ANALYSIS OF THE AP1000® PASSIVE CONTAINMENT COOLING SYSTEM AIR FLOW PATH USING COMPUTATIONAL FLUID DYNAMICS

Richard F. Wright, Hong, Xu, Megan E. Durse, Todd Sutton
Westinghouse Electric Co., LLC
1000Westinghouse Drive
Cranberry Township, PA 16066
wrightrf@westinghouse.com; xu1h@westinghouse.com; durseme@westinghouse.com; suttonat@westinghouse.com

ABSTRACT

The **AP1000®** plant is an 1100-MWe pressurized water reactor (PWR) with passive safety features and extensive plant simplifications that enhance construction, operation, maintenance, safety, and costs. The first deployment of the **AP1000®** plant formally began in July 2007 when Westinghouse Electric Company and its consortium partner, the Shaw Group (now CB&I), signed contracts with China's State Nuclear Power Technology Corporation Ltd., Sanmen Nuclear Power Company Ltd., and Shandong Nuclear Power Company Ltd. for four **AP1000®** units on coastal sites of Sanmen and Haiyang, China. Both sites have the planned ability to accommodate at least six **AP1000®** units. Construction for all four units is largely concurrent.

Additionally, the United States (US) Nuclear Regulatory Commission (NRC) issued combined licenses (COLs) to allow Southern Nuclear Operating Company (SNC) & South Carolina Electric & Gas Company (SCE&G) to construct and operate **AP1000®** plants at the existing Vogtle & VC Summer sites in Georgia and South Carolina, respectively. Although construction at both US sites is underway, the first four China **AP1000®** plants will become operational ahead of the U.S. Domestic **AP1000®** plants.

One of the major passive safety systems, the **AP1000®** plant passive containment cooling system (PCS), is designed to remove heat from the reactor containment building in the event of an accident. After a release of energy inside the containment building, the pressure and temperature of the containment atmosphere increase, and actuates the PCS. Water is supplied to the top of the containment from a large tank of water atop the concrete shield building. An air flow path is provided which admits air through a series of louvers near the top of the shield building. The air travels downward in an outer annulus formed by the inside surface of the concrete shield building, and a baffle structure between the shield building and the containment. The air turns 180-deg at the bottom of the annulus, and enters an inner annulus between the baffle and the containment wall. Air is exhausted through a large chimney in the top of the shield building. Air is heated in the inner annulus by the relatively warmer containment wall, and rises due to buoyancy compared to the cooler, denser air outside the building. Heat is removed from the containment through a combination of convection between the air and the containment wall, and, in the case of an accident, evaporation of the applied water film.

Computer simulation of the **AP1000®** containment requires accurate representation of the important phenomena such as the air flow characteristics and the heat and mass transfer from the containment wall to the water film and air. Natural circulation air flow is highly dependent on the overall flow resistance in the flow path. To better understand these phenomena, a series of scale model tests of the **AP1000®** PCS

were performed. One test, the PCS Air Flow Path Characterization Test determined the flow resistance characteristics of a 1:6 scale model section of the PCS air flow path. The data were used to develop loss coefficients which were used to simulate the **AP1000®** PCS.

A computational fluid dynamics (CFD) model of the test was developed and benchmarked against the test data. The model was expanded to simulate the full-scale **AP1000®** PCS maintaining the mesh density used in the test. The resulting model can be used to confidently predict the PCS air flow characteristics without the need of a full-scale test. The CFD model was used to verify the loss coefficients developed in the scale model test.

KEYWORDS
AP1000®, passive containment cooling, CFD

1. INTRODUCTION

The passive containment cooling system (PCS) is one of the major **AP1000®** passive safety systems (See Figure 1). The purpose of the PCS is to transfer reactor decay heat from inside the containment shell to the environment following an accident without exceeding the containment design pressure limit. The steel containment shell acts as the heat exchanger surface, transferring heat from the containment atmosphere, and into the air adjacent to the outside surface of the containment vessel. An air flow path has been designed to facilitate the removal of heat from the outside surface consisting of air inlets around the top of the cylindrical portion of the shield building.

Arrows show the PCS air flow path. Air enters into the concrete shield building through louvers near the top of the building, and inlet pipes. The air travels downward in a flow path formed by the inside of the shield building and the air baffle, turning 180-degrees at the bottom of the annulus. The air then enters a smaller annulus between the baffle and the containment. It is in this riser section that heat transfer between the exterior of the containment and the air occurs. The heated air then exits through a chimney in the top of the shield building.

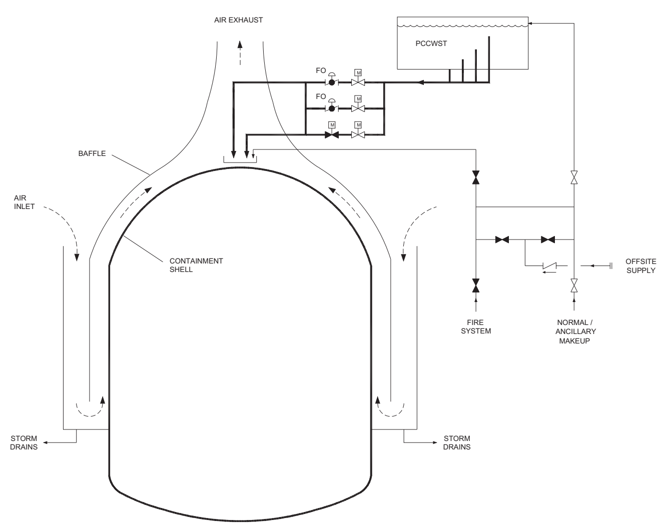

Figure 1. **AP1000®** Passive Containment Cooling System

The primary heat removal mechanism is via evaporation of a water film that is applied on the outside surface of the containment. If a high pressure or high temperature is sensed inside the containment, valves open between the Passive Containment Cooling Water Storage Tank (PCCWST) located at the top of the shield building allowing the contents to gravity drain over the top of the containment. The water film is formed by two weirs into a thin film that flows downward via gravity, and is heated by the relatively hotter containment wall. The heated water evaporates and removes considerable heat from the containment, and the heated mixture of water vapor and air, which is less dense than the ambient air, establishes buoyancy driven flow throughout the air flow path. The buoyancy driven flow is determined by a mass and energy balance between the thermal driving head developed by this density difference, and the pressure drop generated by the friction and form losses in the flow path. This is shown graphically in Figure 2. Should water not be available, the containment is cooled by convective heat transfer to the air adjacent to the containment exterior surface.

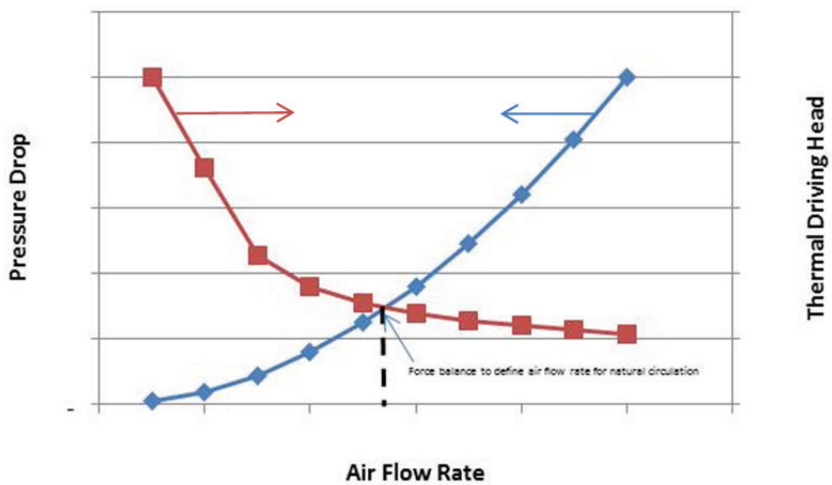

Figure 2. Natural Circulation Flow Force Balance

For evaporative cooling, air flow is not a critical parameter. This is exhibited in Figure 3 which shows the peak containment pressure for a loss of coolant accident (LOCA). Pressure continues to rise until the heat removal from the containment exceeds the decay heat from the reactor. At this time, the pressure slowly falls. Figure 3 shows that for a 100% increase in the air flow path resistance, there is virtually no impact on the containment pressure response.

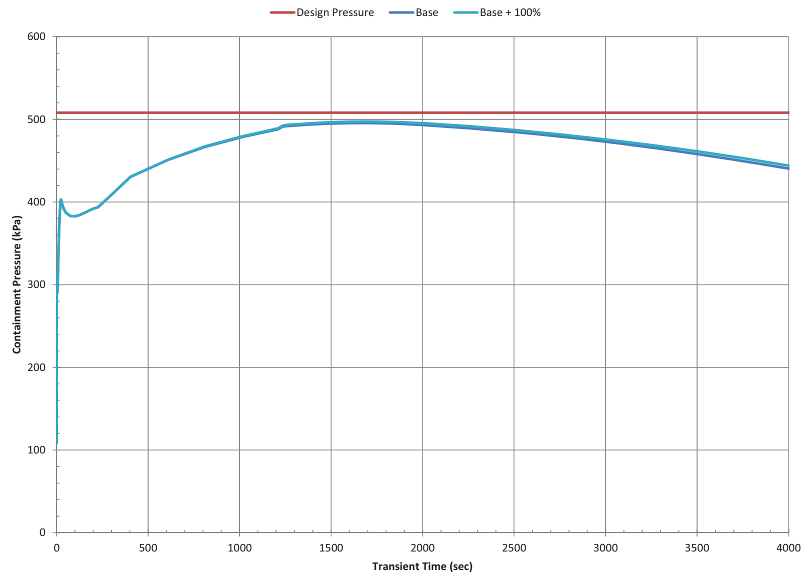

Figure 3. Containment Peak Pressure Sensitivity to Air Resistance – Large Break LOCA

For the case where water cooling is not available, air cooling can remove the heat from containment. The only design basis event where water is not available is a loss of power during refueling which results in a loss of shutdown cooling. When sufficient fuel has been moved from the containment to the spent fuel pool, the PCCWST is used to back up spent fuel pool cooling and is not available for containment cooling. When the decay heat load within containment is less than 4 MW, the containment only needs air cooling to maintain the containment pressure below design limits well beyond seven days.

For this case, the containment pressure is more sensitive to the air flow path resistance as is shown in Figure 4. While more sensitive to the PCS air flow path resistance, there is still large margin to the containment design pressure limit for this event.

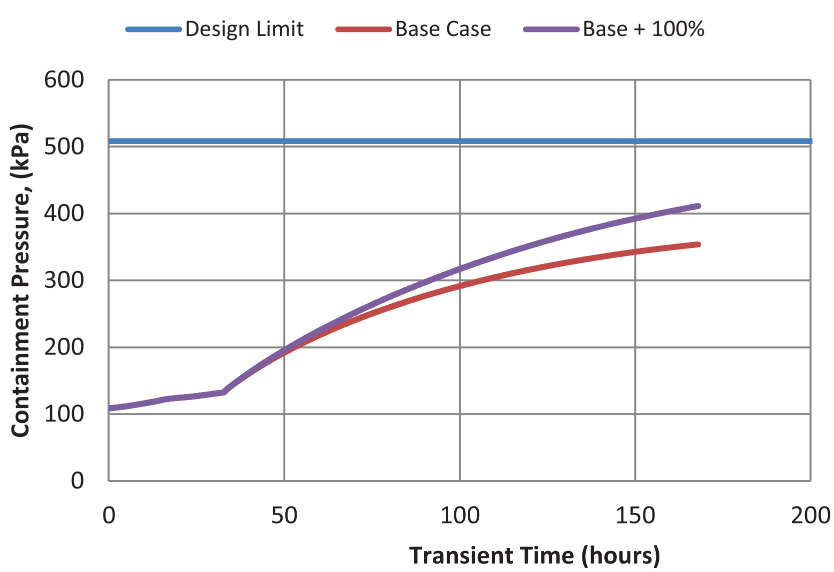

Figure 4. Containment Pressure – Loss of Shutdown Cooling during Refueling (4 MW of decay heat within containment)

Both cases show that the **AP1000®** containment pressure response to a design basis event is not highly dependent on the PCS air flow path resistance.

2. SCALE MODEL TESTING TO DETERMINE THE PCS AIR FLOW PATH RESISTANCE

Two different tests were performed to characterize the PCS air flow path resistance. The first test (Ref. 1) was part of the **AP600™** Design Certification effort, and consisted of a 1:6 length scale, 1/32nd sector model of the PCS air flow path from the inlets to the chimney. A large blower was used to provide air velocities so that the Reynolds number scaling would be maintained, as the velocities for the full-scale plant are always in the turbulent regime. These larger velocities resulted in good pressure drop measurements which were used to develop the loss coefficients used in the containment pressure analysis models (Ref. 2).

The **AP1000®** air flow path resistance was originally the same as the **AP600** design. However, the shield building was re-designed to be resistant to aircraft crash. The reinforced concrete building was redesigned to withstand large impact loads. In addition, the twelve open rectangular air inlets were replaced with 236 circular ducts 42.9 cm in diameter. These significant changes to the shield building design resulted in additional testing (Ref. 3) of the PCS flow path.

The Robust Containment Air Flow Tests (RAFT) test facility was constructed at Oregon State University to represent the revised design. The test facility, shown in Figures 5 and 6, is a 1:6 length scale, 1/32nd sector test fabricated from metal ductwork. A large fan is used to draw high-velocity air flow through the flow path to match the expected Reynold numbers. The facility is located inside a building to minimize the effects of external conditions such as wind and temperature changes.

Figure 5. RAFT Test Facility

The primary instrumentation is a series of flow differential pressure transducers which are located at 13 ports along the flow path, typically on either side of a discrete section of the flow path. The blower speed is also recorded, as well as air velocity measurements at several locations. Tests were performed at several air mass flow rates to cover all expected operating conditions.

All the major structures in the flow path are represented in the test are listed in Figure 6.

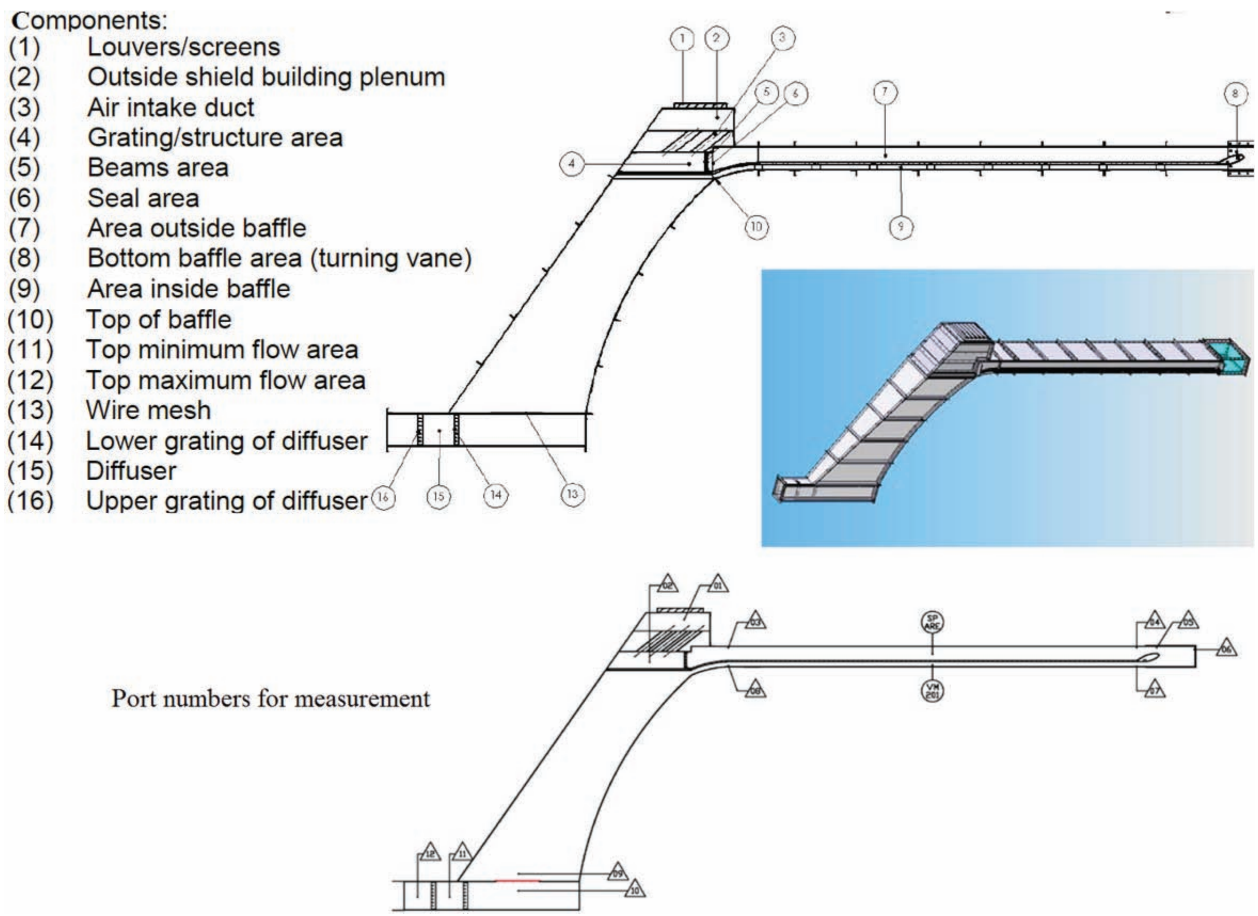

Figure 6. RAFT Test Instrumentation Locations

There are three sources of distortion in the test:

- Wall effects due to the added friction from the side walls
- Inability to model 3D effects such as circumferential variation in the air baffle, and air diversion around large structures such as the PCS valve room near the air outlet
- Small structures such as the baffle supports and structural members are not included in the facility

The loss coefficients developed using the RAFT test data were used in the **AP1000®** safety analysis model (Ref. 3) that was approved by the US-NRC. It should be noted that full-scale testing would require a factor of six smaller velocities to achieve prototypic Reynolds number. This would result in pressure drop measurements that are a factor of 36 times smaller than the RAFT tests. This uncertainty, coupled with likely variations in the actual environmental conditions due to wind, temperature, etc., make it likely that full-scale testing of the as-built **AP1000®** PCS air flow path would not result in meaningful data.

3. BENCHMARKING THE RAFT TEST WITH CFD

The RAFT test provides an excellent opportunity to benchmark CFD. CFX (Ref. 4) was used to model the test facility and the air flow measurement was used to set the boundary conditions at the outlet. The pressure drop was sampled from the model and compared with the test measurements.

The mesh density was varied to determine what the minimum model size would be to accurately model the test. Detailed features such as the meshes and gratings typically require significant detail to explicitly model. For regions where RAFT showed small pressure drops, porous media was used to model these structures. The optimized model contains 1.2 million cells.

A comparison of the pressure drop for Figure 11 shows the pressure drop at several measurement port locations for the test and the CFX simulation.

Port Number	OSU-RAFT-001 test Data	OSU-RAFT-001 (simplified, 1.2 million)		OSU-RAFT-006 test Data	OSU-RAFT-006 (simplified, 1.2 million)	
	Pressure	Pressure	Diff (%)	Pressure	Pressure	Diff (%)
	[mm H_2O]	[mm H_2O]		[mm H2O]	[mm H_2O]	
2	19.304	16.256	-15.77	151.0538	131.0386	-13.244
3	22.2504	21.2852	-4.387	174.9298	172.72	-1.262
4	23.3426	22.733	-2.635	182.245	183.9976	0.964
8	54.4576	55.4482	1.824	420.0398	436.6768	3.961
9	51.2318	48.8188	-4.692	390.8044	377.0376	-3.523
10	55.3974	53.594	-3.235	427.5074	413.9946	-3.162
11	64.9478	63.0174	-2.967	501.3706	498.7036	-0.53
12	66.2178	65.4558	-1.153	509.8796	512.191	0.452

Table 1. Pressure Drop Comparison RAFT vs. CFX

From this comparison, this mesh density provides excellent agreement between the test and CFD.

It was proposed that the CFX representation of the RAFT test be used as the basis for the development of a full-scale **AP1000®** PCS air flow path model.

4. EVOLUTION OF THE RAFT MODEL TO THE FULL-SCALE AP1000® PCS AIR FLOW PATH

Scale model testing is acceptable for characterizing the **AP1000®** passive safety systems. Where possible, these characteristics can be verified once the plant is built. For the PCS air flow path resistance, it is not practical to test at full scale due to the inability to generate high air velocities to obtain pressure drop data along the flow path. In addition, external conditions such as wind in the vicinity of the plant would make any measurable results difficult to distinguish from random wind-induced noise.

CFD can be used for full-scale "virtual testing". It allows the test to be performed under ideal conditions at high air flow rates that would be difficult or impossible to replicate in the plant. The problem with CFD is constructing a full-scale, detailed model that will yield acceptable results, and still be practical to run out today's computer platforms.

4.1. Development of a 1:1 Scale Axisymmetric Model of the PCS Air Flow Path

To develop such a model, a two-phase approach was adopted starting from the benchmarked CFX model of the RAFT test. First, the model was expanded from 1:6 length scale to 1:1 or full scale axisymmetric model. Second, the axisymmetric model was propagated 32 times to obtain a full 3D representation. Circumferential variations in this model including the air baffle around the equipment and personnel hatches, and the inclusion of the PCS valve room near the top of the shield building were also included. In this way, a full 3D model of the **AP1000®** PCS air flow path was developed that could be used to provide data for comparison to RAFT. This process is shown graphically in Figure 7.

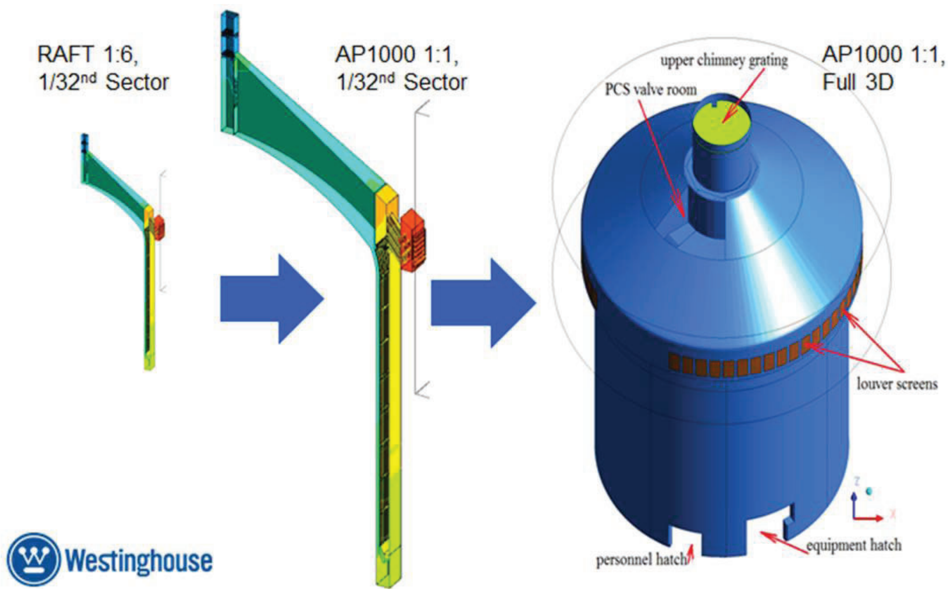

Figure 7. CFD Model Evolution from RAFT to Full-Scale, 3D AP1000®

To extend the benchmarked RAFT model (left in Figure 7) to the full-scale axisymmetric model (center), would typically require keeping the mesh density the same. Since the length scale is 1:6, this would require an increase in the total mesh size by a factor of 6^3, or 216. However, it is reasonable to expect that the 1:6 scale should yield the same results as the 1:1 scale provided the flow regime is the same.

The same nodalization that was used for the RAFT benchmark (1:6 scale) was used for the 1:1 scale axisymmetric model (prototypic plant geometry). The size of the cells was increased, and porous media was used for the mesh and gratings. The results showed that the loss coefficients for each section of the flow path were essentially the same for the same Reynolds number, and that the mesh was adequate to accurately predict the behavior of the 1:1 axisymmetric model. Extending the length scale of critical prototypic geometry (i.e. the riser) will not likely change the flow regime.

4.2. Development of a Full-Scale, 3D Model of the AP1000® PCS Air Flow Path

To extend the 1:1 axisymmetric model (center in Figure 7) to the full-scale, 3D **AP1000®** PCS flow path model (right), the 1.2 million cell model was propagated 32 times circumferentially.

The final model was modified to include the 3D effects such as the baffle variation and the variation in the diffuser region due to the PCS valve room. Figures 8-10 show key details of the CFX full-scale 3D model.

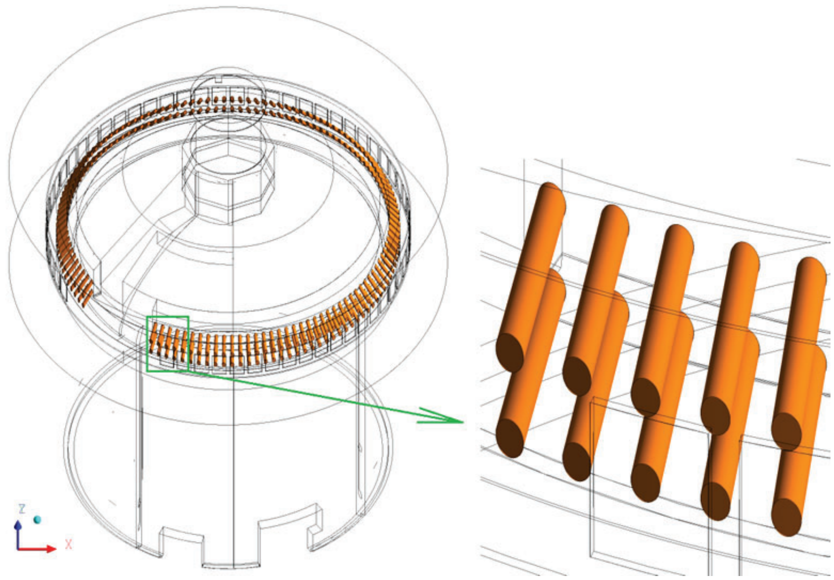

Figure 8. CFX Model – Air Inlet Details

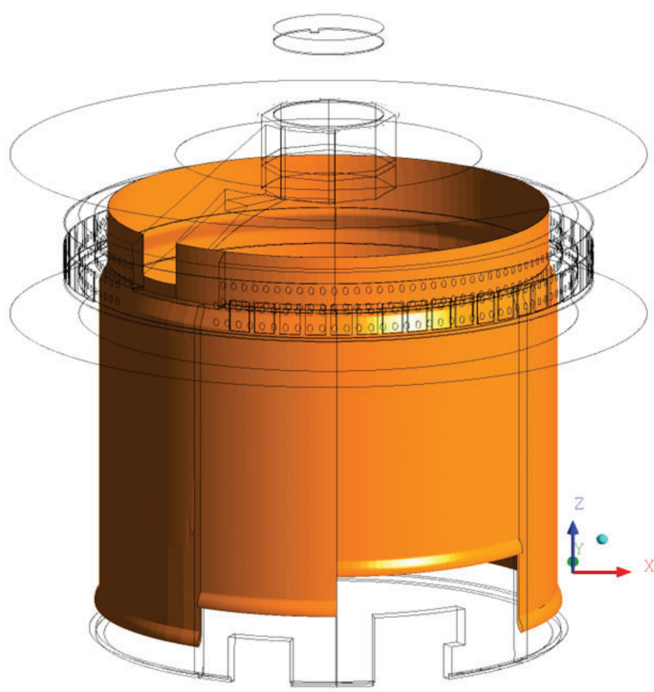

Figure 9. CFX Model – Air Baffle Details

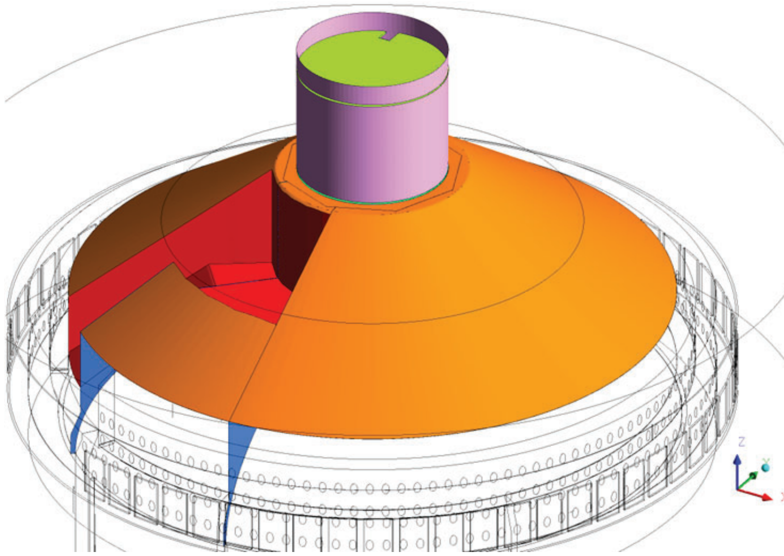

Figure 10. CFX Model Air Outlet Details

4.3. CFX "Virtual Test" Results

To run a "virtual test", the air mass flow rate is specified at the outlet plane. Three cases were analyzed:

- 1.4 m/s at the outlet
- 3.2 m/s at the outlet
- 8.2 m/s at the outlet

The results for these cases are shown in Figures 11 and 12. Figure 11 shows the pressure drop at the air inlets which is the largest pressure drop in the system. Figure 12 shows the pressure drop across the riser section of the flow path between the baffle and the containment, which is the second largest pressure drop. The solid line is the RAFT test, and the dotted line is the CFX simulation of RAFT. The red dots are the three CFX simulations of the full-scale **AP1000®** PCS air flow path. As can be seen, the results are very similar.

Figure 13 shows the streamlines for the full-scale, 3D **AP1000®** PCS flow path. Except for local areas (mainly the conic diffuer section), the flow field is quite axisymmetric. Based on the information provided in Figure 6 and Table 1, the main pressure drops happen for the flow going through air intake ducts, downcomer, riser and the chimney. This demonstrates that despite the 3D features of the model, the RAFT tests were representative of the full-scale flow path, and that the scaling of the test was adequate to provide suitable inputs to the computer simulation.

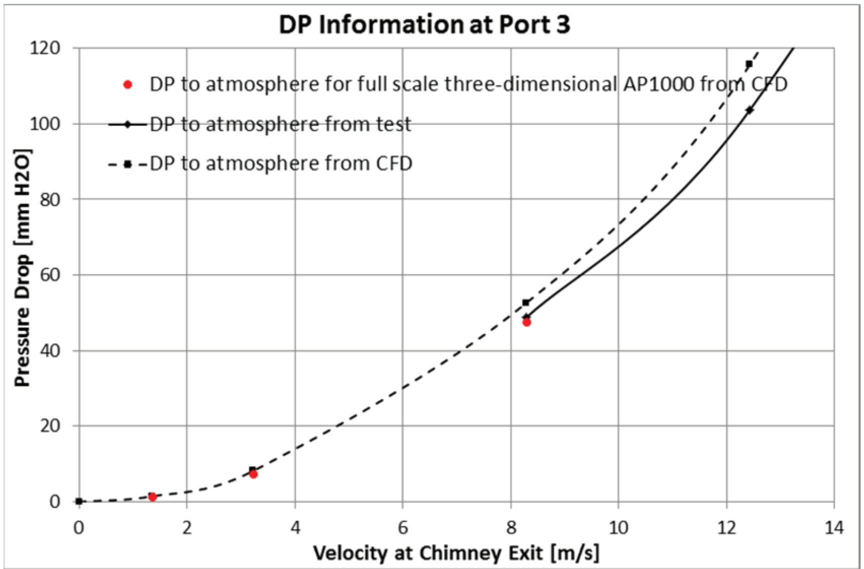

Figure 11. RAFT vs. CFD Comparison for Air Inlets

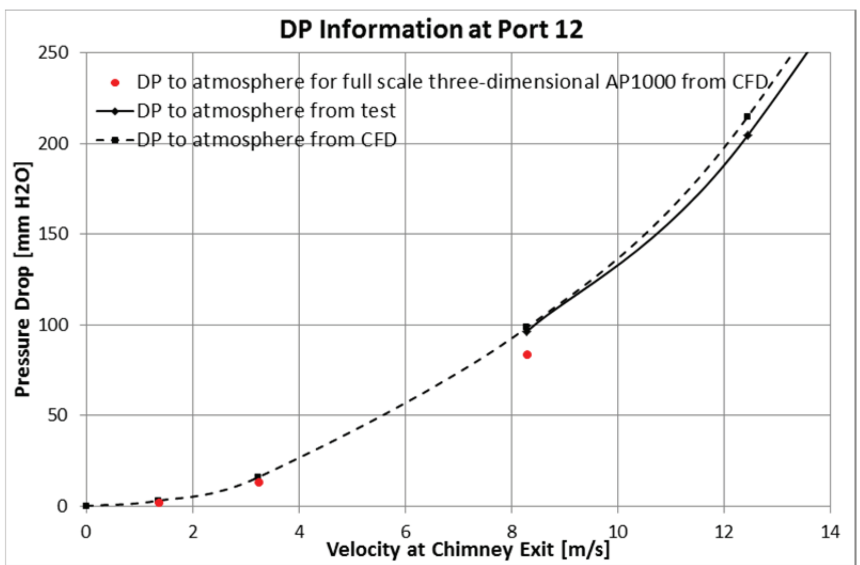

Figure 12. RAFT vs. CFD Comparison for Riser Section

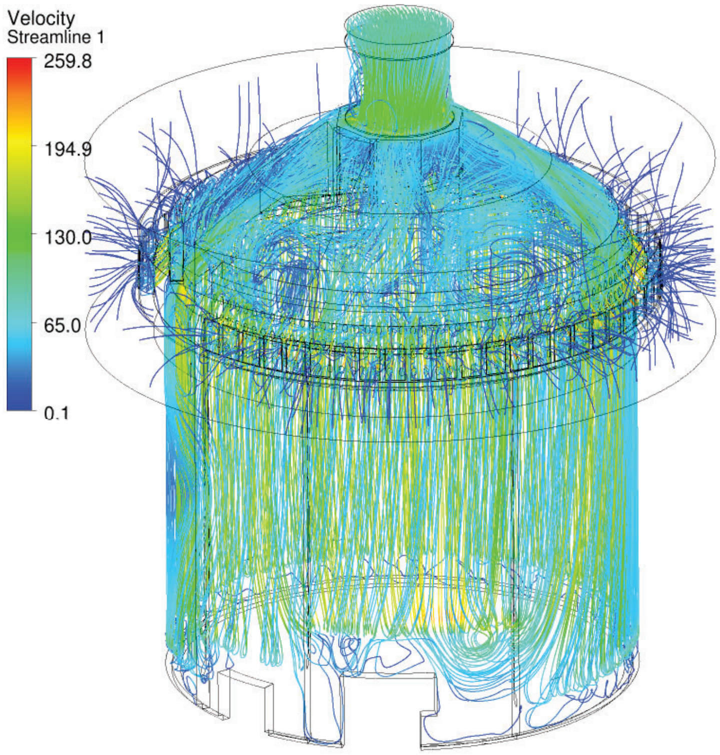

Figure 13. **AP1000®** Full-Scale PCS Air Flow Model – Streamlines

5. CONCLUSIONS

The **AP1000®** PCS air flow path has been characterized by scale model testing and CFD analysis. The analysis verified that the flow resistances measured in the tests were adequate for the development of safety analysis models.

In summary:

- The **AP1000®** passive containment cooling system has a large experimental basis and the analytical models developed from these tests are adequately verified for accident simulation.
- The PCS air flow path resistance is not a significant factor in design basis containment pressure analysis. This is due to the reliance of the PCS on evaporative water cooling for most accident scenarios. For the few cases where water is not available, the calculated pressure is below the design pressure limit with sufficient margin.
- Scale model testing at representative Reynolds number conditions is adequate for the development of the flow resistance safety analysis inputs. This is primarily due to the high velocities needed to offset the reduced scale.

- Full-scale testing is impractical due to the very small expected pressure drops associated with the lower velocities at the target Reynolds number, and the expected noise from environmental conditions.
- Three-dimensional effects are not large contributors to the overall flow field characteristics, or the flow resistance.

REFERENCES

1. W.A. Steward, A.T. Pieczynski, "Tests of the Air Flow Path for Cooling the **AP600** Reactor Containment",WCAP-13220, (1988).
2. Q. Wu, et al, "Scale Model Testing of the **AP1000®** Robust Containment Flow Characterization", OSU/DCP0430 (2012).
3. AP1000® Shield Building – Scale Model Testing of the AP1000® Robust Containment Air Flow Testing (RAFT) at Oregon State University – Test Analysis Report, APP-1208-T2R-101, (2010)
4. ANSYS-12.1-CFX-05, Rev. 0, "ANSYS CFX Solver Theory Guide", (2011).