Celebrating 40 Years
Fall 2015
AIRFLOW SCIENCES CORPORATION
The Airflow Update

Flow Induced Pressure Fluctuations in Duct Systems

Poorly designed duct transitions can result in higher than necessary fan power requirements and limit unit operation. Problems such as high pressure drops and duct vibration can be improved when the duct flow is optimized. Many older industrial plants were designed without modern modeling techniques, such as Computational Fluid Dynamics (CFD), so inherent flow problems can exist in the layout of the duct. Flow separation in poorly designed duct elbows, vortex shedding, and periodic irregularities in the flow behavior can be difficult to recognize with limited plant instrumentation, but can be identified and corrected with flow analysis.

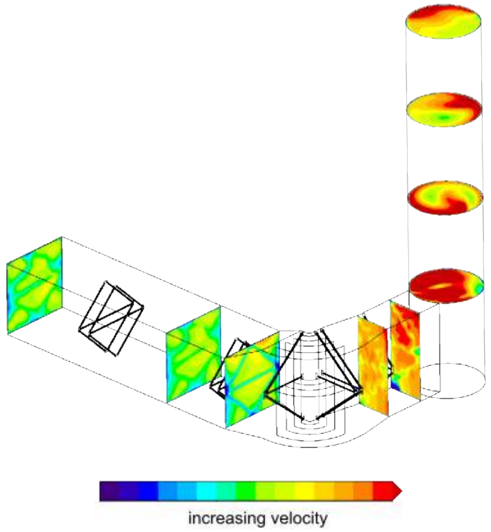


Figure 1: The baseline CFD velocity profiles indicate the severe amount of swirling in the stack, causing the stack vibration that was identified at the plant.

A recent flow model study of the stack and ductwork of a 870 MW power plant was conducted to determine the cause of vibration and transient pressure fluctuations. A CFD model was developed first with the existing ductwork geometry from the ID fan outlet to the stack outlet. An unsteady separation zone in the flow entering the stack was severe enough to prevent the CFD simulation from converging to a steady state solution with the baseline design. The velocity profiles in the stack inlet breeching area revealed unsteady swirling patterns that continued up the stack (Figure 1) due to the flow separation. To

(continued on page 2)

HVAC Exhaust Fan Modeling

The effectiveness of a ventilation system can be influenced by the uniformity of flow through the entire HVAC system, starting with the supply fan, through the ductwork to building loads, and to the exhaust fan. Although the result is typically a decrease in HVAC performance or vibration concerns, the worst case scenario is equipment failure.

A nuclear power plant had experienced blade failure on two of the three exhaust fans multiple times since plant startup. The South fan functioned normally and reliably, regardless of which fans were in operation (Figure 1). To diagnose the root cause of the Center and North fan failures, field testing and CFD modeling were used in conjunction.

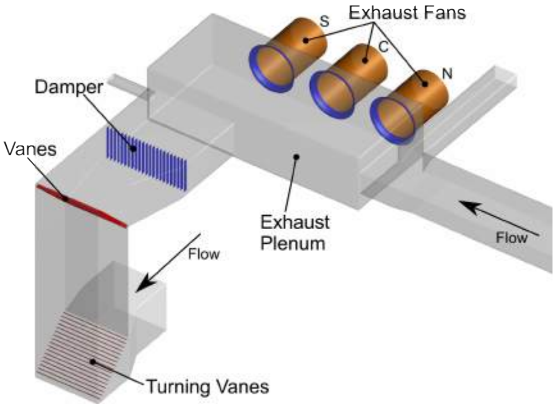


Figure 1: A schematic of the upstream ductwork, exhaust plenum, and three exhaust fans (South, Center, and North from left to right).

(continued on page 3)

Staff News

Kirsten and **Matt Gentry** celebrated the birth of their daughter Lily in June. Congratulations, and best wishes to your growing family.

2015 summer interns **Alex Trecha** and **Tim Harbaugh** have returned to the University of Michigan to complete their studies.

ASC celebrated the **40 year corporate anniversary** at an employee happy hour October 29th.

Edited by Julie Pierce

Flow Induced Pressure Fluctuations in Duct Systems (continued)

correct this flow problem and reduce the pressure drop at the duct elbow, CFD analysis was used to optimize the position and shape of flow control devices. The undesirable flow patterns were corrected by adding turning vanes at the stack inlet (Figure 2), eliminating the vibration, pressure fluctuations, and transient behavior.

A 1:12 scale physical model was constructed and used to confirm the CFD model findings. Portions of the physical model were constructed of clear acrylic, allowing for the observation of flow patterns. A smoke wand visualization test in the model revealed the unsteady swirling flow patterns (Figure 3). The baseline condition also exhibited the periodic pressure oscillations, as measured by a data acquisition system. The physical model was then run with the turning vanes that were designed with the CFD flow model and resulted in steady smoke patterns. Data confirmed that the large-scale fluctuations were eliminated, and that the overall pressure drop was reduced (Figure 4).

Measurements taken at the plant had alluded to these fluctuations in the past, but since most pressure readings were time-averaged, the trends were not completely understood. However, when considered in light of the new flow model study findings, it was apparent that significant pressure savings could be realized by installing turning vanes in the stack. Field testing after turning vanes were installed at the plant have confirmed the study conclusions, and found that on average, **pressure savings of about 2.5 IWC were realized.**

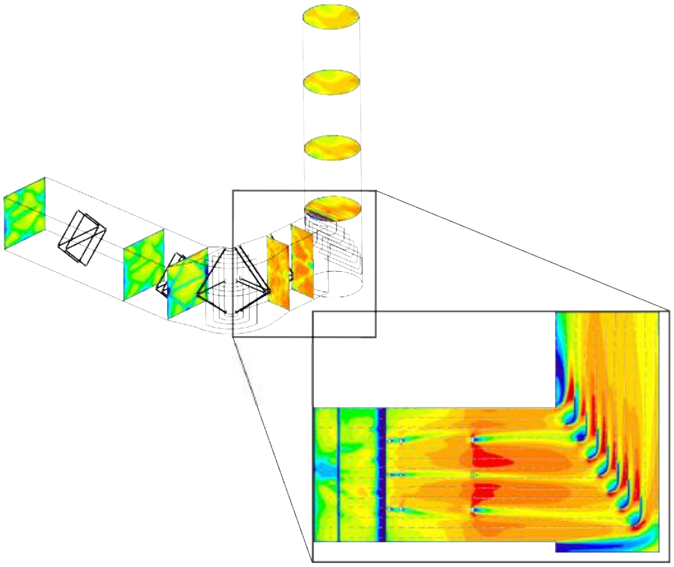


Figure 2: The optimized turning vanes greatly improved the flow at the elbow area, resulting in uniform flow in the stack.

Figure 3: Smoke visualization in the physical model confirmed the flow swirl patterns in the stack that were identified in the CFD model.

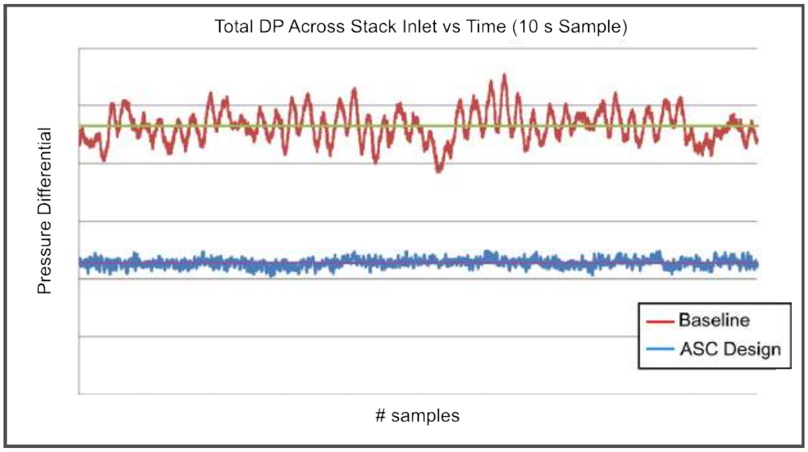


Figure 4: Pressure measurements in the stack over time show the pressure fluctuation (red data) in the baseline was eliminated with the flow control devices (blue data).

2016 Conferences/Exhibits
(complete list on website)

Come visit our booth, or hear a presentation.

Reinhold NO$_x$ Combustion RoundTable & Expo
Orlando, FL
February 1-2

HRSG Conference
New Orleans, LA
February 22-24

SSSAAP (Stationary Source Sampling and Analysis for Air Pollutants) Technical Conference
Point Clear, AL
March 20-25

EPRI CEM (Continuous Emissions Monitoring) Conference & Expo
Detroit, MI
May 4-5

Reinhold APC-Wastewater RoundTable & Expo
Dearborn, MI
July 18-19

MINExpo
Las Vegas, NV
September 26-28

International Baking Industry Expo (IBIE)
Las Vegas, NV
October 8-11

Seminar at Your Office
We make house calls!

HVAC Exhaust Fan Modeling (continued)

Figure 2: 3DDAS flow measurement system

3D velocity measurements were taken on-site to quantify the flow characteristics entering each of the exhaust fans. A high accuracy measurement system (Figure 2) was used. The test data showed that the flow entering the South fan was more uniform and less angular than the flow entering the other two exhaust fans. Even so, the velocity uniformity at the South fan was outside the recommended industry standards (AMCA 803). The flow measurements taken at the plant were used as verification for the next phase of the flow model study - CFD analysis.

A CFD model was developed of the existing HVAC exhaust system. As expected from the field data, the flow characteristics to the South fan were more desirable than the flow to the other two fans. This trend held true whether a single or pair of fans was in operation (Figure 3). The South fan (far left) was directly across from one of the inlet branches; hence the flow was straight and more uniform. The flow from the other inlet branch swirled into an undesirable recirculation zone before being drawn into the North exhaust fan (far right).

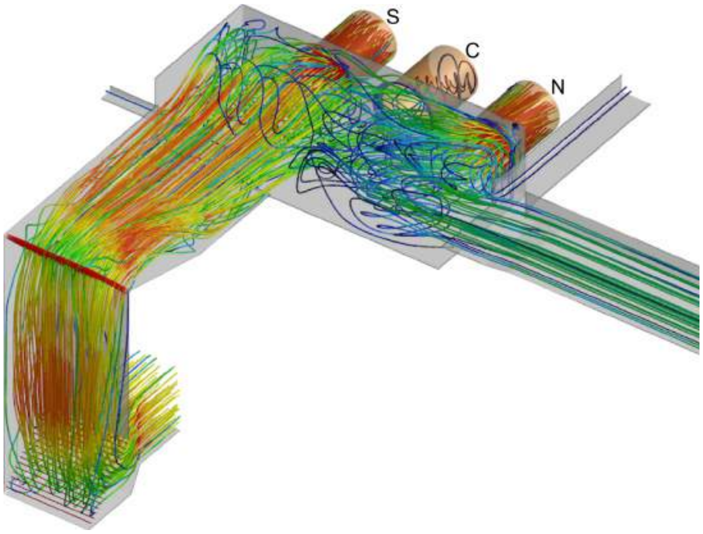


Figure 3: Baseline velocity pathlines with South and North fans in operation show more uniform flow at the South fan inlet (far left) compared to the turbulent flow at the North fan inlet (far right).

The next phase of the flow modeling analysis focused on evaluating various design alternatives that would correct the issues identified in the baseline geometry. Specifically, the new design needed to meet the following goals: improve uniformity to meet AMCA 803 guidelines, allow for any combination of fans in operation, and reduce pressure losses. Minimal design changes, such as adding flow control devices (i.e. vanes, baffles) within the plenum and existing ductwork, were considered first (Figure 4). Most of these alternatives had no significant positive effect on the flow characteristics and did not address the inherent imbalance.

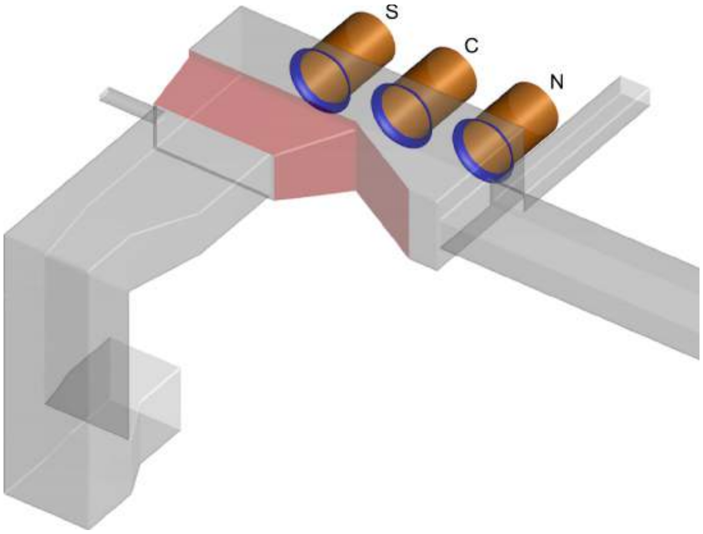


Figure 4: One design alternative added internal baffles (in pink). There was no significant improvement in flow performance, partly because the size of the plenum was effectively reduced.

More aggressive design changes were deemed necessary to resolve the flow problems and prevent future fan failure. An expanded plenum allowed a smoother transition from the ducts to the fans. Larger bellmouths resulted in increased uniformity at the exhaust fan inlets. These changes required rerouting the ductwork farther away from the fan inlets. Because the HVAC system was in a functioning plant, all geometry changes needed to fit within the confines of the available space. Nearly 100 iterations of rerouted ductwork and enlarged plenums combinations were considered. The final recommended

Contacting ASC

www.airflowsciences.com
asc@airflowsciences.com

Headquarters
12190 Hubbard Street
Livonia, MI 48150
phone: (734) 525-0300

Western Region Office
P.O. Box 22637
Carmel, CA 93922
phone: (831) 624-8700

Southeastern Region Office
1906 Arrowhead Dr NE
St. Petersburg, FL 33703
phone: (727) 526-9805

HVAC Exhaust Fan Modeling (continued)

configuration (Figure 5) routed most of the inlet ducts to the rear of the plenum and included flow control devices for improved flow distribution and reduced pressure loss.

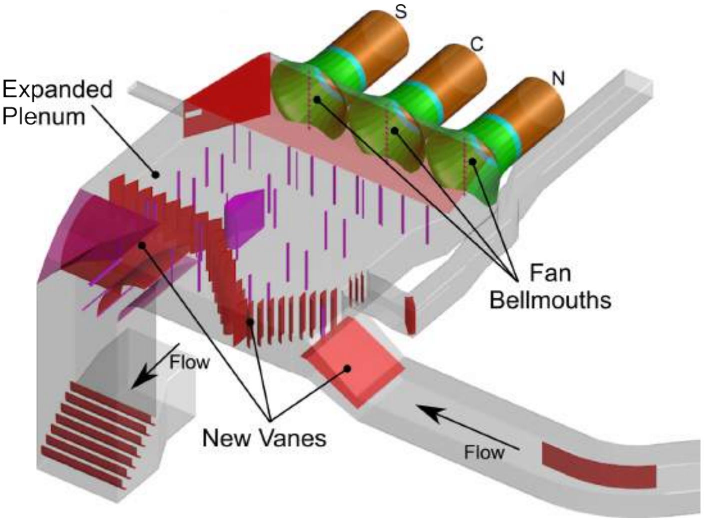


Figure 5: The recommended design included rerouting of ductwork, an enlarged plenum, extended bellmouths (in green), and the addition of flow control devices (in red).

CFD model results show significant differences from the baseline design, notably smoother flow transitions (Figure 6) and lower pressure drops. The swirling flow near the North exhaust fan in the baseline condition has been eliminated. The flow uniformity at each of the three exhaust fan inlets meets the AMCA 803 guidelines for all combinations of operating fans.

The HVAC geometry updates were implemented in 2014. Field test data was collected and verified the improved flow that was expected from the CFD analysis. For the past year the nuclear power plant has been running without fan failures. Plant personnel have been pleased with the improved flow performance.

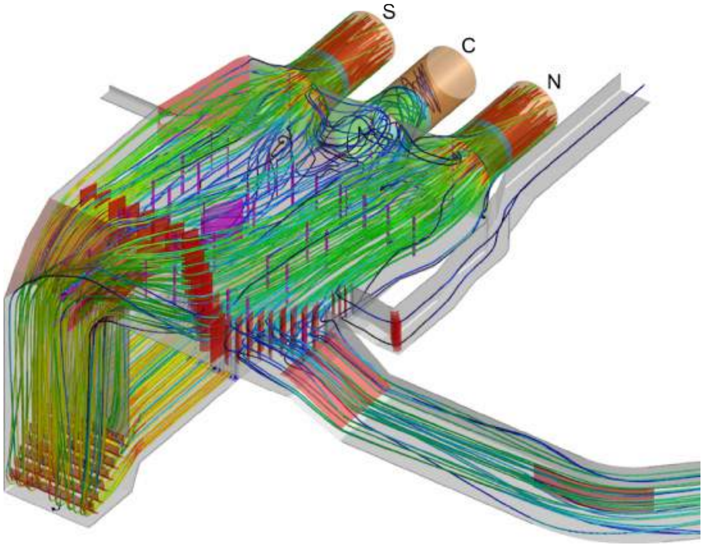


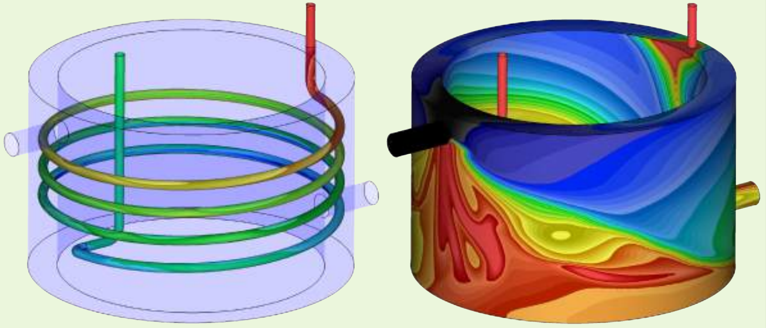
Figure 6: When the South and North exhaust fans are operating, the final design pathlines show greatly improved flow uniformity and balance.

The results of this CFD flow model were presented with DTE at the July 2015 NHUG Summer Meeting in Baltimore. The powerpoint presentation can be found on our website in the Resources section.

Azore® Version 2.0 Released

Airflow Sciences Corporation participates in the development of Azore®, a CFD software suite owned and distributed by Azore® Technologies, LLC. Version 2.0 of the flagship product Azore was recently released and includes the following improvements for solving fluid flow problems:

1. Conjugate heat transfer - Providing heat transfer predictions that span through fluid and solid materials.
2. Multiple fluid domains - Supporting heat transfer predictions between two or more separate fluid paths (example shown).
3. Improved parallel scaling - Allowing parallel runs on laptop computers / desktop work stations and off site compute clusters.
4. Higher quality rendering – Restructuring of the graphics pipe line to take advantage of modern multi-core GPU's (capabilities shown in renderings).
5. Advanced scripting features - Providing improved tools for efficient post processing of simulation results.



The heat transfer between two distinct fluids through a solid material is depicted in this shell and tube heat exchanger model. The hot fluid inside the tube (left) is cooled via convection and conduction through the tube walls by the fluid inside the shell (right).

Airflow Sciences has been awarded a DOE Phase I research grant to explore advanced polyhedral mesh generation technologies and has licensed the foundation for these concepts from Azore Technologies, LLC. The mesh technology development will lead to automated polyhedral meshes that have properties typically associated with hand crafted hex mesh topologies. Airflow Sciences will be participating in the development of these technologies with Azore Technologies, LLC and ultimately utilizing the improved CFD capabilities to make high quality numerical predictions for its customers.