Green Tools: Computational Fluid Dynamics

CFD (often wryly referred to as “Colorful Fluid Dynamics”) is a building science tool increasingly employed by researchers and designers in the buildings industry to understand airflow and contaminant transport in buildings. Steven Winter Associates, Inc. (SWA) has used CFD in research as well as design projects for several years to simulate airflow in and around buildings. CFD2000, a commercially available software developed by Adaptive Research, is a general purpose and affordable CFD modeling software. How does it work? It’s designed to solve numerically the Navier-Stokes equations (partial differential equations) that are the fundamental physical governing equations of mass, momentum, and energy conservation for a fluid flow in a flow field (like air currents moving through a room). The program is capable of solving laminar fluid flows (such as low velocity water flow in a pipe) as well as turbulent fluid flows (airflow from a room air diffuser, for example).

Although CFD software has gotten easier to use, it still requires expertise in fluid dynamics and adequate training. SWA has found that ASHRAE Research Project 1133 [Ref.1] on “How to verify, validate, and report indoor environment modeling CFD analyses” is a valuable resource for performing CFD simulations accurately. A recent SWA project for an appliance manufacturer included a simulation of natural gas leak in an appliance room. This simulation studied the effect of infiltration on the distribution of methane to determine if there is a concentration that reached lower explosion limit. Figures 1 and 2 show the distributions of methane concentration. SWA also used CFD to understand airflows in an atrium at the award-winning School of Environmental Sciences of Oberlin College in Ohio (Figures 3 shows the behavior of natural ventilation).

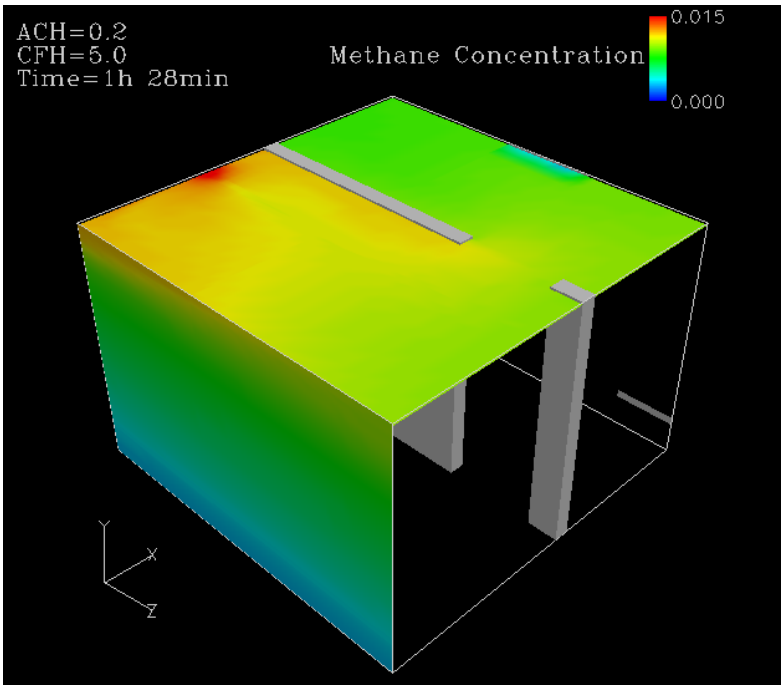


Figure 1. Natural Gas Plume Concentration Profile at Ceiling
(Leakage Flow Rate at 5.0-SCFH)

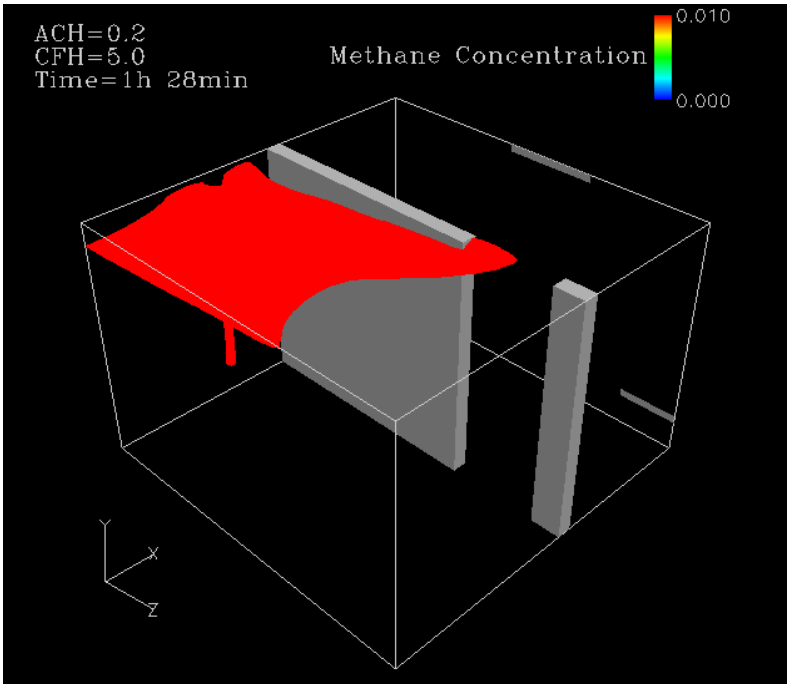
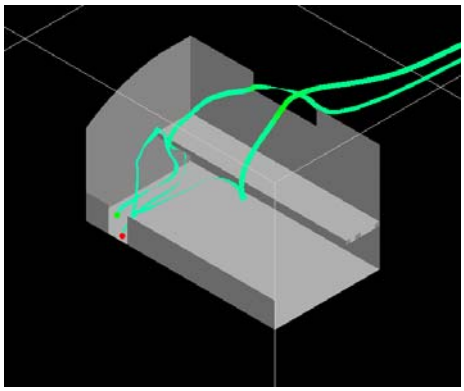


Figure 2. Natural Gas Plume Concentration Profile at Ceiling (1% Profile)
(Leakage Flow Rate at 5.0-SCFH)

CFD Simulation of Natural Ventilation in the Atrium of Oberlin College
Steven Winter Associates, Inc.



Airflow Streamlines from One Inlet

Figure 3. CFD results showing path of air flow seeds at the entrance and their streamlines

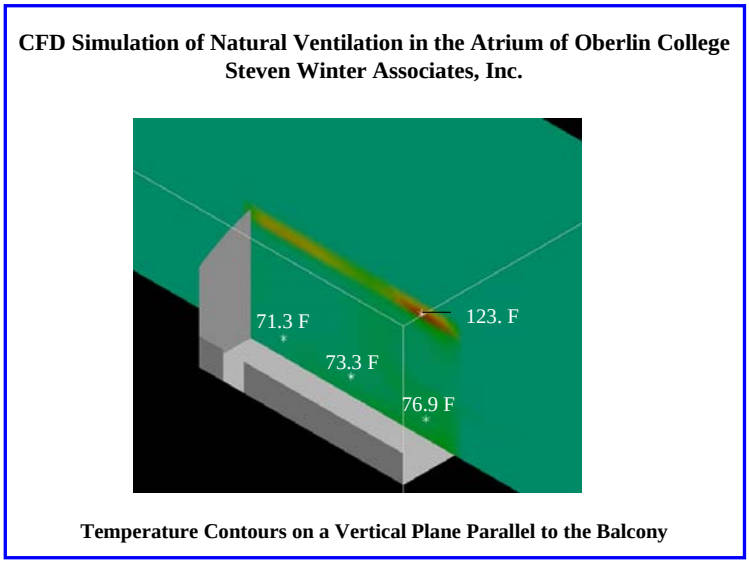


Figure 4. CFD results indicating a hot spot near the ceiling.

Reference:

[1] Chen, Q. and Srebric, J. 2001. How to verify, validate, and report indoor environment modeling CFD analyses. Final Report for ASHRAE RP-1133. Atlanta, GA.