

Correlation of Computational Fluid Dynamics and Physical Scale Models for Flue Gas Desulfurization Systems

Jeffrey D. Franklin, P.E. Robert G. Mudry, P.E. Holly A. Skelton Airflow Sciences Corporation Providing engineering services to industry since 1975 Specializing in:

- Fluid Flow
- Heat Transfer
- Particulate Transport
- Combustion

- Physical Modeling
- Laboratory Testing
- Field Testing
- Mass Transfer
- Computational Fluid Dynamics







ANSYS[®]

Introduction



Flow Modeling of Air Pollution Control Equipment

- Optimize Performance
 - Velocity patterns, uniformity
 - Chemical species injection
 - Particulate injection
 - Flyash drop-out/re-entrainment
- Avoid Maintenance Issues
 - Erosion
 Corrosion
 Pluggage
- Provide Input for Structural and Process Design
 - Turning vane aerodynamic loads
 - Pressure loss predictions for fan sizing

Introduction



Wet Flue Gas Desulfurization (FGD)

- New FGD systems, when retrofit to existing plants, face significant design challenges
 - Long duct runs
 - New or upgraded fans
 - Flow uniformity goals for fans and FGD absorber
- Flow modeling is usually performed to support the FGD system design
- Physical Scale Modeling vs. CFD Modeling
 - Physical scale modeling has long been industry standard
 - CFD offers a more cost and time efficient alternative

Modeling Methods – CFD Model

ANSYS[®]

- Typical Model Set-up
 - Includes internal features
 - vanes, baffles, mixers
 - pipe trusses, girders, gussets
 - resistance zones, straighteners
 - injection systems
 - Hybrid mesh
 - 5 million to 15 million cells typ.
 - Handcraft/flow aligned hex dominate meshes used
 - K-epsilon turbulence model
 - Steady state solution
 - Multiple operating conditions often simulated



Modeling Methods – Physical Model

Methodology

- Maintain sufficient similarity to ensure test result applicability to full scale
- Geometric similarity is ensured by directly scaling the geometry from actual to model scale.
- Sufficient dynamic similarity is achieved by matching the velocity head from the actual model in the scale model

Example:

- Full Scale Reynolds Number = 3,741,985
- Model Scale Reynolds Number = 383,840
- Turbulent regime maintained in scale model

ANSYS

Modeling Methods – Physical Model

- Construction
 - Typically 1/12th to 1/16th scale with accuracy of 1/16th inch
 - Made from clear acrylic
 - Internal blockages >5% included



Typical Physical Model

ANSYS[®]





- Cases 1-5 are part of studies done for two power companies to optimize complicated ductwork which occurred when new FGD absorbers were added to existing power stations.
- New ductwork included long runs, mergers, new fans and splits of flow. Maintaining current performance with the addition of the new ductwork was imperative.
- In each case, velocity profiles throughout the system, as well as overall system pressure loss, was evaluated using CFD and Physical Scale Modeling.

Velocity Distribution



Objective: <15% RMS at all loads

Case	Load	CFD	Physical	Difference (%)
1	Full	8.6	9	0.4
4A	Full	3.2	3.6	0.4
4B	Full	3.2	3.4	0.2
5	Full	7.4	6.6	0.8
1	Intermediate	8.8	8.6	0.2
2A	Intermediate	3.6	5.2	1.6
2B	Intermediate	4.2	4.5	0.3
2C	Intermediate	4.6	3.2	1.4
3A	Intermediate	6.5	9.6	3.1
3B	Intermediate	7.1	8.1	1
3C	Intermediate	6.7	7.8	1.1
1	Minimum	21.1	17.6	3.5
2B	Minimum	4.8	3.9	0.9
2C	Minimum	4.5	3.7	0.8
3B	Minimum	11.4	9.9	1.5
3C	Minimum	5.7	7.2	1.5

Velocity (%RMS) at Various Loads

Velocity Distribution – Case 3



FirstEnergy Sammis Plant – Units 5-7





 Three units (700 MW, 700 MW, 350 MW) combine to feed three new booster fans (A, B, C)

Velocity Distribution – Case 3





Case 3A Normalized Velocity at Booster Fan Inlet • Similar gradient high to low velocity







Velocity Distribution - Case3

ANSYS[®]



Physical Model



Case 3B: Normalized Velocity at Booster Fan Inlet • Similar pattern of low

velocity



Velocity Distribution – Case 3





Velocity Distribution

ANSYS[®]

Conclusions

- Average % RMS difference between modeling methods of 1.2%
- Flow patterns in normalized contour plots show similar trends between two modeling methods
- This indicates that results obtained by running a CFD model only would prove statistically and visually similar to those found if physical model was run as well.
- Both modeling methods are equally acceptable to design flow control devices that meet velocity uniformity goals

Pressure Loss



Objective – Minimize system pressure loss

Case	Load	CFD	Physical	Difference ("H2O)
1	Full	4.7	5.9	1.20
4	Full	2	1.9	0.10
5	Full	0.8	1	0.20
1	Intermediate	4.5	5.3	0.80
2	Intermediate	0.7	0.6	0.10
3	Intermediate	1.2	1.3	0.10
4	Intermediate	1.1	1.1	0.00
5	Intermediate	0.5	0.6	0.10
1	Minimum	4.2	4.3	0.10
2	Minimum	0.1	0.1	0.00
3	Minimum	0.4	0.3	0.10
4	Minimum	0.5	0.5	0.00
5	Minimum	0.2	0.3	0.10

System Pressure Loss ("H₂O) at Various Loads





FirstEnergy Sammis Plant – Units 5-7



Physical Model

Pressure Loss – Case 3



FirstEnergy Sammis Plant – Units 5-7



CFD Model

Largest Total Pressure Drop 2.03 – 0.84 = 1.2 "H2O





- Discussion of Case Study 3
 - In both models, the flow path from Unit 5 to the booster fans showed highest pressure loss
 - Overall difference in pressure drop 0.07 "H2O

Pressure Loss



Conclusions

- CFD and Physical Model results varied, on average, by 0.22 "H2O
- Correlation is independent of load
- Only one case had differences between CFD and physical scale model results of > 1"H2O

Conclusions



- Physical Modeling vs. CFD Modeling
 - Both methods provide similar numerical and visual results when performed on FGD ductwork.
- Benefits of Using CFD Model
 - Faster time to build model/easily modified
 - Precise use of full scale plant conditions
 - Design iterations completed more quickly
 - Limitless ability to acquire and analyze data at various planes throughout model
 - Multiple designs can be assessed in parallel on different computers