N93-13386

Applications of CFD and Visualization Techniques

James H. Saunders, Susan T. Brown, and Jeffrey J. Crisafulli
Battelle
Columbus, Ohio

Leslie A. Southern
Ohio Supercomputer Center
Columbus, Ohio

Introduction

Computational fluid dynamics (CFD) and data animation are powerful tools for understanding and solving complex engineering problems. The large data sets generated by time-dependent simulations can be dramatically illustrated with computer animation, often readily revealing the physics of the flow field.

In this paper, three applications are presented to illustrate current techniques for flow calculation and visualization. The first two applications use a commercial CFD code, FLUENT, performed on a Cray Y-MP. The results are animated with the aid of data visualization software, apE. The third application simulates a particulate deposition pattern using techniques inspired by developments in nonlinear dynamical systems. These computations were performed on personal computers.

Details of the simulations are presented elsewhere [refs. 1, 2, 3]. In this paper, we focus on visualization of the data.

Air Flow Within Air Conditioned Rooms

In the first application, we simulated the three-dimensional air flow in two air conditioned rooms connected by a doorway, with the goal of understanding the effects of blower fan on-time and return air vent placement on comfort level and air exchange within the room. Although real house flows are usually more complex, this simplified case represents the essential physics, and thus can be used to investigate basic flow patterns.

Figure 1 shows the rooms, which were 3.0 by 2.4 by 4.3 m and 3.9 by 2.4 by 4.3 m with a single 0.9 by 2.3 m door and insulated outside walls. The outdoor temperature was 90 F to simulate a hot summer day. The ceiling and floor were held at temperatures of 90 F and 73 F, respectively. 260 cfm of cool air at 55 F entered the room through three inlet vents on the floor and exits through an outlet vent. The outlet vent could be located either on the floor or high on a wall. We considered two modes of fan operation: (1) running the fan only when the air conditioning was on, and (2) running the fan continuously. The air conditioner cycle of 15 minutes had an on-time of 6 minutes and an off-time of 9 minutes.

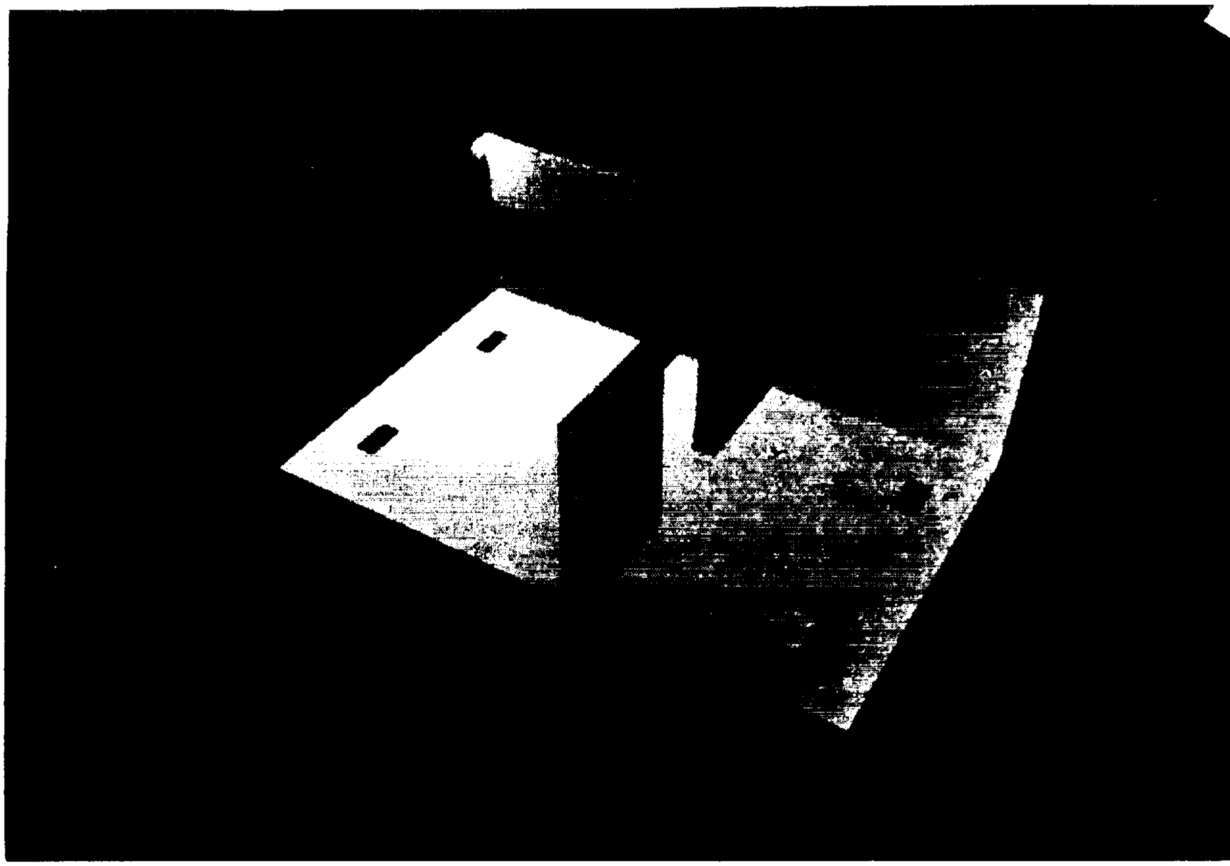

Figure 1. Animation of two air conditioned rooms

The room was modeled with 6061 nodes, using FLUENT, which solves the time dependent mass, momentum, and energy equations using a finite volume method. The temperature and velocity fields were then processed by apE to visualize the results. Three-dimensional objects and scenes were rendered by apE, using a scanline Z buffer approach to obtain photorealistic images that appropriately handled lighting, transparency and shading [ref. 4]. Polygonal iso-valued surfaces were constructed from the FLUENT data using a marching cubes algorithm [ref. 5]. For each timestep, three temperatures (77 F, 75 F, and 73 F) were illustrated with red, yellow, and blue isosurfaces, respectively.

The primary purpose of the visualization effort was to help characterize the air exchange within the two rooms as a function of fan on-time and outlet vent location. To visualize the air exchange process, a set of "glyphs" was used to mark the fluid. These massless particles, which track but do not interact with the flow, had two different shapes: pyramid shapes for existing room air and spherical shapes for the air entering through the vents as shown in Figure 2. For the glyphs to track the flow accurately, they had to interact with the velocity data generated by FLUENT. A special facility was written to perform this function in apE. The glyphs were color coded to indicate the local air temperature.

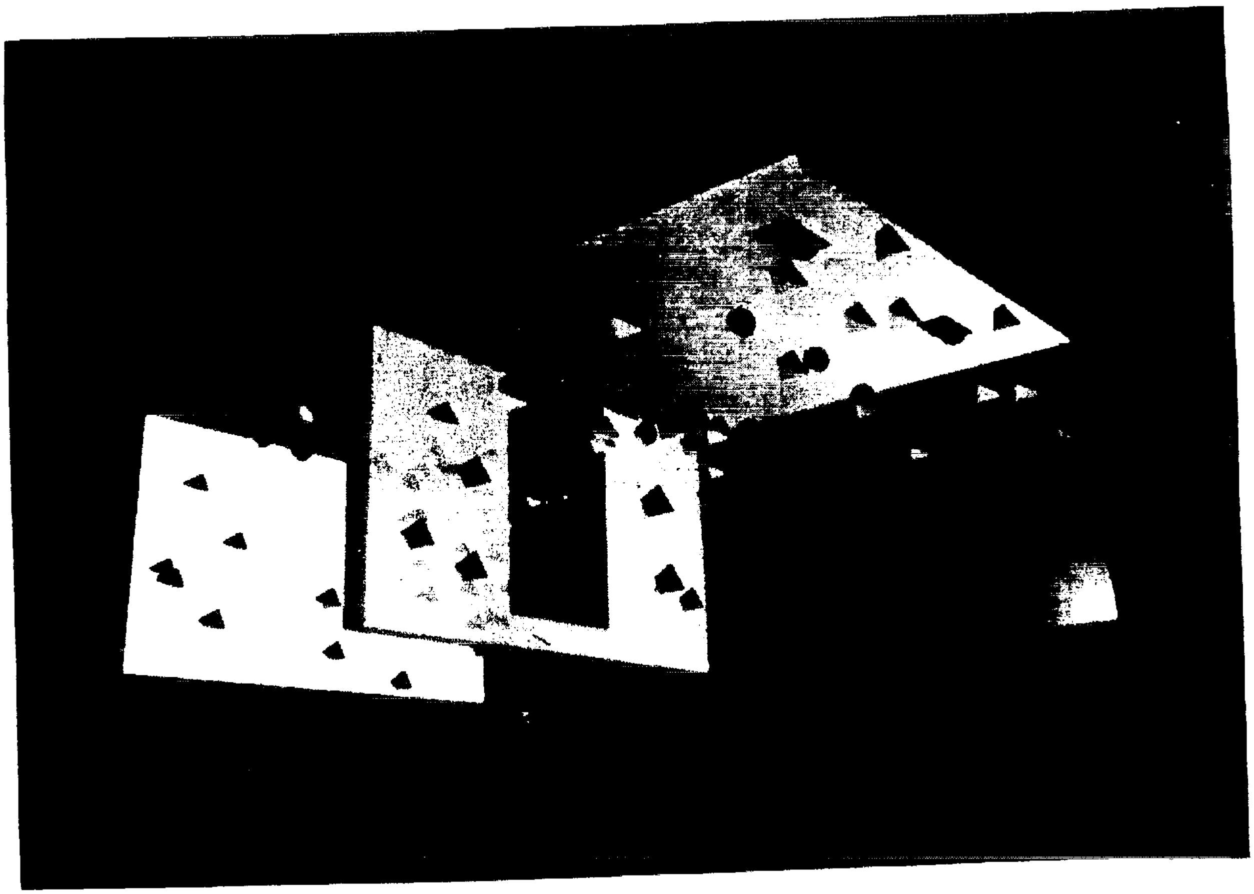

Figure 2. Visualization of the air exchange process: pyramid glyphs represent existing room air, spheres represent entering air

With the outlet vent on the floor and the fan running either intermittently or continuously, the isotherms are very flat, indicating poor mixing within the rooms. The glyphs clearly showed that the air short-circuited from the inlet vents to the outlet; the primary air flow, which was cool and dense, remained close to the floor and exited through the outlet vent without appreciable mixing with the older air in the room.

Placing the outlet high on the wall solved the poor air-exchange problem. The glyphs showed that the air flowed through a larger portion of the room volume as it passed to the outlet vent.

Indoor Flammable Plumes from CNG Leaks

Buses are often stored and maintained in large transit facilities, which may hold a large number of buses. A concern with natural gas-fueled buses is that a leak could create a flammable atmosphere in the transit building. Knowledge of the size of the plume for representative leaks is very important for developing future ventilation standards.

We analyzed the dispersion of leakage plumes inside a typical transit building that was 119 m by 108 m by 5.5 m high. During our simulation, the building was fully occupied with parked buses and all doors were shut. The ventilation system was on and operated at a rate of 5 air changes per hour.

Two leak scenarios were investigated:

1. A rapid leak corresponding to a ruptured fuel manifold line connecting the CNG cylinders or the failure of a pressure relief device.

2. Slow leaks from a poorly fitting fuel line connection. The leakage rate was up to 2.0 g/s.

Because of symmetry, one-fourth of the room was modeled with FLUENT using a grid of 12,000 cells. The effect of using a coarse grid on a flow with a wide range of geometrical length scales was assessed with some preliminary calculations. We found that leaking gas that entered the region between buses was strongly driven toward the ceiling by buoyancy forces. The details of the flow under or within buses were not important in determining the overall evolution of the plume.

The flammable concentration was tracked in time using apE. Two iso-valued surfaces were constructed; one represented the minimum flammable concentration, and the other represented the maximum. Transparency property effects were used for the maximum isosurfces, so the flammable region was clearly depicted. The extent of the plume as a function of time was dramatically displayed with animation. Figure 3 shows a gray scale rendition of the fast-leakage-rate plume. The region between the dark and light surfaces of the plume represents the volume of the building with a flammable concentration of gas.

Particulate Deposition in Flow Systems

Particulate deposition and plugging in flow systems are important in a variety of industrial applications. We have simulated deposition in high-velocity gas

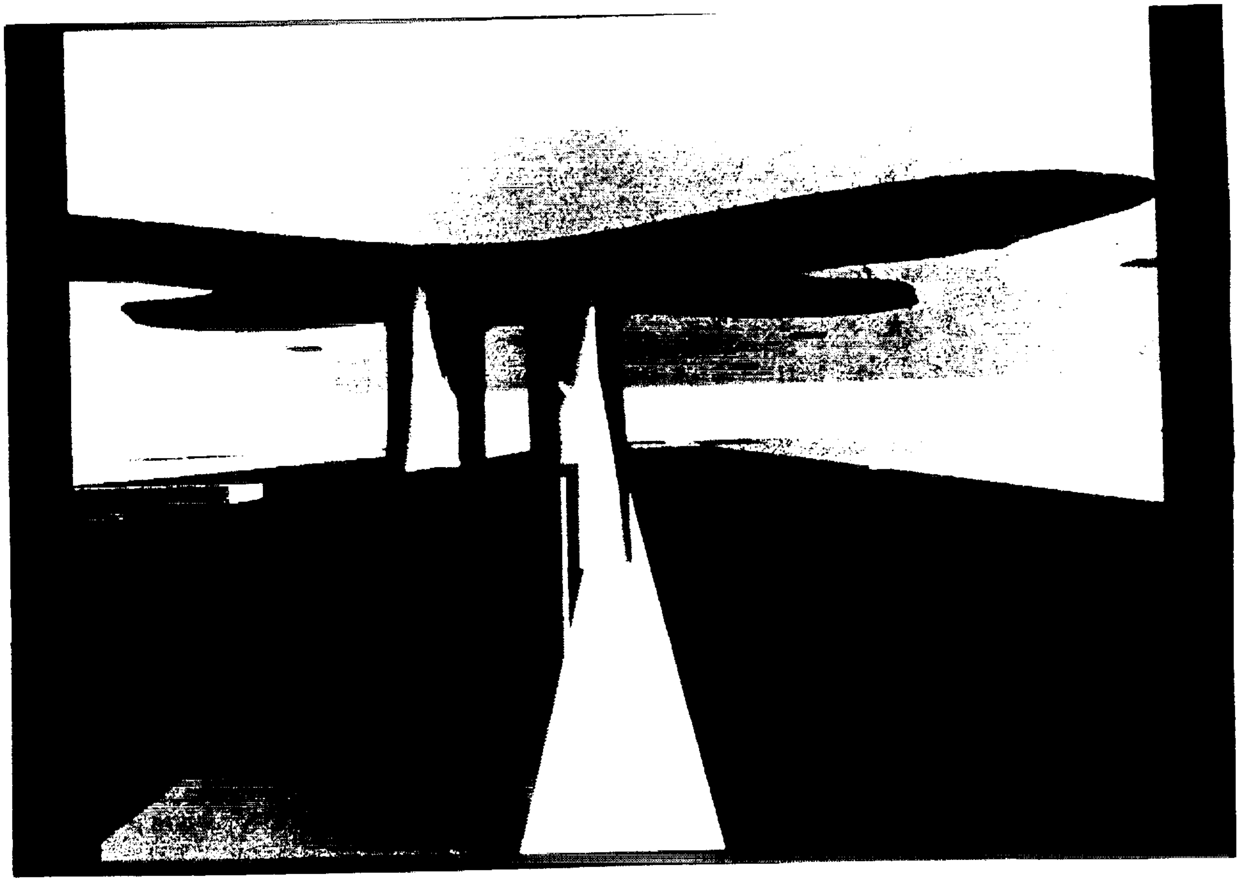

Figure 3. Rendition of fast-leakage rate indoor plume

flows where the flow is normal to a porous plate or collecting surfaces. Under these conditions, the particles travel in essentially straight lines without lateral diffusion or response to changes in the direction of the flow streamlines. Particles may collide with any surface they encounter. Because of the high velocity and high particle loadings, the deposit layer grows rapidly.

We modeled this process by tracking individual particles moving on a two-dimensional lattice as they form the deposit layer. Rules based upon the microphysics of the gas-particle-surface interactions determine whether a particle sticks to a deposit site, misses it, or bounces off. The computer algorithm displays the result on a high-resolution monitor, so that the development of the deposit can be observed continuously.

Our initial motivation for developing this technique came from studies on diffusion limited aggregation [ref. 6] and later from studies on ballistic deposition [ref. 7]. Near the end of our work we became aware of other work on similar deposition models [ref. 8] with extensions shown in [ref. 9].

Figure 4 shows the results of simulating particles depositing on a porous plate. Particles, which are assigned one or more pixels on the graphics screen, are released at a random location above the deposit layer and are tracked as they move in a straight line toward the deposit. Deposited particles are shown as colored pixels on the screen. At each timestep, the algorithm examines pixels that are in and near the immediate path of the particle. If a collision with a deposited particle is imminent, the sticking probability is computed for that set of circumstances. If the collision will be a frontal collision, the particle may either stick or bounce. If the collision involves the sides or corners of the deposit, then the particle may stick or pass by.

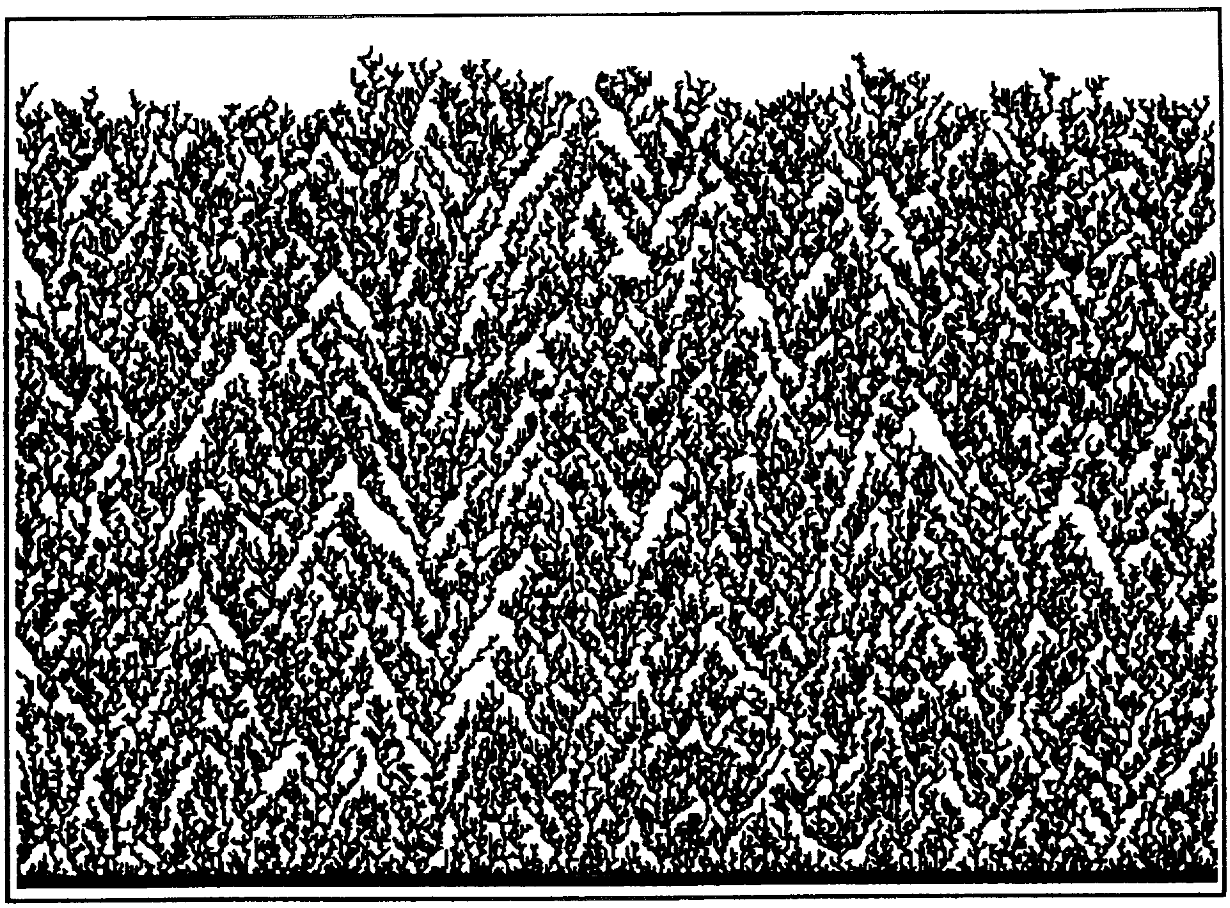

Figure 4. Simulation of particles depositing on a porous plate

Because of the continuously updated graphic display, the effects of rule changes can quickly be seen, allowing the researcher to evaluate the ramifications of the assumptions and develop an understanding of the role of the microphysics on the formation of the resulting structure.

Our work has shown that the resulting deposit structure is sensitive to the form of the rules [ref. 5]. However, rules can be estimated from the detailed microphysics and future research should focus on extending this ability. For instance, in high speed flows, lateral dendritic growth may be strongly limited by shear-induced breakage of the dendrites. Rule selection should therefore be guided by careful comparisons of the predicted morphology of the deposit structure with detailed experimental measurements.

Conclusions

Visualization has been shown to be an important part of three engineering research problems using hardware ranging from supercomputers to personal computers. In the room ventilation example, the visualization revealed the impact of vent placement on air mixing in the two rooms. The visualization of the flammable gas plume in the transit building analysis gives the ventilation engineer a much clearer indication of potential weaknesses in the ventilation scheme than would be possible with traditional techniques. The graphical display of particle deposition gives the researcher a unique perspective on the growth of particle beds and allows detailed investigations of the particle microphysics in these processes. These visualization techniques have many applications that dramatically increase the usefulness of scientific data.

References

apE is a trademark of The Ohio State University

FLUENT is a trademark of FLUENT, Inc.

1. Murphy, M. J., S. T. Brown and D. B. Philips, "Extent of Indoor Flammable Plumes Resulting from CNG Bus Fuel System Leaks," to be presented at the SAE 1992 Truck and Bus Meeting and Exposition, November 16-19, 1992.

2. Brown, S. T., J. J. Crisafulli, D. B. Philips, L. A. Southern and D. F. Knight, "Analysis of Vent Placement and Fan Operation for Air Conditioning in a Partitioned Room Using Advanced Visualization Techniques," submitted for presentation at ASHRAE Winter Meeting, Denver CO, 1993.

3. Saunders, J. H., J. J. Crisafulli and G. H. Stickford, "Analysis of Particulate Depositions and Plugging in Flow Systems," *ASME Fluids Engineering Conference*, Los Angeles CA, June 21-24, 1992.

4. Snyder, J. M. and A. H. Barr, "Ray Tracing Complex Models Containing Surface Tessellations," *Proceedings of SIGGRAPH 1987, Computer Graphics,* Vol. 21, No. 4, July 1987, pp. 119-128.

5. Lorensen, W. E. and H. E. Cline, "Marching Cubes: A High Resolution 3D Surface Construction Algorithm," *Proceedings of SIGGRAPH 1987, Computer Graphics,* Vol. 21, No. 4., July 1987, pp. 163-170.

6. Witten, T. A. and L. M. Sander, "Diffusion-Limited Aggregation, a Kinetic Critical Phenomenon," *Phys. Rev. Letters*, Vol. 47, No. 19, 1981, pg. 1400-1403.

7. Meakin, P., P. Ramanlal, L. M. Sander and R. C. Ball, "Ballistic Deposition on Surfaces," *Physical Rev. A*, Vol. 34, No. 6, 1986, pp. 5091-5103.

8. Tassopoulos, M., J. A. O'Brien and D. E. Rosner, "Simulation of Microstructure/ Mechanisms," *AIChE J.*, Vol. 35, No. 6, 1989, pp. 967-980.

9. Tassopoulos, M. and D. E. Rosner, "Simulation of Vapor Diffusion in Anisotropic Particulate Deposits," *Chem. Eng. Sci.,* Vol. 47, No. 2., 1992, pp. 421-443.