Pros and Cons

of CFD and Physical Flow Modeling

A White Paper by:
Kevin W. Linfield, Ph.D., P.E.
Robert G. Mudry, P.E.

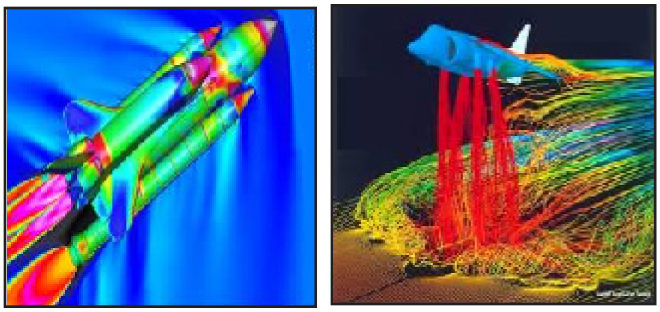
August, 2008

Airflow Sciences Corporation

When it comes to flow modeling to optimize performance or to develop solutions for flow-related problems, a frequent question that industry engineers ask is "Which is better – a CFD or physical (scale) flow model?". The short answer is "It Depends".

Background

Computational Fluid Dynamics (CFD) is a method of simulating fluid flow behavior using high speed computers. There are well-known mathematical equations that define how air and gases behave (Conservation of Mass, Momentum, and Energy). These equations are extremely complex (differential equations), and thus can not be solved by hand calculations except for very simple geometries such as flow around a cylinder. As computer power increased in the 1970s, the aerospace industry led the way in developing software to approximate solutions to these equations for complicated flows around air and space craft. Over the past few decades, these software tools have advanced to a point where accurate solutions can be obtained for complex flows, including heat transfer, particle tracking, and chemical reactions.



NASA

Figure 1 – CFD was first developed for the aerospace industry

In a CFD model, the three-dimensional domain is built in the computer via a CAD model. A computational mesh is then inserted into the domain – this mesh divides the region where flow travels into many control volumes, or cells. It is not uncommon for a CFD model to contain millions of these cells. The software then solves the equations of fluid motion (Conservation of Mass, Momentum, and Energy) in every one of these cells. The results are plotted as color contours to depict the flow parameters at any location within the domain. Thus, it is possible to analyze millions of velocities, pressures, temperatures, species concentrations, and other values. Computer-generated animations can also be created that provide flow visualization to observe the "real-time" motion of the flows.

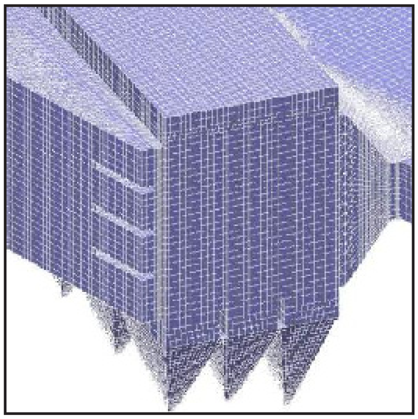
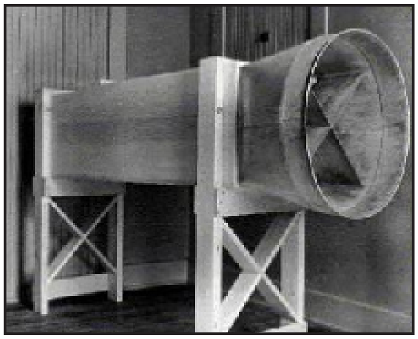


Figure 2 - CFD mesh for an electrostatic precipitator

It is difficult to determine how long physical flow modeling has been used in engineering applications. Obviously, full-scale versions of land and sea vessels were tested via trial-and-error for centuries to optimize designs. In the early 1900s, the Wright Brothers tested a scaled version of an airfoil in a small wind tunnel that led to the age of flight. Since the 1960s, scale models have been used to assess flow patterns in power plant duct systems, pollution control equipment, and boilers. Today, many of these models are built to a scale of 1:8 to 1:16, with 1:12 being a common scale factor.



www.wright-brothers.org

Figure 3. Wright Brother's wind tunnel

Figure 4. Physical flow model of a power plant dry scrubber and baghouse system

Once the physical model is constructed, large fans are used to draw air through the model at a flow rate that provides similar fluid dynamic behavior to the full scale system. Flow characteristics are measured over a grid of traverse points with an inserted probe. Values for velocity and pressure at select locations are thus obtained. Dust can be injected into a model to simulate the behavior of particulate in a system (to assess ash deposition, for example). Of course, the model is constructed with clear walls or windows so that flow patterns can be observed via smoke flow, strings, or bubbles. Model results can be presented as color contours, histograms, or other plotting methods similar to field testing.

Figure 5. Smoke flow through an SCR physical model

With either type of model, the flow patterns through the system are quantified and the model geometry is iteratively altered in order to optimize the flow. The location and shape of control devices such as turning vanes, mixers, baffles, and dampers are thus determined such that the design objectives are attained.

Accuracy

With the proliferation of high speed computers, the resolution and cell size of CFD models has improved dramatically over the past few decades. Airflow Sciences Corporation, which has used both modeling methods since 1975, has made numerous comparisons between CFD modeling, physical modeling, and field testing. Results indicate that both types of models share the same accuracy when it comes to velocities and pressures. For more on this please visit ASC's web site (www.airflowsciences.com) for *Conference Proceedings* which make this comparison with respect to ESP and scrubber modeling.

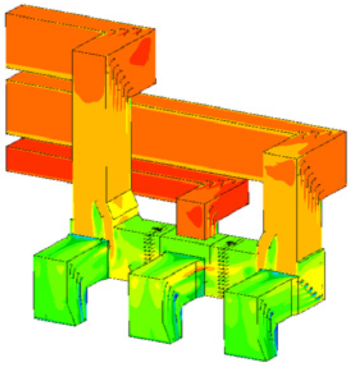
 

Figure 6. Comparison of CFD and physical model results for an FGD duct system where flow from 3 units (1,750 MW total) combine to feed 3 new booster fans (CFD pressure drop 1.19 IWC; Physical pressure drop 1.27 IWC)

There are certain areas where CFD and physical model results differ and it is not clear which provides the best real-world results. For instance, in SCR modeling, CFD models tend to predict slightly worse ammonia uniformity at the catalyst compared to physical models. Industry comfort is with the physical model in this case, and it is possible that the underlying mesh is not fine enough to resolve all the details of the injection and mixing. That said, there is not a lot of specific data published that shows how well either model matches real-world test data.

Similarly, for wet FGD absorbers and stacks, physical models are often used with liquid water injected into the models. Though the droplet size is not scaled properly, and evaporation is not represented accurately, some industry designers find value in the results and utilize their experience to interpret the results of the wet modeling. This is a very complex flow phenomenon, where two-phase flow momentum effects the droplet agglomeration exist. This is equally difficult to simulate with a CFD model, even with evaporation and thermal effects simulated. So both model types have drawbacks and industry experience in applying the results to the real world become important.

Schedule

CFD modeling is almost always faster than physical modeling. In many cases, design results from a CFD model are available several weeks before similar results from a scale model. And the more complicated or repetitive the model geometry is, the more advantage the CFD model has. This has to do with three factors: 1) the CFD mesh can usually be built faster than a scale model can be fabricated, 2) for repetitive or symmetric duct systems, portions of a CFD model can be copied and pasted while all pieces of the physical model need to be built separately, and 3) once a CFD model is built, it can be run simultaneously on separate computers. Thus, several designs can be evaluated at the same time, while only one physical model exists to evaluate designs.

Modeling Cost

CFD model studies are generally 20-40% less than a comparable physical model effort. This is tied quite strongly to the labor difference in model construction that influences the schedule. Also, many CFD tasks can be automated with the computer, including the design optimization process, whereas these tasks are primarily manual with the physical model.

Scale

Most physical models are built to scale, typically 1:12 or 1:16 for power plant models. CFD models are almost always built full size (1:1 scale). Care must be taken in computer models to ensure that the correct number, size, and shape of computational cells are used, and the level of detail to include must be considered in a scaled model to ensure geometric and dynamic similarity is maintained. In a CFD model, the Reynolds Number is often matched exactly, while in a physical model industry generally tries to match the Reynolds Number regime (i.e., laminar or turbulent). Both are fine as long as the boundary layer is negligible. This is generally the case for large power plant duct systems. Note, however, that one must closely match the exact value of the Reynolds Number if the objective is to determine lift or drag characteristics, or any system where the boundary layer along a surface is important.

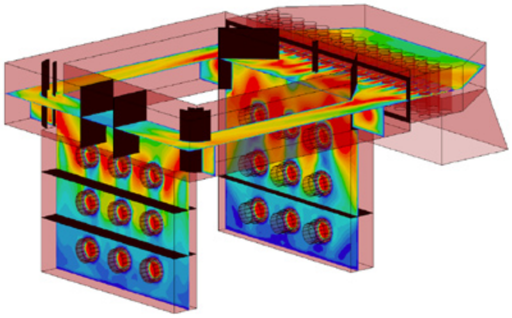


Figure 7. Comparison of CFD and physical model of a windbox

Particulate

In general, solid particle drop-out or re-entrainment is more accurate in a physical model. These tests help assess whether particulate (such as flyash) will fall out of the gas stream at lower unit flow rates. It is important to run the physical model at comparable velocities to the actual system, taking into account particulate aerodynamic characteristics which can be determined via wind tunnel tests. CFD results can be used to assess potential areas for particulate drop-out by examining low velocity regions near duct floors and other surfaces, but CFD cannot yet predict re-entrainment of particles as the system flow rate ramps up. This is because particulate build-up and re-entrainment are time-dependent phenomena. A physical model can be used to observe the particle behavior over time, but a CFD model is generally run as a steady-state simulation.

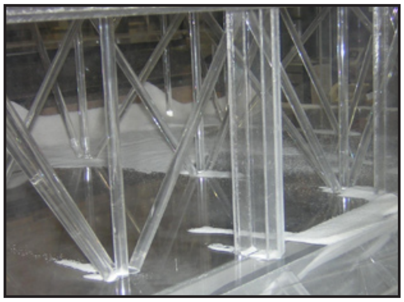


Figure 8. Physical model dust testing (dust accumulation simulated with fine white powder)

Particulate tracking is often desired to assess items such as Large Particle Ash pluggage, activated carbon/sorbent injection, or flyash erosion issues. Particles "in flight" are better simulated in a CFD model. This is because the CFD model is run full scale and can thus match all the important factors for particle behavior simultaneously (gravity, particle drag, gas velocity, gas viscosity, particle Reynolds number, particle mass and size). Some qualitative assessments of particle behavior "in flight" can be performed with physical models, but because all the scale factors and fluid dynamic properties are challenging to match simultaneously, quantifiable results are more difficult to obtain.

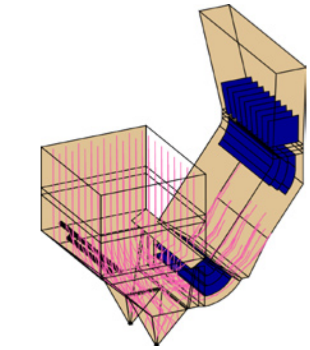


Figure 9. CFD tracking of ash particles in flight to assess LPA screen capture

Heat Transfer

For complex temperature problems (especially those involving conduction, convection, or radiation), CFD is really the only option. Physical models are often called "cold-flow models" since room-temperature air is drawn through the domain. Methods have been devised to simulate thermal mixing in a physical model (such as the merging of gas streams of differing temperature) via an injected tracer gas. Unless they are run at temperature, however, physical models cannot simulate heat transfer, addition of heat, or similar phenomena. CFD models are run at the correct temperature, and take into account changes in density, viscosity, thermal conductivity, and the heat transfer coefficient. CFD models of boiler combustion processes, heat exchangers, and evaporative processes are thus possible.

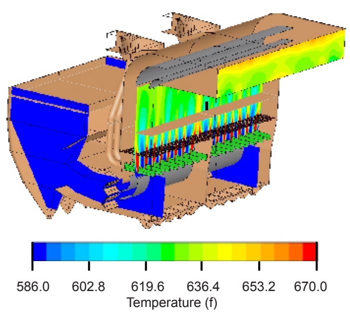


Figure 10. CFD modeling of thermal mixing (SCR inlet duct with economizer bypass flow)

Figure 11. Physical model testing of ammonia injection in an SCR via tracer gas simulation

Chemical Reaction

Simulation of a chemical reaction (such as combustion or change-of-state) can realistically only be done with a computational model or a laboratory test that includes the reactions. The latter would not really be referred to by industry as a "physical flow model" as much as a lab test (such as a combustion test chamber). Short of such a lab test, computer flow modeling can be used to simulate complex processes, incorporating individual species and compounds via reaction equations. Furnace combustion models are done via CFD to assess items such as burner/OFA systems, NOx creation, gas temperature uniformity, SNCR performance, slagging, and corrosion. Also, evaporative processes can only be fully simulated in a CFD model due to the changes in temperature and the moisture transfer from one state to another.

Visualization

Both types of models rely on color contour plots and flow statistics (uniformity, min/max values, etc.) to quantify results. Smoke injections and string tufts are also used to visualize the flow field inside a scale model. These are videotaped and photographed to document the flow patterns. Dust testing results are also videotaped so observations of particulate drop-out and re-entrainment can be documented. Flow animations from CFD results can provide similar views on the motion of the flow as a physical model smoke test. CFD animations can also present characteristics that are difficult to quantify in a physical model (i.e., a visual tracking of injected gas molecules, such as SO_3 or NH_3, through a duct).

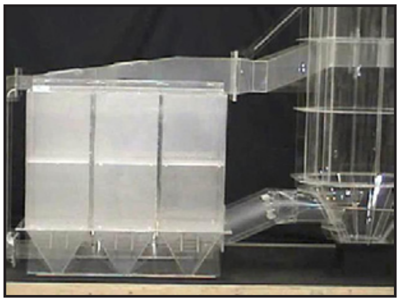


Figure 12. Smoke flow details in a physical model

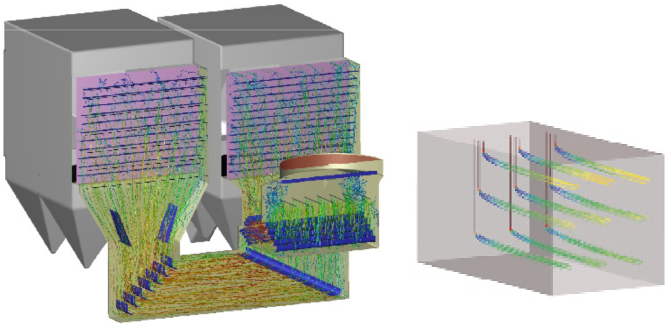


Figure 13. CFD injection of activated carbon upstream of an electrostatic precipitator

(Left – full ESP; Right – close up at lance location)

Touch & Feel

Seeing and touching a laboratory model can be more satisfying than looking at color contour plots and animations of a virtual model. Many clients appreciate walking around a 3-D scale model and examining flow details around the vanes, through perforated plates, and near internal structure. What's best depends on personal preference.

Storage

CFD models are usually stored on tape, CD-ROMS or DVDs which typically have a much longer storage life and negligible space requirements. Physical models can take up considerable space in a warehouse. A benefit of the physical model after the design effort is that it can be used for other purposes, including as a training tool for plant staff or as a display item for a plant lobby.

Conclusion

As noted above, there are certain flow characteristics that are best simulated with a particular type of model. Since there are advantages and disadvantages of both models, a number of new systems, particularly the more expensive pollution control devices such as SCR and FGD, utilize both modeling methods to get the optimal design. For ductwork systems, ESPs, or fabric filters, both methods have shown they offer similar results and acceptable designs; in these cases, the selection of the method often comes down to personal preference of the OEM or the end user.

Airflow Sciences Corporation

Corporate Office
12190 Hubbard Street
Livonia, MI 48150-1737 USA
Tel. (734) 525-0300
Fax (734) 525-0303
www.airflowsciences.com

Western Region Office
PO Box 22637
Carmel, CA 93922-0637 USA
Tel. (831) 624-8700

Southern Region Office
3709 Foster Hill Drive North
St. Petersburg, FL 33704-1140 USA
Tel. (727) 526-9805

Southeast Asia Agent
HANA Evertech Co. Ltd.
Jeff Jang
jjang@airflowsciences.com
+82-31-777-3780

Any comments relating to the material contained in this document may be submitted to asc@airflowsciences.com.

www.airflowsciences.com