Chapter 1

Using CFD to Study Air Quality in Urban Microenvironments

J.D. McAlpine and Michael Ruby

Envirometrics, Inc.
4803 Fremont N, Seattle WA 98103
jdmcalpine@ envirometrics.com, mruby@ envirometrics.com
website: http://www.envirometrics.com

Abstract

The study of building effects on air quality has grown beyond stack plume downwash. The relative placement of air intakes and exhausts and cooling tower exhausts on buildings can significantly affect the indoor air quality. Earlier studies of effects on building air intakes have been limited to relatively simple situations, unable to treat the complex envelope of most buildings and building groups. Computational Fluid Dynamics (CFD) is becoming available as a tool to assist with modeling the airflow and dispersion of pollutants among complex urban geometries on the scale of a section of a building exterior up to several city blocks. This tool allows more accurate predictions of impacts over a range of meteorological scenarios and alternative building designs and placements relative to roadways and other pollutant sources. Recent projects are discussed to illustrate the capabilities of CFD in modeling urban microenvironments. The steps in a CFD application are presented including geometry and mesh creation, simulation of meteorological conditions, handling of pollutant sources, and post-processing visualization. The benefits and shortcomings of this approach are also discussed.

Introduction

Recent studies have found that urban Americans spend about 90% of their time indoors (USEPA, 1995). Most of today’s commercial and institutional buildings have mechanical air handling systems that are designed to provide proper ventilation, to ensure indoor sources of pollutants are quickly vented to the outside. It is equally necessary to design the systems so the source outdoor air is free of odors or contamination. If the source air is contaminated, the effects can range from odor complaints to serious illnesses such as Legionnaire’s disease. The positioning of air handling units and exhaust stacks on buildings should be carefully analyzed to ensure that building inhabitants are not exposed...
to irritants, carcinogens, and odors that originate from outdoor sources or the building’s own exhausts.

Studies of airflow around buildings generally involve the positioning of cooling towers, exhausts and air-handling units. These projects have ranged from assisting architects with the placement of large cooling towers and air-handling units to addressing odor complaints due to exhaust sources near to air intakes.

Until recently, analytical methods were most often used to model the airflow in these types of projects. The ASHRAE static model is commonly used to determine roof recirculation and turbulence zones (Wilson, 1979) and downwind building recirculation zones (Fackrell, 1984). A simple picture of the descriptive "zones" which can be calculated by this method is included in Figure 1. Consultants have also utilized theatrical smoke releases in existing situations to illustrate the flow of sources to intakes. Wind tunnels have also been popular tools in the industry to model airflow and pollutant dispersion in scale models of urban environments, especially for proposed buildings.

![Figure 1: Illustration of building wake boundary, recirculation zones, turbulence zones, and streamlines calculated by static analytical methods. (Wilson, 1979)](image)

Each of these traditional methods has shortcomings that call for a better approach. Simple mathematical methods can not account for complex urban spatial relationships or meteorological conditions. Theatrical smoke release can achieve this, but can require an extended effort to capture more than one or two conditions and can not yield quantitative results without intricate instrumentation. Theatrical smoke is also limited to examination of nearby existing conditions. Wind tunnels offer more control, but are quite costly. Meteorological conditions are difficult to imitate in the wind tunnel environment because arrangements of upstream obstacles must be placed strategically to simulate incoming turbulence.
Recently, a new tool has become more widely available that overcomes many of the shortcomings of traditional methods. Computational Fluid Dynamics (CFD) has a long history of use in computing fluid flow around obstacles (Anderson, 1995). As it has become more available in the desktop-computing environment CFD has been applied to a wider range of problems, most recently urban microclimates. With CFD the urban spatial geometry can be represented in computer simulation with reasonable accuracy, including more than just the building surfaces. Topography, vegetation, and mechanical features such as stacks and air intakes are easily described. CFD also offers the ability to experiment with solutions by making it easy to alter site geometry and features or stack and inlet parameters. A host of meteorological conditions including wind directions and magnitudes, temperatures, and atmospheric stabilities can be simulated in a CFD project in a reasonable time.

Using CFD as the main tool in addressing these problems has been successful in recent projects. These projects have involved using the tool to assist with stack and cooling tower positioning, air intake positioning and odor source diagnosis. Details of several of these projects are discussed in this paper.

As with all computer simulations that are used to model real world phenomena, validation of the results is always crucial. However, this may be difficult for CFD because of the complexity of flows in the atmospheric surface layer and wide varieties of problems that may be addressed with CFD. The results of a few simple methods to attempt some qualitative validation of the modeling results are presented at the conclusion of this paper.

**CFD Availability**

With the advent of fast powerful desktop computing, the ability to model complex computational fluid dynamic problems in economical timeframes has increasingly become a reality. In the past, large mainframe computers were required to compute CFD problems. Now, using an efficient commercial CFD package, a problem can be solved in a matter of hours on a desktop computer. We have found that a computer with 1Gbyte of memory and operating at 1800 MHz on a single processor is adequate to provide runs in a reasonable time. For example, a model in rectangular Cartesian coordinates with approximately one million cells solving only the wind fields took about 6 hours to run for 4 minutes of model time.

Many commercial CFD packages are now available that offer a variety of features, but the main aspects of the CFD approach is rather similar for all packages. First, a package has a geometry creation and visualization program where the CFD problem is set up. The project geometry and mesh including buildings, topography, vegetation, and boundary conditions such as air inlets, outlets, and exhaust stack parameters are all established in this program. When the model parameters are all defined, the CFD project is sent to a second program, the solver. In the solver, the equations of fluid motion are solved for the given geometry using a time-marching technique. Since the steady state solutions to the
equations of motion are parametric, a time-marching technique, given adequate initial conditions, can solve for a steady state solution. Finally, the solution is loaded into a visualization program, where streamlines, vectors, scalers, and a variety of other features can be graphed to observe the results.

The commercial CFD package we are using is CFD2000 by Adaptive Research, which is equipped with its fast, efficient STORM solver (http://www.adaptive-research.com). This program provides the classical first-order closure scheme, the k-\(\varepsilon\) turbulence model to account for steady state turbulence and advanced numerical schemes to solve CFD problems efficiently. Several types of k-\(\varepsilon\) turbulence models are available to use with the solver including a Re-normalized Group (RNG) model. The most recent version of CFD2000 also offers a Large Scale Eddy model, but we have not yet tested it.

The Equations of Motion

The basic equations of motion are known as the Navier-Stokes equations. They are a set of equations that relate the conservation of mass, momentum, and energy. Initially we have five equations (the continuity equation, the conservation of momentum equation in each of three dimensions, and the energy equation) and seven unknowns (pressure \([P]\), density \([\rho]\), temperature \([T]\), x-component velocity \([u_1]\), y-component velocity \([u_2]\), z-component velocity \([u_3]\), and internal energy [expressed in terms of the enthalpy \(H\)]). The continuity equation for an open Newtonian system with possible sources or sinks \(S_{mp}\) for mass at a point within the system can be written:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = S_{mp} \tag{1}
\]

where the subscript \(i\) identifies the directions through the faces of each sub-volume of the system. Similarly, the three, separable equations for the conservation of momentum can each be written:

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_i} = \frac{\partial \tau_{ij}}{\partial x_i} - \frac{\partial P}{\partial x_i} + \rho B_i + S_{mp} \tag{2}
\]

where \(B\) includes body forces on the sub-volume (e.g., gravity), \(S\) is again source or sink terms and \(\tau_{ij}\) is a collection of cross product terms of velocity differential components identified as the viscous stress tensor. Finally, the conservation of energy equation can be written for a fixed sub-volume as:

\[
\frac{\partial \rho H}{\partial t} + \frac{\partial (\rho u_i H)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \frac{\kappa}{C_p} \frac{\partial H}{\partial x_i} \right] + \frac{\partial P}{\partial t} + \frac{\partial u_i}{\partial x_i} \frac{\partial P}{\partial x_i} + \Phi + Q_p + S_{Hp} \tag{3}
\]

where \(\Phi\) is another collection of cross product terms, identified as the Stokes molecular dissipation function, \(Q\) is the rate of energy added as heat to the sub-volume and \(S\) is a source or sink term for enthalpy. The additional terms \(\kappa\), the thermal conductivity, and
C_p, the specific heat, which are measured data for a specific problem, are necessitated by our use of H instead of the internal energy.

Such a system of equations is not solvable without additional assumptions. We can assume a relationship between the fluid density and the other thermodynamic variables. In our case we use the perfect gas law, T = P/ρR, where R is the gas constant, which is reasonable for air at the surface of the earth. In addition, we can assume that we are able to express the conservation of turbulent kinetic energy k and the dissipation rate ε of turbulent kinetic energy with similar transport equations (usually expressed in a more complex form than would be useful to reproduce here) that relate back to the other equations through the stress tensor τ and the molecular dissipation function Φ and do not include any more unknowns because we are able to assume values for k and ε that are constant across the system.

With the five initial equations, the assumption of the perfect gas law and the k-ε assumption we now have a system of equations that can be solved. Any number of scalar transport equations as well as chemical reaction equations that couple to this set of equations can also be used in the solution.

The interesting problems for CFD involve moving air and obstacles to flow which require the flow to have zero velocity at the object wall. This results in a gradient in the velocity across the flow field even with no changes in the air direction or velocity. Similarly the real atmosphere under gravitational force yields a pressure gradient that must be considered even at the small scale of urban buildings. The perfect gas law assumption is then valid only locally, which limits the size of the solution space to the microscale.

The most useful, steady state solutions will be in dynamic equilibrium averaged over both space and time. The model does not provide information about the range of fluctuations about this equilibrium solution. Reynolds (or “ensemble”) averaging is applied to the equations to derive the steady state solutions. With Reynolds averaging the instantaneous value of a variable is assumed to be the sum of the mean value and a perturbation from that value. For example, if \( u(t) = \bar{u} + u'(t) \) then the steady state solution is the mean of this quantity, or \( \bar{u}(t) = \text{mean}(\bar{u} + u'(t)) \). Replacing each velocity variable with the Reynolds averaged variable results in the Reynolds averaged Navier-Stokes equations, which can be solved for the steady state flow. The vector cross products generated when the mean value is taken of the velocity perturbations are known as the Reynolds stresses. It is these cross product terms that allow information, for example about the location of a wall, to be disseminated throughout the model space. Turbulence modeling is concerned with accounting for these terms and the viscous dissipation term. The k-ε equations assume isotropic eddy-viscosity, which simplifies the viscous dissipation term.

The method of parameterized Reynolds stresses and viscous dissipation by turbulent kinetic energy transport is a popular one that has been used rigorously, mostly due to the comparatively lower computational burden of the method. The two turbulent kinetic energy and dissipation equations replace a complex numerical representation of the flow, which would be difficult to model. Further, isotropic eddy-viscosity is a good assumption
in lower velocity atmospheric boundary layer flow. Though this method is not perfect, and inaccurate in certain situations, it has proven to be sufficiently accurate in representing the steady state flow of situations where the length scales of velocity and obstacles are much greater than the length scales of turbulence.

**The Focus of a Local Scale Analysis**

On the local scale, in the urban microclimates around buildings and sets of buildings, we can identify six pollutant sources that are of the highest concern. The majority of CFD projects performed to address HVAC positioning, health or odor complaints, or exposure analyses will focus on one, several, or all of these. They are, in no particular order: lab hood exhaust, cooling tower exhaust, automobile exhaust, odor sources (from kitchen stacks, garbage storage, industrial exhausts, etc.), allergens such as grass clippings or cigarette smoke, and general building exhaust. Little regulation beyond the common law of nuisance exists for these pollutants or the conditions that make them a problem. Vague guidelines for exhaust stack heights, filtering, and safety are incorporated in different standards documents but no one standard can be applied for the world of circumstances that can occur at the local scale. Also, these types of exhausts effect different people in different ways. For example, an allergen source near an air intake may have a dramatic impact on sensitive individuals while non-allergic persons have no notice. An odor from a kitchen stack may be quite pleasing to one passing on the street, but may be highly objectionable to the nearby office worker exposed to the exhaust for hours on end. Some details of the handling of these exhausts in the CFD modeling follows.

*Lab hood exhaust stacks*

Exhaust from lab hood stacks can potentially be harmful depending on the agents used in the lab. Various chemicals with toxic or acidic properties may be ejected from these stacks. In some cases, biological hazards may be emitted from similar stacks. In any case, this exhaust may be the most dangerous exhaust element of this scale and being so, it is very important to keep this exhaust away from air intakes or any place where the public may encounter it.

The volumes of these stacks are often quite small (ranging from 1,000 – 10,000 cfm). They are often small in diameter, ranging from 6 inches to 2 feet and are often placed in groups or in lines along building roofs. The relatively low volume of flow means that the effective stack heights (physical height plus momentum and buoyant plume rise) of these exhausts may be quite low during stronger winds. Thus, the heights of these stacks often need to be quite high to avoid entrainment into recirculation zones which may bring the exhaust to the surface or concentrate it in regions near intakes, etc. CFD can assist with the analysis to determine placement and height of these stacks.
Odorous sources

Odorous sources are highly variable in their make-up. The most common of these are often stacks similar to lab hood exhausts that emit kitchen odors, lab odors, locker room odors, or industrial odors. Airflow volumes from these stacks can be quite large (>30,000 acfm) such as those from larger kitchens or industrial sources. Other sources of odor may be from street level garbage bins or diesel exhaust from idling vehicles.

The impact of each odor source is highly dependent on the amount of dilution of the exhaust that is needed before the odor is not offensive. However, this is a highly subjective area, considering that odor sensitivity varies considerably throughout the population. Standard descriptions of odors are based on the number of dilutions of the original source odor that are needed to render the sample gas odorless (that is, below the threshold of detection) to a majority of a panel of seven to thirteen average individuals. Thus, an odor strength may be cited as “1000 dilutions to threshold” (d/t). Typical odor strengths for an industrial source might be 2,000 to 5,000 d/t, while a kitchen exhaust might be 5,000 to 10,000 d/t. Because this represents the odor strength evaluated by 50% of the population it may be necessary to go to more than twice the dilutions of the measured detection threshold to avoid notice by all but a very few in the general population.

The best way to handle these types of odors in CFD modeling is by examining the dilution of the exhaust at locations throughout the CFD domain from the concentration at the initial source. Exhaust dilutions can then be compared to the d/t measured in odor studies to estimate impacts. For visualization purposes, different outlines of the plume can be illustrated at various dilution levels of the exhaust to assess areas of impact.

Motor vehicle exhaust

Emissions from motor vehicles are one of the main sources of pollutants in the atmosphere, contributing to emissions of nitrogen oxides, carbon monoxide, benzene, and many other chemicals. Therefore, it is ideal to place intakes away from busy intersections or roads where exhaust may build up in adverse traffic or wind conditions. Street canyons can concentrate motor vehicle exhaust under certain wind conditions or funnel high levels of exhaust to sensitive regions. CFD can simulate street canyons and other recirculation zones to assist in the placement of the air intakes of a building on the street.

Simulating motor vehicle exhaust in the CFD environment is difficult. Tailpipes from vehicles are placed at different heights depending on the type of vehicle and diesel engines have different kinds and amounts of emissions from gasoline engines. Also, vehicle movement creates its own airflow that disperses the pollution itself. This poses a challenge in CFD modeling. Perhaps the best way to account for this is by using a volume source in the CFD domain, with emissions calculated from the EPA vehicle emissions models (e.g., MOBILE 6). If only plume path needs to be analyzed, an array of plume streamlines released from the traffic exhaust volume region may be adequate.
Cooling tower exhaust

At first thought, cooling tower exhaust seems a harmless aerosol, consisting of water mist. However, the possibility of bacterial growth in the cooling tower water stream makes it a possible source of contamination. The infection of a cooling tower by bacteria makes it a biohazardous source. Steps taken to reduce the chance of bacteria infection may make the tower a chemically hazardous source depending on the antibacterial agents used in the stream. Ammonia, bromine, and chlorine are common chemicals used to treat cooling tower water which make the exhaust potentially hazardous to nearby receptors.

Mist drift from cooling towers has been implicated as a source of the infectious proteobacteria *legionella pneumophila*. The conditions in cooling towers can be ideal for the growth of *legionella*, which is present in low concentrations in most water supply systems. The conditions which promote growth of the bacteria are

- water temperatures between 95 and 115 degrees F
- sediment and food sources in the water which support the growth of algae, protozoa, etc.
- the presence of l-cysteine-HCl and iron salt

*Legionella* belongs to an unusual group of bacteria with special properties that can defeat the respiratory disease response system, a group which includes tuberculosis and salmonella. *Legionella* is widely distributed and occurs in five different varieties. Infections commonly appear with only two of the forms, one occurring relatively infrequently but manifesting as a mild respiratory disease in approximately 95% of those exposed and the other a more troublesome form that only matures in 2 to 5% of those exposed. The milder form causes flu-like symptoms that pass in less than a week. The other results in severe symptoms, often requires hospitalization and is fatal in about 10% of the cases.

In the best known cases of disease outbreak (an American Legion convention in Philadelphia and the Oakland County Health Department in Pontiac, Michigan) the building air intakes were close to the infected cooling tower. However even with significant separation it is judicious to determine the likelihood of cross-contamination in order chose relative locations that will minimize the opportunity for infection.

Because a cooling tower uses sprays of water to cool the working liquid, the exhaust air from a cooling tower contains fine droplets of water, called mist, that can drift with the exhaust air away from the cooling tower in a plume. If the cooling tower water has developed a *legionella* growth, the mist will contain the bacteria. The mist will evaporate quickly in warm dry conditions, but may remain as droplets for quite a distance in humid conditions. Even when dried, *legionella* can retain its infective capability.
Concentration Guidance and Visualizing CFD Results

There are several effective ways to visualize the results of your CFD modeling either for analysis, verification, or demonstration. Perhaps the most effective way to visualize dispersion in the CFD environment is by examining dilutions of the exhaust from a source.

In most cases, actual concentrations of harmful entities in the exhaust, such as acid in a lab hood exhaust stack plume or bacteria per cubic meter of cooling tower exhaust, will not be known. Stack test data from similar sources is usually not available as the exhaust makeup is often unique.

If the concentration of, for example, a release of sulfur dioxide in a lab fume hood exhaust is approximately 0.5%, a high but not unreasonable concentration for short period discharges during certain procedures, a dilution of only 1,000 times would reduce it to 5 ppm, at approximately the industrial hygiene limits for an eight-hour exposure and 10 times the recommended short-term exposure for sensitive members of the public. Thus a dilution of only 1,000 averaged across an entire air intake (assuming 100% outside air circulation) or in an area of public exposure would be considered unacceptable and a dilution of more than 10,000 times would be marginally acceptable.

For a kitchen exhaust stack, an adequate dilution is somewhat debatable, being highly dependent on the type of food being cooked and the nature and sensitivity of the receptor. Kitchen exhaust itself is not harmful and often even has a desirable odor, especially as advertising at lunchtime. However, a sensitive individual working at his or her desk may find an odor to be unbearable after repeated exposures. Even mild coffee odors from roasters lead to vigorous complaints to air quality agencies. As a general rule, a stack from a kitchen using garlic, soy sauce or high in grease, such as an Asian or a hamburger grill would require about 5,000 dilutions to prevent significant complaints and 10,000 dilutions to avoid detection by most of the population. A more standard kitchen exhaust from well maintained systems would require 1,000 dilutions to avoid significant complaints.

The necessary dilution of cooling tower exhausts to avoid exposure to *Legionella* bacteria from an infected cooling tower has not been established. Obviously at a higher level of contamination, a greater dilution would be needed to minimize the possibility of exposure. The U.S. Centers for Disease Control have recommended treating cooling tower exhaust as potentially infective even at dilutions greater than 100,000. It is not known how conservative this is. Clearly proper maintenance of cooling towers must be the first line of defense but proper siting can minimize the problems that might occur if an infection arises in spite of ongoing maintenance.

In some CFD cases, the volume of gas from a source is too small or the resolution of the CFD model domain is too great to incorporate the plume from a stack. In these cases, it is necessary to model the plume centerline using the visualization of a streamline originating from the effective release point of a stack. The “effective” stack height
accounts for rise due to mechanical lift and thermal buoyancy of the plume after release. Simple analytical plume Gaussian dispersion models can be sources of guidance for the plume dilution at certain points if properly handled. The stack parameters and average wind speed of the plume throughout the CFD modeling domain can be used to assess a dilution at downwind points.

Successful Projects Using CFD

CFD can be used to examine the movement and dispersion of pollutants around buildings for length scales of tens of meters up to hundreds of meters. So far, CFD has been used successfully to analyze the flow around buildings and building clusters in urban microenvironments. From our experience, the more common types of projects can be divided into two classes. The first type of project involves cases where the client is dealing with current odor or contamination complaints inside or in sensitive areas outside an existing building. The second type involves projects where the client seeks consultation on the placement of air intakes or exhaust stacks to avoid future odor or contamination problems. Several of both types of projects that we have conducted are discussed in this section.

Though model geometry is simple in several of these cases, the possible complexity of the geometry in a CFD case is unbounded. More complex geometry, including minute details, takes more time to construct and thus must only be included in the model when necessary. The most critical region in the model domain is often the source or receptors and the region surrounding each. The shape and dimensions of a stack or other source must be established correctly to ensure the parameters of the stack gas flows are as near to reality as possible. Building surfaces or topography far away from the regions of interest in the model can have less detail as long as the necessary dimensions and surface roughness are accounted for to properly simulate wind flow in that region of the model.

*Chemical analysis lab exhaust near building air intake.*

In this project, building occupants were complaining of odors in their offices, which were located in a building housing several chemical analysis labs. Most of these labs contain fume hoods, which were vented out short stacks to the building roof. Several of these stacks were very near to the two air intakes for the office areas, in an area of complex building geometry, as can be seen in Figure 2.

When the odor complaint log was compared to meteorological conditions, it was observed that many of the chemical odors occurred during the periods of southwest and northwest winds. Complaints in rooms served by the south air intake corresponded to periods of northwest winds, which are rare winds for the region. Complaints in rooms served by the north air intake corresponded to periods of southwest winds, which are common winds for the region.
For CFD modeling, the exhaust stacks, the air intakes, and the three most prevalent wind directions during complaint periods were modeled in a simple geometry domain. All positions of exhaust stacks were noted and visualized using unit emissions and/or plume centerlines. Considering the light color of the building roof, building roof heat was not considered to have a large impact on the flow, so was not included in the model. For each wind case, neutral atmospheric conditions were assumed.

![Figure 2: Plume centerline streamlines under light southwest wind. Several of these plumes travel across the building roof, and down into the air intake (the red square on the side of the building).](image)

The modeling results showed that several of the exhaust plumes directly entered the air intake. Plume-centerline visualization showed that the plumes from several exhaust stacks were completely taken up by the air intake, as seen in Figure 2.

To alleviate this problem, several alternatives were suggested. The best alternative was to move the air intake out of a re-circulation “well” that forms in the area where it is currently located. Another alternative was to raise the height of the exhaust stacks above the building top roof. Both these alternatives are illustrated in Figure 3. (The stacks are shown with a square cross-section because the model was created as all rectangular solids to speed execution time.)

The green dot on the top of the exhaust stack represents the existing arrangement, with a streamline originating there which enters the air intake. The red dot, positioned directly above the green dot, represents the elevation of a raised stack which will allow the exhaust to avoid the re-circulation zone and travel over the top of the adjoining roof.
Positioning of lab exhausts and cooling towers for a new building.

In this project, two new buildings, including a medical research facility, were proposed to be constructed together on a sloping hillside. These buildings required air intakes, cooling towers, and vivarium (i.e., housing for laboratory animals) and lab hood exhausts as part of their HVAC design. CFD modeling was used to assist with the proper positioning of the HVAC elements to lower the risk of exhaust entering the air intakes.

Figure 3: CFD modeling with alternatives included. The air intake (in red on the near side of the building on the left) is shifted to the left, and the most important exhaust stacks are raised to adequate heights to avoid the building re-circulation zone. In this figure a northwest wind is shown.

CFD modeling confirmed that the initial stack height was adequate for the lab hood stacks. Exhaust from the hoods remained well above the ground, avoiding sensitive receptors such as pedestrian walkways and air intakes. However, the initial configuration of the vivarium exhausts and cooling towers allowed the exhaust to reach the air intakes.

The vivarium exhaust points, potentially carrying significant levels of odor, reached the air intakes at around 300 dilutions, as illustrated in Figure 4 on the following page. The CFD model found that the initial design, which vented the exhaust from roof-level louvers, would not lift the exhaust out of the roof and lee re-circulation zones of the building. CFD was used to test different alternatives, leading to a final solution of venting the exhaust out taller stacks, which allowed the plume to avoid the air intakes.
Air intakes near busy road and truck/bus-idling location

In another project, odors from outside had become a nuisance indoors. The building was completely climate controlled, with the high capacity air intake near to a busy intersection with a high volume of diesel traffic. In addition, a bus stop was located directly adjacent to the intake and the far lane of the street was a sharp uphill ramp for traffic. Building occupants throughout the buildings were complaining about diesel exhaust odor in their offices. Site visits and building occupant observations confirmed that the truck traffic required increased power on the uphill ramp, emitting large amounts of diesel exhaust. Also, delivery and other trucks used the bus stop and adjacent sidewalk as a parking and idling spot.

![Figure 4. Plume centerline streamlines (red streamers) originating from vivarium exhaust louvers, flow freely at roof level and do appear to stay at that elevation. However a portion of the dispersing edge of the plume is entrained in the lee recirculation zone and travels down the wall. A measurement plane is placed directly above the surface air intake of the building (the red and yellow streak at ground level on the near side of the building) to measure the concentration of exhaust entering the intake. Exhaust enters the building at a dilution of around 300, a potentially high odorous concentration. (The measurement plane only reports in concentration units, the inverse of dilution. So the 0.0025 on the red end scale is 400 dilutions while the 0.0017 in the yellow is 588 dilutions.) CFD was used to analyze the significance of the various exhaust source areas and to analyze alternatives that might relieve the impacts. Air movements around the building and from the potential source areas were modeled for several wind directions. The complaint record indicated that rare easterly winds were very common when complaint were made, so careful attention was directed to the easterly winds. For all winds, neutral atmospheric conditions were assumed. A standard sharp wind profile for neutral conditions and high ground turbulence due to the presence of many obstacles was assumed. Visualization was performed of plume centerlines released from a typical truck exhaust height at these source areas. The source areas examined included a nearby loading dock,
idling locations on the adjacent access road, a stop sign/acceleration zone near the intake, the uphill ramp, the bus stop idling zone, and several locations along the main road.

The CFD modeling demonstrated that only in certain of these locations did the exhaust plumes reach the air intake under a variety of wind conditions, and most especially during periods of winds common when complaints were recorded. An example visualization of the results is provided in Figure 5.

Figure 5. An illustration of results visualization showing flow vectors, pressure anomalies, and plume centerlines. The plume centerlines travel directly from nearby acceleration zones (marked by the red and green dots at the end of the ribbons) into the air intake (represented by the red squares on top of the structure adjacent to the main building). The yellow blocks included in the modeling were porous objects representing trees along side the air intake. The coloration along the ground represents surface atmospheric pressure, with the red area at higher pressure than the yellow and green. The vectors on the ground illustrate the direction of air flow at each cell along the ground surface.

Several alternatives were proposed for the existing design. Modeling was required to test some of these options. For one alternative, raising the height of the air intake structure to lift it away from the exhaust plumes was examined. CFD modeling showed that this alternative was would not solve the problem as the air flow would simply rise up the side of the intake carrying the plumes with it, as demonstrated in Figure 6 on the next page. In order to prevent the extra lift created from a single mass intake extension, several design modifications are modeled. The modeling identified a design which utilizes an “organ-pipe” configuration by extending the intake opening to the region of clean air well above the street as potentially beneficial. The space between the pipes allows for the flow of air through the obstruction, limiting the lift that was seen with the total extension alternative. An illustration of this design is shown in Figure 7 on the next page.
Figure 6: Plume centerlines, released on the road, enter the alternative raised air intake.

Figure 7. Alternative design of air intake in CFD modeling. Illustrated is the flow of a plume around and through the intake structure, avoiding lift into the intake opening. Measurement planes were placed on top of the air intakes. The blue in the scale shows 0.00001, which means the pollution was at 100,000 dilutions.
Assisting architects with the placement of air intakes and cooling towers on new buildings in a building complex.

In this case, several new buildings are being added to an existing building complex. CFD gives us the ability to examine future conditions and also to easily analyze different arrangements of building air intakes and exhausts. This project is about as large as can be modeled using a desktop computer. An illustration of the modeling domain can be seen in Figure 8.

Many exhaust points and sensitive receptors had to be addressed due to the number of buildings in the domain. Cooling tower exhaust points, kitchen, and a diesel emergency generator had to be carefully modeled to address any impacts and avoid cross-over from exhausts to air intakes.

A number of meteorological conditions needed to be examined to cover the range of possibilities. Exhaust from nearby roads and freeways were also examined. Though not related to air quality, pressure forces and air velocities in critical areas were also examined to ensure that the building configuration did not create any dangerous conditions for pedestrians and to determine the types of hangers needed for wall ornamentation. These are some of the many features of CFD that can be applied to assist architects and city planners.

Figure 9 illustrates the impacts from a set of cooling towers on one of the complex buildings on its own air intake. Impacts were significantly low for treatment chemicals, but of some concern for bacterial agents. CFD was used to assess the impacts of certain
alternatives such as raising the cooling towers to eject the exhaust above the recirculation zones.

Figure 9. Two views of cooling tower impact at a building air intake. The blue and blue-green screen in the top view shows the roof-level exhaust readily entering the air intake. The bottom view shows the raised cooling towers, which reduced the impact of the cooling tower exhaust. Note the change in the scale of the measurement plane between the two views. For the upper view the blue-green portion represents only 125 dilutions while in the lower view even the red portion of the measurement plane represents 10,000 dilutions.

Odor from a kitchen stack impacting air intakes

In another project, remodeling of a building included the construction of an Asian grill, which was expected to exhaust at a source odor of 7,500 to 10,000 dilutions-to-threshold. The exhaust stack from the restaurant is located on the roof of the building. An adjacent building is home to offices and is served by air intakes in the roof penthouse. It was necessary to place the stack properly to prevent constant uptake of restaurant exhaust into the office building.

In this project, dilution levels of the exhaust were examined at the air intakes on the office building and an adjacent residential building. In Figure 10, the plume is visualized with the various surfaces represented at isopleths of dilutions from initial stack
concentration. The interior yellow surface represents the 2,000 dilutions isopleth and the red exterior surface represents the 10,000 dilutions isopleth. The modeling helped demonstrate the height of stack needed to avoid impacts at air intakes on both buildings.

Figure 10. Exhaust plume from kitchen stack showing size of plume. The inner 2,000 dilution isopleth is represented in yellow and the 10,000 dilution outer isopleth is represented in red. The entire office building surface is treated as a measurement plane to determine odor levels at critical points along the building surface such as air intakes and operable windows.

A Comparison of CFD to the Standard EPA Model ISCST3-Prime.

The most widely used air quality model is the U.S. Environmental Protection Agency’s Industrial Source Complex Short Term Model 3 (ISCST3). This model is used to address the air quality impacts from stationary sources to demonstrate compliance to permitting regulations. While the model is useful for estimating pollutant concentrations beyond the zone of building wakes it is not intended for analysis within a recirculation zone nor can it account for air movement affecting the path of a plume caused by obstructions downwind from the source complex.

In a recent project, where CFD was used to address the level of odor from a building exhaust at the air intake of a neighboring building, the regulatory agency requested ISCST3 modeling to compare to the CFD modeling. The results of this modeling demonstrated how much more effective CFD was at determining the impacts due to its advanced handling of airflow around structures. The ISCST3 modeling could not account for the upward jet at the face of the receptor building caused by the high-speed wind directly hitting the face of the building. CFD handled this feature nicely, as illustrated in Figure 11. The result was a substantial over-estimate of concentration at the air intake with the ISCST3 model. CFD modeling demonstrated that odorous exhaust was entering the intake at nearly 43,000 dilutions, where the ISCST3 modeling indicated odor entering the intake at 950 dilutions. This particular model, along with a comparison of both the
CFD and ISCST3 results to the EPA model AERMOD, has been described in detail by Ruby and McAlpine (2004).

![Figure 11](image1.jpg)

**Figure 11.** An aerial view (top) and cross-sectional view (bottom) of plume movement over a building with roof air intakes (right building). The upward jet at the leading edge of the second building that pushes the plume up and over the building is not accounted for in ISCST3.

### Design of Exhaust Systems

Another application of CFD in the air quality field is in the design of exhaust systems. In one project, concern was raised over the accuracy of assuming two plumes merged quickly into one effective plume. Two exhausts from a process were oriented at right angles, one horizontal, with the other directly under it and vertical. It was proposed to model a single vertical plume. CFD demonstrated that the plumes did merge quickly and could be considered a single plume for dispersion modeling. An illustration of the results is provided in Figure 12 on the next page.
An Example CFD Project

Setting up and executing a CFD project consists of several steps: creating the model geometry and mesh, model setup and execution, and visualization. An example CFD case is analyzed here to sample the CFD process. Some of the basic “rules of thumb” of CFD modeling are also discussed.

In this simple sample case, an office building is proposed to be constructed adjacent to a busy family restaurant. The high volume kitchen exhaust stack is located on the roof of the restaurant. HVAC planners proposed two alternative locations for the office building air intake, off a mechanical room on the second floor or on a mechanical penthouse on the building roof. A CFD air quality analysis is to help choose between the alternatives.

The first step in CFD model creation is the establishment of the modeling domain. Careful attention must be paid to the project site to ensure that all possible receptors are included in the domain. Air intakes and open windows are the most critical receptors for all types of exhaust. Occupants of offices or apartments are likely to be at the most risk to long term exposure. Less critical, but important to include nevertheless are pedestrian walkways.

When all of the potential receptors have been chosen, the next step is to include all of the buildings in the area that will have influence on the airflow at your subject sources. The behavior of air pressure and windflow is not always intuitive so all adjacent obstacles should be included. Adjacent buildings are very important to include, even if the building is upwind of the subject source only during infrequent winds. More distant buildings and obstacles should be included that are upwind during stronger wind
conditions. Generally it is not feasible to include every building, tree, or obstacle in the area in the model; one must use judgement in each case to ensure that the modeled airflow is as realistic as possible within the time and computer resources available.

After all of the receptors, sources, and buildings have been selected for inclusion in the model, the size of the domain can be established. The edges of the domain must be placed at a distance adequate enough to allow the full development of airflow features around the structures. A rule similar to the “5L” rule used as a guide in air quality modeling to account for the building wake zone has been found to be a useful in setting the domain size. Each edge of the domain must be at least \( L_b + 4w_m \) the distance from the nearest building edge where \( w_m \) is the maximum wind speed modeled and \( L_b \) is the smaller of building height or projected building width (in consistent units). The domain roof must be tall enough that no signature of turbulence from lower layers effects the roof layers of airflow. A general rule of thumb for the height of the domain is at least three times the height of the tallest building in the domain. However, this can vary depending on the case. For example, if a tall buoyant plume is being modeled in light winds, it will be necessary to have a height that can account for the high plume rise. If the plume is significantly buoyant, it may still reach the domain roof. This does not create problems at the surface as long as any reentrainment of the plume to sensitive receptors is fully accommodated.

Since the modeling domain size guideline depends on the maximum wind speed modeled, a full analysis of the site meteorology must be conducted before model construction. Each CFD model run will only use one wind condition, so a discrete amount of wind conditions must be selected for modeling based on the climatological record for the site. The modeler must be selective and develop a modeling scheme that includes the more important of the possible wind directions and wind speeds. First, the modeler must choose wind directions that correspond with the worst case re-circulation zones that may form around the buildings. These generally occur at the roof and in the lee wake of the building when the wind direction is perpendicular to a building face. For wind speeds, one generally wants to select the likely worst case wind to perform a conservative estimate. However, the client will be interested in the frequency and severity of impacts. Therefore, modeling must account for the distribution of wind speeds at each direction modeled. As a general guideline, it is advantageous to model three wind speeds per direction. This limits the number of modeling runs that must be conducted, while accounting for the distribution in wind speeds. Generally we have had success modeling the 98th percentile, 90th percentile, and median wind speed in frequency of occurrence per direction. These magnitudes generally will account for the worst case wind speed, the most common high wind, and the average wind.

Now, for the simple kitchen stack case we apply these guidelines to establish the model domain. We find that the sensitive receptors are our proposed air intake alternative positions and pedestrian sidewalks at the surface. The only buildings required in the modeling are the subject restaurant and the proposed office building. An illustration of the site and dimensions are included in Figure 13.
To determine the distance from the sides of the domain to the closest building surfaces we apply the general guideline of $L_b + 4w_m$. To determine this, we need to examine the building dimensions and site meteorology. For sake of an example, we will presume that the wind blows from one direction, where the restaurant is directly upwind of the office building. Examination of the meteorology of the region reveals that the 98th percentile wind is 8 m/s, 90th percentile wind is 5 m/s, and the mean wind is 3 m/s. Therefore, our $w_m$ is 8. The office building is 30 meters tall, 30 meters long, and 20 meters wide, so the office building $L_b$ is 20. With $L_b + 4w_m=52$, our corresponding domain sides will be located at 50 meters (rounded to simplify the model creation process) from the office building. The top of the domain will be 90 meters above the surface. The restaurant is 7 meters tall, 40 meters long, and 20 meters wide. Then $L_b + 4w_m=39$ gives us a guideline distance of 40 meters to place the domain walls from the faces of the restaurant.

After completion of the domain walls, the building geometry itself is placed in the domain. Small details of the building can be left out to simplify the modeling process. Details near sources or other regions important to specific airflows near receptors or sources should be included to make the modeling runs as realistic as possible. In this case, there are no roof obstacles near to the stack on the restaurant. On the roof of the adjacent office building, the roof access stairway door is upwind of the air intake, so we must include this detail in the modeling. Both the surfaces of the restaurant and office

Figure 13. Illustration of the sample project site domain. The structure on the left is the proposed office building and the building on the right is the restaurant. Site dimensions are labeled with the domain edges determined by the rule of thumb described above.
building are smooth brick with little detail. The street surface is fairly flat and distant from the source and receptors, so no details such as sidewalks and planter boxes are necessary. To simplify the example, the site is level so there is no topography to include. Topographical variation is critical to include in modeling cases where buildings are located on slopes, to account for the impact of the topography on vertical airflow.

The next step is to initialize the gas flow of the source kitchen stack and the office building air intake. For sake of demonstration, the atmosphere is assumed stable at an annual average temperature. The HVAC elements are shown in Figure 14.

After all features have been placed in the modeling, the domain must be cut into individual discrete volumes or “cells” where the actual computations for solving the Navier-Stokes equations take place. Enough cells must be initiated in the domain to accurately simulate the interaction of the gas flow with the domain features. However, one also wants to minimize the number of cells in the domain in order to shorten the length of time the computer has to run to solve the problem. This can be most easily accomplished by placing a high concentration of cells around the subject obstacles such as the stack, air intakes and building edges. The domain edges can have a lower concentration of cells because there is little change in the flow field gradient away from obstacles. As a general rule of thumb it is necessary to divide the smallest side of an object in the domain into at least 3 cells if it is to be resolved. In our example project, this would mean that each side of kitchen exhaust stack (the smallest obstacle in the domain) would have at least 3 cells per side if the stack were square. If more accuracy is needed, then more cells are required. However, one must take care that the gradient of cell size is even and smooth to ensure the solution is as accurate as possible. For a large domain, too many cells on the smallest object side may require too great a number of cells overall as you gradually step down from wider to finer mesh near small objects. Excessive computational time per case is undesirable. Figure 15 illustrates the mesh generated for our project domain. The lines in each coordinate plane are projected into the domain.
space to create individual cells. They can be easily seen on the surface of the obstacles but they also exist in the free space of the model. Each small cell represents a separate set of calculations during each time step.

![Illustration of "meshing," the partitioning of the domain into discrete volumes or "cells." Cell sizes must be smaller near areas of significant variation in the wind vector so all features of the flow are properly computed.](image)

When our domain has been properly "meshed" with good cell coverage, the domain walls must be initialized as proper boundaries. The domain floor will be initialized as a "wall" to simulate the impermeable ground surface. The domain roof, and sides can be initialized as “equal pressure” barriers, to ensure that air can flow evenly in or out of the domain depending on the wind field, or for the two walls at the appropriate sides, as “inlets” for air flowing into the domain at the defined wind speed and direction.

Each cell is initialized with a horizontal wind direction and speed, an initial atmospheric pressure and temperature and local turbulent intensity. The wind speed can be varied with height according to known conditions for an urban or rural surrounding and mean wind speed, as appropriate. The choice of the horizontal wind speed at an elevation is determined by the median, 90th and 98th percentile test wind velocities described above and the height of the meteorological tower providing the historic data. As mentioned above, neutral stability is assumed. No vertical velocity is provided in the initialization.
but is allowed to develop as the model evolves just as some of the cells close to walls will evolve lower horizontal velocities.

Now we have finished model construction and the problem can be solved. We must first initialize the amount of real-time that we wish the model to simulate. There are two factors that must be considered when considering the modeled length of time. First, enough time must be allotted for transport of all pollutants to the domain edge. This ensures that all the features of the plume behavior are evident in the final result. You can easily find this amount of time by considering the distance of the sources to their respective downwind domain boundaries and the windspeed being modeled. The other factor one must consider in determining the amount of time to run in the model is the amount of time that the model takes to “spin-up”, that is, to approach the nearest solution to steady state. This time must be determined from the experience of the modeler with a particular program and choice of closure model. This can be determined initially for a project with a run which is visualized at several intervals, looking for fully developed turbulence that finally does not change significantly with additional run time. In our sample project, the model time needed for the plume to reach the downwind wall should be about 50 seconds at most and the time needed to acquire full turbulence is about 8 seconds. Once the turbulence is fully developed, the solution of the full equation set can be halted and the plume allowed to disperse in a frozen wind field.

The k-ε turbulence model is selected and the solver is initiated. The solver uses its time-marching technique to solve the steady state solution from the initial conditions we prescribed.

After the solution has been reached, and it is seen to be physically reasonable, the results can be displayed graphically. In Figure 16, the visualization at the 98th percentile wind of 8 m/s includes mean wind vectors, exhaust concentrations, and the plume centerline.

We can see from these results that a 2nd floor mechanical room location of the air intake may pose problems during strong wind events in the vicinity. The plume is trapped in the re-circulation zone between the restaurant and the office building, and a significant amount of exhaust reaches the air intake. For a standard steak-house kitchen grill, we would probably consider a dilution threshold of about 2,000 as a target measurement. As illustrated, exhaust enters at the intake are at 100 dilutions, significantly above the threshold.

At slower, more common wind speeds (3 m/s), the plume does not entrain into a recirculation zone, but rises significantly as illustrated in Figure 17. Unfortunately, the plume reaches an air intake located on the roof. Exhaust enters the intake in this case at around 2,500 dilutions. This is marginally below our nuisance threshold value, but will not be below detection level for many other kitchen plumes.
The next step demonstrates the primary usefulness of CFD; the ability to modify existing conditions to examine alternatives. If the roof position of the air-handling unit is chosen, then odor of the plume may be evident in the building from time to time. Raising the kitchen stack may lower the frequency of odor if the plume does not reach the intake during lower wind speeds. Modeling conducted with the stack raised 2 meters to a 3-meter stack demonstrates that the plume will be more dispersed when it reaches the air intake, as illustrated in Figure 18. The plume reaches the air intake at around 8,000 dilutions in this case, which is significantly more dilutions than with the initial stack.
height. However, this doesn’t mean that the plume will no longer be a problem. Considering that we chose only several of the possible wind magnitudes that occur, impacts just as high as observed with the lower stack are likely during stronger winds, which will knock down the plume rise. However, since the stronger winds occur less frequently, raising the stack will lower the amount of time of higher impacts. The modeler must work closely with the site meteorology to develop an idea of the frequency of impacts to determine the best alternative.

Figure 18. Model run with the kitchen stack raised 2 meters to a 3-meter stack. During the lighter 3 m/s winds, the plume impacts the air intake at about 8,000 dilutions.

CFD: a Valid Approach to Studying the Surface Layer of the Atmosphere?

Though CFD shows much promise for the future of air quality analysis in urban environments, it should be used cautiously. Examining the steady state flow alone may not always be the best method when dealing with dispersion of exhaust plumes. It is very difficult to accurately model the surface layer of the atmosphere. It is a highly-variable chaotic entity which rarely resembles a steady state flow. Additionally, CFD turbulence modeling should be treated with care. Parameterizing turbulence may not result in realistic solutions when flows are highly turbulent.

Steady state flow

When dealing with air quality, some pollutants are a concern for exposures as brief as a few minutes (particularly odors) while for others long term exposure is the main concern. CFD steady state solutions can be a useful tool to analyze the long-term risks of outdoor pollutant impacts. However, the atmosphere itself is not steady, so analyzing air quality in the surface layer for shorter term exposures requires additional evaluation. In these
cases it is useful to analyze a spectrum of wind directions and velocities for each climatological mode of wind that records indicate are common, or for the suspect wind in an odor/pollutant complaint study. Standard deviations of wind angle can be estimated if one knows the stability properties of the atmosphere. For example, during neutral stability the standard deviation of an hourly wind direction is 7.5 to 12.5 degrees around the mean wind direction (EPA, 1987).

Wind speed variance cannot be estimated so easily, being greatly dependent on the nature of upwind obstacles. The surface layer itself is in a constant rolling turbulence, even without upwind obstacles, due to its viscous nature and the mixing of higher momentum wind down towards the surface. Therefore, for each wind direction, a range of wind speeds should be modeled. The modeler should note that each climatological or suspect wind case can only be investigated by examining the output of several CFD runs which encompass the range of variability that wind case may endure. Even so, this technique does not fully result in modeling of true atmospheric phenomena, but in a screening type analysis it can be used to estimate impacts.

Domain turbulence initialization

Even with a rigorous method to account for wind variability, the nature of the incoming flow may not be represented adequately. Upstream objects such as buildings, hills, or vegetation may create large eddies and other flow variations which will have an impact on the incoming flow of our domain. Therefore, a CFD analysis would not be recommended for a building located among many other buildings of similar or larger size without special care. To account for incoming turbulence, every building which has a moderate to large impact on the incoming flow would have to be included. In a dense urban environment such as a downtown district with many skyscrapers, a CFD analysis might require modeling the entire city skyline to accurately predict flow at one building! Therefore, each CFD case has to be examined carefully to determine what factors need to be included to initialize realistic flow.

Model properties

Another factor in CFD is the mathematics of the model itself. As mentioned before, there are many different approaches to handling the turbulence terms in CFD modeling. The k-ε model, or first order closure model, is an efficient and often used model that returns adequate results in modeling air flow at the surface layer of the atmosphere. Several other more accurate methods are becoming more popular as desktop computers become more powerful. Reynolds Stess Transport Models (RSTMs), or second order closure models, and Large Eddy Simulation (LES) are two methods that are being used more frequently. However, these models can be computationally demanding. A model run which may only take a few hours on a fast desktop computer using the k-ε model may take a one or two orders of magnitude longer using many implementations of the LES model. In validation tests, LES and RSTMs generally have performed better than the k-ε model, that is, generating results that more closely match wind tunnel streamlines. Nevertheless, the k-ε
model results are sufficiently accurate in tests involving airflow in the scales of concern with air quality modeling.

One popular method in validation of CFD models is the “block” test, where CFD steady state flow results are compared to the mean flow streamlines around blocks in wind tunnels. Air quality analyses often are concerned with the lengths of re-circulation zones, so these are often compared in validation tests, as seen in Figure 19.

Most validation tests have found that the $k-\varepsilon$ model often over-predicts the length of the lee re-circulation zone and under-predicts the length of the windward re-circulation zone. The roof re-circulation zone tends to be comparable, but may not be seen if the project is not initialized correctly or the mesh is not small enough to properly characterize the leading edge (a common failing). Even with these problems CFD for stack placement purposes is useful and even conservative in respect to re-entrainment. Since the re-circulation zones often grow with increasing wind speed, an over-predicted re-circulation zone analysis would then represent the zone of the high range of wind speed variability.

![Figure 20: Airflow around a block for a validation test. Re-circulation zone lengths are compared to wind-tunnel and water-channel tests.](image)

Another factor involving model accuracy in CFD is plume modeling. So much attention is given to dispersion and plume centerline streamlines in air quality analysis, it is important to make sure that the model is accurately modeling plume rise and dispersion. Some qualitative comparisons of CFD plume dispersion to the U.S. Environmental Protection Agency’s basic dispersion model, SCREEN3, were carried out. These comparisons found that the CFD plume rise and dispersion is quite comparable to the SCREEN3 output for lower wind speeds. At higher wind speeds, the CFD model predicted a buoyant plume rise greater than the initial plume rise modeled by SCREEN3.
Conclusion

Air quality analysts can successfully use CFD using the k-ε turbulence closure approach included in the commercial model CFD2000 for evaluation of plume movement in urban microenvironments. CFD offers a cost-effective new tool for analyzing pollutant dispersion around buildings and sets of buildings. It is a useful tool to help guide the placement of exhaust stacks and air handling units on buildings and in investigating sources of odor which impact building air intakes. Better initialization schemes for the atmospheric conditions and improved turbulence closure models will improve CFD results.
References


Wilson, D.J., 1979: Flow patterns over flat-roofed buildings and application to exhaust stack design, ASHRAE Transactions, 85, 284-295.

United States Environmental Protection Agency, June 1987: On-site meteorological program guidance for regulatory modeling applications, EPA 450/4-87-013