inlet do not correlate as well for "diffuser-style" ESP inlets (Figure 2). Also referred to as a "funnel-style", "nozzle-style", or "camera-style" inlets, the flow travels horizontally through the ductwork and expands both vertically and horizontally in the diffuser. Two or more perforated plates are generally located within and at the end of the diffuser, and some designs have further flow devices such as vanes and baffles.

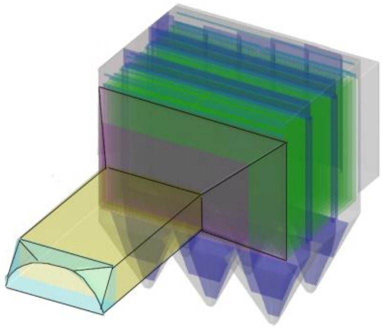


Figure 2 – Diffuser-style ESP Inlet

ESP outlet geometry can be of either type, a square-style contraction with flow turning up/down, or a horizontal contraction, the inverse of the diffuser-style. Interestingly, the authors have observed that flow model predictions of velocity at the ESP outlet generally correlate well, regardless of the ESP inlet or outlet geometry. The evidence thus points to the complex geometry of the diffuser-style ESP inlet as the problem. It is difficult for the models to predict flow behavior in the diffuser-style inlet.

It is well-known in CFD modeling that the solution algorithms can create a phenomenon called "false diffusion"[7] that degrades accuracy when the flow directionality and the CFD mesh are misaligned. Thus it is understandable that velocity patterns in the inlet diffuser, with highly angular flows in two directions, plus the potential for separated/reverse flow, can be challenging to predict accurately. CFD experts rely on meshing experience and best practices to counter the false diffusion effects, using high cell count and wise selection of cell shape, preferring hexahedral and polyhedral and minimizing tetrahedral topologies[ref?].

Physical modeling, on the other hand, does not have the mesh-dependence of CFD, but still suffers from sub-optimal correlation of velocity predictions at the ESP inlet for diffuser-style geometries. Further, the CFD and physical models tend to differ oppositely, with physical models underpredicting expansion and CFD models overpredicting expansion.

In examining accuracy of over 100 ESP models over the past 25 years, the authors began to focus not on simply the geometry of the inlet, but on the angular flow through the perforated plate as a key factor that affects accuracy for both modeling methods.

A commonality is noted in that flow approaching the perforated plates of the square-style inlet is primarily axial to the plate. Similarly, flow approaching the exit perforated plate of any ESP geometry is primarily axial. Meanwhile, flow through the perforated plates of a diffuser-style inlet can have significant angularity. An investigation thus ensued to better understand why current methods for both CFD and physical models have difficulty predicting the behavior of angular flow through perforated plates**.**

Modeling Representation of Perforated Plates

The typical representation of perforated plates in flow models, whether CFD or scaled laboratory models, is based on the loss coefficient found in industry handbooks[8]. The physical modeler selects commercially available punched perforated plates that match the loss coefficient of the actual plates installed in the full scale ESP. The physical model plate porosity (% open area) is often very close to the full scale, with adjustments made as required for scaling parameters such as Reynolds number and geometry.

For CFD models one could attempt to model the ESP inlet diffuser with an extremely high mesh resolution to obtain an accurate result. This would mesh the full detail of a perforated plate, including all the holes and plate thickness. This, however, leads to an intractable model size in terms of cell count, likely on the order of one billion (1e9) or more cells. Such a model could be run on a supercomputer cluster, but costs would be high, on the order of USD$5-10,000 to run each simulation. Combine that with the fact that it often requires 10-20 simulations to optimize an ESP design, the overall computational costs would be staggering. This does not include the labor required to mesh such a model, which likewise would be costly.

Thus, the conventional CFD approach uses the "porous jump" to represent the resistance of a perforated plate over a cross section. The porous jump provides a loss coefficient, based on a handbook value, for a given perforated plate porosity.

It is noted that the basis for the handbook loss coefficients is from laboratory experiments in a smooth, horizontal wind tunnel with axial flow passing straight through the plate. Though this may match the flow behavior for a square-style ESP inlet or an ESP outlet, this does not match the complicated geometry of a diffuser-style ESP. The loss coefficient derived from industry handbooks is thus not necessarily applicable to the perforated plates of a diffuser-style ESP inlet.

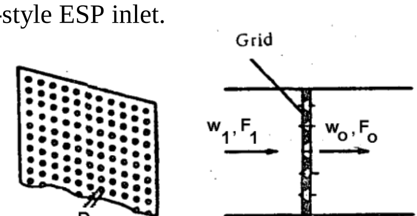
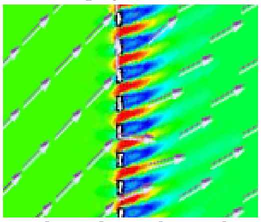


Figure 3. Standard Wind Tunnel Test of Perforated Plates (from reference 8)

Further, it has been observed through CFD, scale models, and field testing of ESPs that flow traveling through a perforated plate at an incoming angle experiences not just a back pressure due to the loss coefficient, but a straightening effect that alters the flow direction (Figure 4). This is due to the plate thickness and is generally not considered in standard industry practice for physical or CFD models.

Inlet Angle 40°



Outlet Angle 28°

Figure 4 –Angular Flow Through a Perforated Plate

Angular Flow Wind Tunnel Testing

To examine the detailed behavior of angular flow through perforated plates, a custom wind tunnel was constructed (Figure 5). The design features a rotating test section, allowing the incoming flow angle to be varied. The test section, where the perforated plate is mounted, is a 12” x 6” (305mm x 152mm) cross section. The tunnel was a once-through design with an inlet bell-mouth and flow conditioning to smooth out the velocity profile. A centrifugal fan with VFD flow control is located at the tunnel exit.

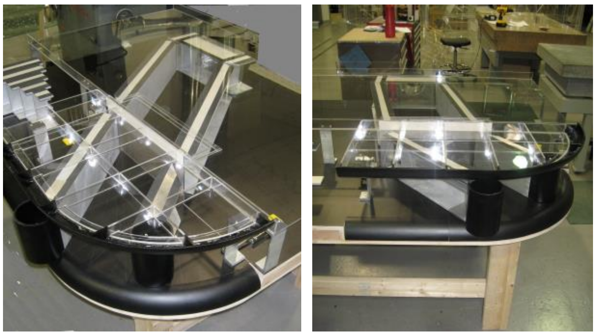


Figure 5 –Angular Flow Wind Tunnel Design

Testing was conducted on a number of typical perforated plates used for scale physical modeling of ESPs. The porosity, or % open area, ranged from 10% to 60%. Each plate was tested at 5 air flow velocities and 7 different incoming flow angles. Measurements included the pressure drop across the plate along with the incoming and outgoing flow angle. A nulling procedure was used to ensure that the wind tunnel walls did not introduce a straightening effect. Thus, only the straightening effect of the perforated plate was measured. Flow visualization from the wind tunnel testing is shown in Figure 6.

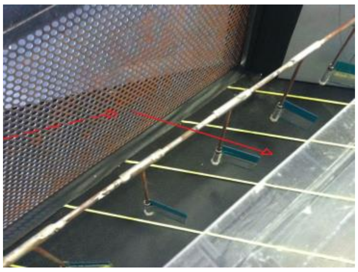


Figure 6 – Angular Wind Tunnel Flow Visualization

A sampling of the test results showing loss coefficient versus incoming flow angle are provided in Table 1. For reference, Plate 13 and Plate 15 are both 40% open with 0.125” (3.2mm) holes. Plate 13 is 0.06” (1.5mm) thick while Plate 15 is 0.03” (.76mm) thick). The data clearly indicate that the loss coefficient of the perforated plate, thus the backpressure influence, varies with incoming angle.

Plate	Angle In	Loss Coeff	Plate	Angle In	Loss Coeff
13	0	2.76	15	0	3.52
13	10	2.94	15	10	3.48
13	20	3.27	15	20	3.54
13	30	3.43	15	30	3.57
13	40	3.71	15	40	3.91
13	50	4.39	15	50	4.44

Table 1. Angular Wind Tunnel Data: Loss Coefficient

Also of interest is the straightening effect of the perforated plate. This is found to be a function of the plate thickness (L) and hole diameter (D). The ratio of L/D is a good parameter to compare straightening effect against, shown in Table 2 for Plates 13 and 15.

Plate	% Open	L/D	Angle In	Angle Out
13	40	0.48	10	7.9
13	40	0.48	20	14.0
13	40	0.48	30	21.9
13	40	0.48	40	25.9
13	40	0.48	50	32.6
15	40	0.24	10	8.7
15	40	0.24	20	15.5
15	40	0.24	30	24.0
15	40	0.24	40	30.9
15	40	0.24	50	35.8

Table 2. Angular Wind Tunnel Data: Straightening

In all, 44 perforated plates have been tested to date.

Physical Modeling of Various Perforated Plates

A 1/8 scale physical model of a biomass boiler ESP was selected for evaluation with the angular wind tunnel data for an actual ESP. This ESP featured a diffuser-style inlet, and is shown in Figure 7.

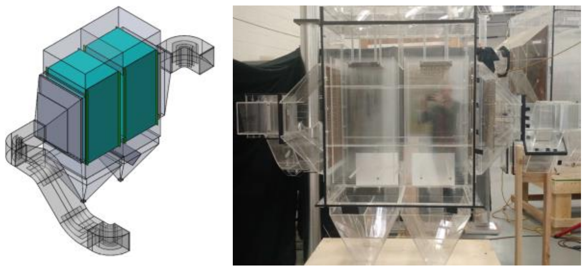


Figure 7 – 1/8 Scale ESP Physical Model

The full scale ESP was an existing unit undergoing flow modeling for a rebuild. There were 2 perforated plates in the inlet diffuser; the first was 47% open and the second was 40% open. The outlet perforated plate was also 40% open. All plates had 3” [75mm] holes and 3/8” [9.5mm] thickness (L/D = 0.125).

For the physical modeling, a baseline model of the existing ESP was performed to determine current flow distribution. The model was then used to redesign the perforated plates and flow devices to optimize performance for the ESP rebuild.

As an experiment, the baseline was run with two variations to the second perforated plate to determine L/D effect on velocity patterns at the ESP inlet. The porosity remained consistent at 40% open, the thickness was constant, but the hole diameter (and thus L/D) was varied. Table 3 shows model results relating uniformity statistics for these two different variants of perforated plate 2. The flow statistics include the ICAC uniformity criteria[5] and the %RMS (also called CV) uniformity[3].

Perforated Plate 2 Details			ICAC criteria		%RMS
Variant	% Open	L/D	115%	140%	(CV)
1	40	0.58	57	66	52.5
2	40	0.14	61	75	58.6

Table 3 – Physical Model Results with Varied Inlet Perforated Plate

Looking generally at Table 3, besides seeing that the baseline flow distribution is poor, one might conclude that the variation in plates of this experiment shows that L/D only has a slight influence. The %RMS is similar, and the ICAC statistics are different but not significantly.

The velocity profiles of Figure 8, however, tell a completely different story. Because the flow at the top and bottom of the ESP enters perforated plate 2 at a considerable angle from horizontal, the loss coefficient and straightening effect vary greatly. With the high L/D perforated plate (Figure 8, left), the velocity profile is higher in the middle. The plate has strong straightening effect and starves the upper and lower regions of the ESP. With the low L/D perforated plate (Figure 8, right), the opposite is seen. The plate does not straighten the flow significantly, and thus flow continues in an angular direction up/down. The result is a velocity profile with high zones near the top/bottom and low in the middle.

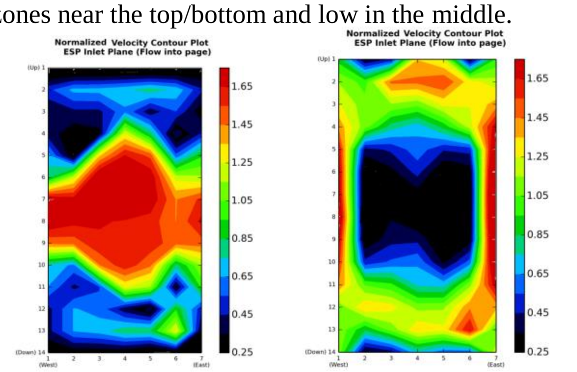


Figure 8 – Physical Model Inlet Velocity Profile with Left: Variant 1 of Perforated Plate 2, L/D=0.58 Right: Variant 2 of Perforated Plate 2, L/D=0.14

The results of this experiment support the angular wind tunnel test results, and indicate how greatly the loss coefficient and straightening effect can influence ESP flow uniformity. The results also show the danger of only looking at uniformity statistics. These can be deceiving because there are many different ways to achieve a particular statistical uniformity. One must look at the velocity profile when correlating models to each other. Finally, this experiment shows the importance of perforated plate selection in physical models. Not every 40% open perforated plate is the same.

Improved Methodology – Physical Modeling

The current research shows that both the loss coefficient and L/D influence ESP velocity patterns greatly. Thus, a preferred modeling methodology is to match both these values as closely as possible between the model and the full scale ESP.

Although this seems logical, it has not been practiced to large extent in the past. Model perforated plates have been generally selected based on matching the percent open area (and thus loss coefficient), without significant consideration of L/D. This is because matching the L/D is not necessarily straight-forward. It is common practice in the physical modeling to utilize commercially available sheet metal perforated plates. The problem is that the most available sheet metal plate is too thick, from a scale perspective, making the L/D in the model greater than the full scale.

Typical full scale ESP perforated plates have holes that are 1.5-3” (38-75mm) diameter and are of 3-10 gauge steel, which is 0.14-0.25” (3.6-6.4mm) thick. Thus, L/D of full scale perforated plates is on the order of 0.1. Porosity generally ranges from 20-50 percent open area.

Commercially-available sheet metal perforated plate can be obtained in the same porosity, but the L/D tends to be much larger than the full scale ESP. This is mostly due to the thickness, which tends to be 16-24 gauge, or 0.02-0.06” (0.6-1.6 mm). This is much larger than scale would dictate. The hole diameter varies, with smaller holes generally used on the lower porosity plates. This results in an L/D on the order of 0.2-0.5 for the higher porosity (40-50% open) and 0.5-0.7 for the lower porosity (20-30% open). Thus, physical model perforated plates tend to straighten the flow more than the actual full scale ESP plates.

An example of this is shown in Figure 9, a comparison of physical model results to actual field data on an ESP. In this case, the physical model uses traditional off-the-shelf perforated plates with L/D on the order of 0.5, while the actual ESP has perforated plates with L/D on the order of 0.1. Figure 9 (left) shows the physical model velocity profile in the inlet collection field. Deviation from average velocity is indicated by the color contours. Figure 9 (right) shows the corresponding field test on the ESP using vane anemometers to measure velocity. The plant data show higher velocities in the upper and lower regions of the data plane, and lower in the middle. This is typical of a case where the physical model, with larger L/D, underpredicts the flow expansion in the ESP inlet diffuser.

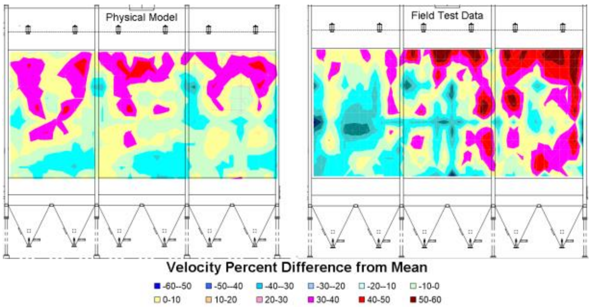


Figure 9 – ESP Inlet Flow Deviation
Left: Physical Model
Right: Plant *Field Test Data*

Ideally, selecting perforated plates for the model with larger holes would help offset the thickness differential and match the L/D better. This is the key recommendation of the improved methodology for physical modeling.

This approach may require custom fabricated perforated plates rather than using off-the-shelf sheet metal plates. Unfortunately, this would increase the costs for a physical modeling effort. This again points to the balance between modeling cost and accuracy, which has to be judged on a case-by-case basis.

CFD Modeling with Porous Jump

As described previously, the standard CFD method of representing a perforated plate is via a porous jump. This is prescribed on a planar surface in the model at the perforated plate location, and applies a resistance coefficient in the perpendicular direction to that plane. The momentum source terms at the plane are modified internally in the solver to account for the resistance and pressure loss across the plane. This does affect the flow three-dimensionally, but unfortunately not in the same manner as suggested by the angular wind tunnel data. There is no influencing parameter for the L/D of the perforated plate holes. Because of this, the porous jump causes the CFD model to overpredict the flow expansion in the ESP inlet.

As an example of this, Figures 10 and 11 show results of a CFD model for the geometry of Figure 1. The CFD model had a mesh of approximately 18,000,000 cells. Flow simulations were run with two different polyhedral CFD solvers: Azore® (by Airflow Sciences Corporation) and Fluent® (by ANSYS Corporation). The results between the two codes were nearly identical. Figure 10 shows the CFD results in the side view. The flow expansion through the inlet diffuser and perforated plates is indicated by both color contours and velocity vectors. Figure 11 provides the CFD model prediction of velocity distribution at the ESP inlet plane. High velocities are shown in the upper and lower regions, with lower velocities in the middle. This is typical overexpansion through the perforated plates, as the porous jump does not include the straightening effect.

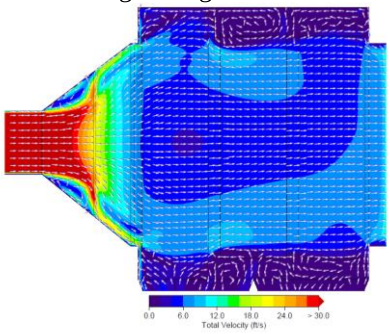


Figure 10 – CFD Results with Standard Porous Jump, Side View

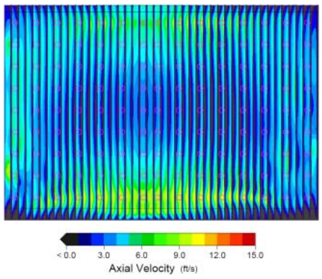


Figure 11 – CFD Results with Standard Porous Jump, ESP Inlet Plane

Improved Methodology – CFD Modeling

Several different methods were examined via CFD to include the influence of the perforated plate L/D. A volumetric resistance, with a depth in the flow direction, was attempted, but the thickness of said volume is very small and contributes negatively toward mesh resolution. This method also requires a specific loss coefficient to be prescribed in the cross-ways directions, rather than being a function of the incoming flow angle as observed in the wind tunnel. This method did qualitatively appear to improve predictions, but was not sufficiently rigorous to be defended.

To properly account for the L/D straightening effect, a custom logic was incorporated into the CFD solver that modifies the fundamental relationship between the momentum and the pressure across the plane of the perforated plate. The empirical data from the angular wind tunnel test is used to prescribe the functionality between incoming flow angle, loss coefficient, L/D, and exit flow angle. The pressure and momentum are modified based on these parameters.

An advanced methodology for CFD modeling of ESP perforated plates is thus introduced. Rather than a standard one-dimensional loss coefficient, or "porous jump", the Azore CFD solver was modified such that the loss coefficient can have a three-dimensional influence on the flow. This better represents the flow-straightening effect of perforated plates when flow approaches the plate at an angle, without having to mesh the details of the plate holes and thickness.

An example of the first implementation of the new perforated plate logic is shown in Figures 12 and 13. This uses the same CFD model as described above related to Figures 10 and 11. Instead of a standard porous jump, with no influence of L/D straightening effect, the new logic was used and the L/D of the full scale plant perforated plate was implemented (L/D=0.12). This results in the model predicting some straightening, instead of no straightening. Figure 12, the side view, can thus be compared to Figure 10. Figure 13, the ESP inlet velocity profile, compares to Figure 11. It is evident that the flow expansion is lesser with the new perforated plate logic. In the standard porous jump (Figure 10), the flow angle does not change at the perforated plate location. With the new methodology (Figure 12), the flow angle downstream of the perforated plate is roughly 10 degrees less, and matches the wind tunnel data for this particular L/D. The result is a flow profile at the ESP that does not over predict the expansion.

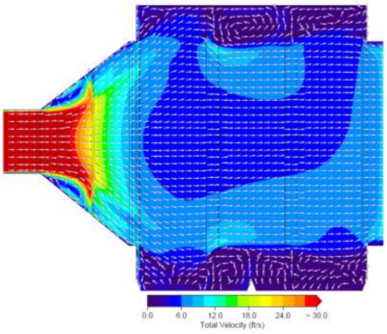


Figure 12 – CFD Results with New Perforated Plate Logic (Plant L/D=0.12), Side View

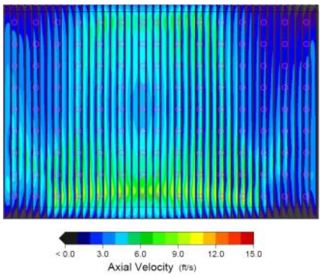


Figure 13 – CFD Results with New Perforated Plate Logic (Plant L/D=0.12), ESP Inlet Plane

The new perforated plate logic was also run simulating a typical physical model. In this case the plate L/D was 0.54, considerably larger than the actual plant. Results are shown in Figures 14 and 15. In this case, the flow expansion is underpredicted due to the more significant straightening effect of the higher L/D plate in the physical model.

Though not tested with enough cases yet, this new logic is a step in the right direction for perforated plate representation in a CFD model. The methodology is still under development, and additional comparisons need to be conducted between the new CFD method,

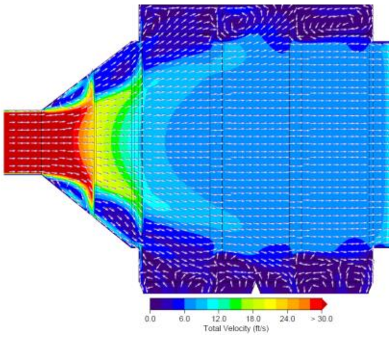


Figure 14 – CFD Results with New Perforated Plate Logic (Physical model L/D=0.54), Side View

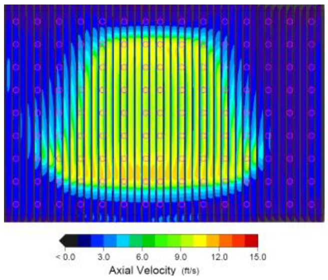


Figure 15 – CFD Results with New Perforated Plate Logic (Physical model L/D=0.54), ESP Inlet Plane

physical models, and actual plant test data. Additional angular wind tunnel data may be required as well, of more sheet metal perforated plates for models but also of steel perforated plates of full scale ESPs. This will further expand the database for the new perforated plate logic.

Additional Observation – Manufacturing Methods for Perforated Plates

Through the course of the research, another item was shown to influence model results. This was particularly noticed for physical model perforated plates, which are punched using a die. The plates feature a “smooth side” and a “rough side” to the touch, having to do with the direction that the die punches through the plate. The smooth side is the upstream side, and features a minute bell-mouth radius on each hole. The rough side is the downstream side of the punch, which is sharp and even can contain burrs extending beyond the plate thickness.

During the angular wind tunnel testing process, the smooth side was placed facing upstream to the flow direction. The loss coefficients of Table 1 are based on this consistent orientation. For each of the plates, a test was also conducted reversing the plate orientation, with the rough side facing upstream. As one might expect, there was a difference in loss coefficient since the loss through a sharp-edged orifice is larger than through a radiused or beveled orifice. The difference was considerable in some cases; as indicated in Table 4 the loss coefficient can be up to 30% larger if the rough side is faced upstream. This makes it important for physical models to use a consistent installation process for the perforated plates. Since the loss coefficient from a wind tunnel test is used for CFD, this also affects the CFD predictions.

Plate	Upstream Surface	Loss Coeff	Plate	Upstream Surface	Loss Coeff
13	smooth	2.76	13	rough	3.34
15	smooth	3.52	15	rough	4.60
2	smooth	6.67	2	rough	8.77
5	smooth	6.41	5	rough	6.95
8	smooth	2.04	8	rough	2.07
9	smooth	1.69	9	rough	2.04

Table 4. Loss Coefficient Variation of Perforated Plate Punch Direction (0 degree incoming angle)

Of course, the goal of the CFD or physical model is to match the loss coefficient of the full scale ESP. It is thus uncertain whether it is better to orient the laboratory perforated plates with smooth side or rough side facing upstream.

Industry personnel note that the manufacturing process of full scale perforated plates varies, with many being gang punched with a die, but others being laser cut, plasma cut, or water jet cut. These different processes lead to different geometry of the holes (sharp, radiused, beveled, etc.). It is not necessarily industry practice to specify a degree of sharpness of the perforated plate holes, nor to specify which way to orient plates during installation. Industry personnel have noted that in some extreme cases, especially as dies wear, the burrs on the downstream side of the punch are visually evident and an extra step of shop grinding the lips off the sheets is performed.

There is thus no conclusion at this point related to the sharpness or orientation of the perforated plate. The data of Table 4 does indicate that this can be a large influence, and this topic may warrant further investigation by the industry to obtain the most accurate ESP designs.

Summary and Conclusions

Research into the detailed flow behavior of flow through perforated plates has indicated a potential deficiency in physical and CFD modeling methods for ESPs. The typical representation of perforated plates in flow models, whether CFD or scaled laboratory models, is based on the loss coefficient found in industry handbooks. The basis of these standard loss coefficients is from laboratory experiments that are somewhat idealized compared to the complicated geometry of an ESP, particularly the “diffuser-style” ESP inlet.

Using experimental data from a custom wind tunnel that takes into account angular flow entering a perforated plate, new methodologies for physical and CFD modeling have been developed. These methodologies better account for the variation in loss coefficient and the straightening effect of a perforated plate versus the incoming flow angle.

An improved practice for selection of perforated plates for scaled physical models is suggested that takes into account not only the loss coefficient but the ratio of plate thickness to hole diameter (L/D). This factor plays a large role in the straightening effect of a perforated plate. Further, an advanced method is introduced to represent a perforated plate in a CFD model. Rather than a standard one-dimensional loss coefficient, or “porous jump”, custom logic is incorporated into the CFD solver to affect the fundamental relationship between the momentum and the pressure across the plane of the perforated plate. This allows the loss coefficient to have a three-dimensional influence on the flow, better representing the flow-straightening effect of the perforated plate, without having to mesh the details of the plate holes and thickness.

By adopting these advancements in modeling practices that carefully address the loss coefficient and straightening effect of a perforated plate, it is anticipated that correlation of velocity distribution predictions from either type of flow model will be closer to each other and closer to real-world measurements. Having more accurate flow models will aid the industry in developing more highly optimized ESP designs within tractable and economical modeling budgets and schedules. More work is required, but when perfected these methods will advance the current ESP design process, leading to the end goal of minimized emissions and improved ESP operation.

References

[1] Katz, J., *The Art of Electrostatic Precipitation*, Precipitator Technology Inc., 1979.

[2] Gretta, W.J., and Grieco, G.J., *“Consideration of Scale in Physical Flow Modeling of Air Pollution Control Equipment,”* International Joint Power Generation Conference, 1995.

[3] Dumont, B, and Mudry, R., *Computational Fluid Dynamic Modeling of Electrostatic Precipitators*, 8th International Conference on Electrostatic Precipitation, May 2001.

[4] Mudry, R., *Flow Modeling and Ash Deposition*, NOx-Combustion Round Table & Exhibition, February 2013.

[5] Institute of Clean Air Companies, *“Electrostatic Precipitator Gas Flow Model Studies,”* ICAC Publication EP-7, 2004.

[6] Lindeburg, M., *Mechanical Engineering Reference Manua*l, Professional Publications, Inc., Belmont, CA, 1995, p. 3-33.

[7] Patankar, S., *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing, 1980.

[8] Idelchik, I.E., *The Handbook of Hydraulic Resistance*, Jaico, (2003), pp. 516.