THE THREAT OF CHEMICAL AND BIOLOGICAL TERRORISM: PREPARING A RESPONSE

When a deadly contaminant is released in a city, the window of time for meaningful response is brief. High-performance computing can play a major role in preparing an effective response. This article describes one such effort, which exploits detailed 3D computational fluid dynamics simulations of the airflow in buildings and cities.

The September 11th terrorist attacks on the World Trade Center and the Pentagon resulted in thousands of deaths and shattered many Americans' illusions about the country's invulnerability to attack. Military experts, political analysts, and the President all agree that chemical and biological (CB) weapons have now joined nuclear weapons as the major threat against the United States. Media headlines reflect a growing fear that people and cities in the US might now be the main targets of CB terrorism.

One way the US Department of Defense is addressing the need for more effective counterterrorism is by building improved modeling and simulation capabilities. In the event that a CB agent is unleashed on a US city, authorities must be able to quickly predict that agent's spread and deposition and subsequently direct efforts to manage the consequences. One fundamental element in preparing for such a response is to use detailed high-fidelity modeling of contaminant transport and dispersion prior to such an attack. Accurately predicting wind flow patterns and a CB agent's contaminated "footprints" on a city—and then using this information to construct an effective civil defense plan—could save countless lives.

This article describes the physics and fluid dynamics needed to solve urban CB transport problems accurately and how high-performance computing is helping combat the threat of intentional and unintentional contaminant releases. Results computed with the US Naval Research Laboratory's FAST3D-CT model show that solving these CB problems in an urban environment is practical with today's high-performance computing (HPC) resources. Researchers have run the FAST3D-CT model extensively for the last five years to test and validate it, extend its capabilities, and better understand the characteristics and physical limitations of CB transport in cities and near buildings. This article also describes an initiative called *dispersion nomographs*, which uses the simulation of multiple independent contaminant releases in a multikilometer section of a city to develop a much faster tool for operational prediction. (In response to the events of September 11th, this article presents results generally available previously and draws no conclusions on operational matters.)

1521-9615/02/$17.00 © 2002 IEEE

JAY BORIS
US Naval Research Laboratory

Using CFD models for CB transport and dispersion

Current operational hazard prediction tools for outdoor CB scenarios are based on relatively fast-running "common use" models that use limited topography and weather or wind data. These models generally use Lagrangian similarity solution approximations to the fluid equations and give simplified solutions that ignore the effects of flow-encountering 3D structures. The air flowing over and around buildings in urban settings is fully separated and is characterized by vortex shedding and turbulent fluctuations throughout the fluid volume. In this regime, the usual timesaving fluid-flow approximations such as steady-state flow, potential flow, similarity solutions, and diffusive turbulence models don't apply. Therefore, a clear need exists for high-resolution numerical models that can accurately compute the flow and deposition of contaminant gases within and around real buildings under a variety of dynamically changing wind and weather conditions. Such detailed fluid models require HPC resources for their effective execution.

Because fluid dynamics is the most important physical process involved in CB transport and dispersion, we should invest the greatest care and effort in modeling the fluid flow. Computational fluid dynamics (CFD) is often defined as the accurate numerical solution of the equations describing fluid and gas motion. Thus we should use high-resolution, three-dimensional CFD—the most accurate computational approach we can consider—to scientifically underpin other simpler modeling approaches. CFD models can accurately simulate dynamic flow to high spatial resolution in and around complex geometry. In a city, this resolution can be a few meters if we use modern computers.

CFD is truly predictive and generally uses convergent algorithms whose solutions get better (meaning they converge) as we bring more resolution and computer power to bear. Many approximations made in the interest of reducing computer requirements in the past do not have this property. Because fluid dynamics equations are nonlinear and time-dependent, computing high-resolution numerical solutions of the mathematical model is the only way we can know that the results representing important physical phenomena resemble reality. When we apply CFD, numerical errors in solving for the most important physical processes should not limit the solutions' accuracy and reliability. Because of the resources and expertise required, applying CFD models to CB agent transport in cities is relatively new.

The CFD representation's advantages include the abilities to quantify complex geometry effects, predict dynamic nonlinear processes faithfully, and handle problems reliably in regimes where experiments (and thus model validations) are impossible or impractical. We can therefore use CFD solutions to understand the complex interactions of phenomena in experiments and field trials and even help design these experiments. We can also use the simulations in an engineering model to help design new platforms, systems, and devices. Developing new sensors and systematically assembling them into robust, effective detection systems is one particularly important use of CFD that does not suffer from the relatively long time needed to perform the computations to high accuracy.

However, one question still persists: How can we use CFD in an operational environment that requires immediate answers? At recent conferences, I described a new approach called dispersion nomographs to do just this.[1] This new approach separates the HPC computation completely from the zero-latency (no delay) recall of the previously computed 3D results. Thus the best of both the CFD and the common use approaches is possible.

The high-fidelity FAST3D-CT model

FAST3D started evolving into its present form in 1992 as a model to compute airflow over the superstructure of ships underway at sea. It became fully scalable as an initiative of the DoD High Performance Computing Modernization Program (HPCMP) from 1995 to 1998, a necessary precursor to effectively using the model for realistically complex problems. The first engineering-scale applications of the new general-geometry model (see Figure 1) contributed to the design and analysis of ship superstructures for turbulent vortices interfering with landing operations.[2–4] Concerned with personnel safety and the safety of delicate electronic equipment above deck, subsequent shipboard applications included transporting and dispersing the hot contaminant stack gases[5] from a FAST3D simulation. Recognizing the importance of these applications, the Navy supported a series of wind tunnel studies for validating models. FAST3D was one of two models that passed these tests.

The extension to CB contaminant transport over and around buildings was just a small step. The characteristic sizes, flow speeds, and physics, along with the ship's topside airflow

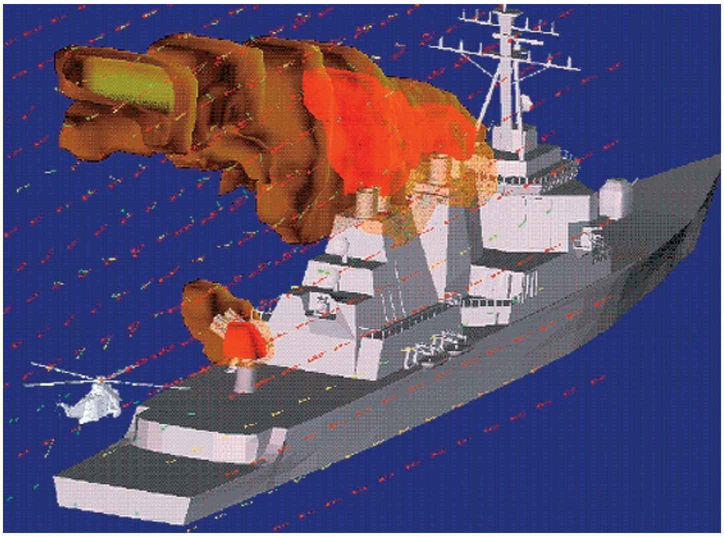


Figure 1. DDG-51 flight deck analysis and redesign computations.

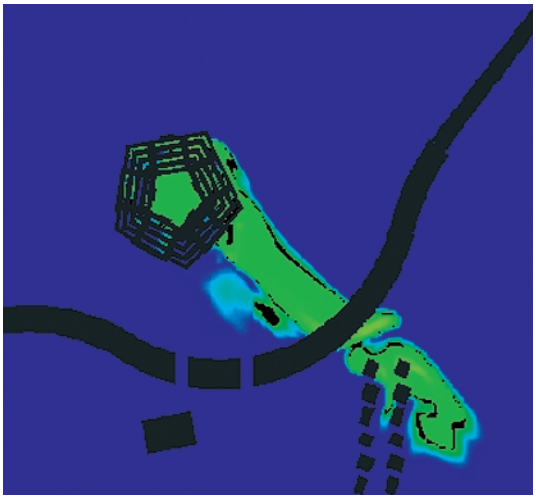


Figure 2. An idealized contaminant cloud passing over the Pentagon as computed by FAST3D.

geometric complexity, are quite comparable to the fully separated flow encountered in urban contaminant transport. As part of the DoD's HPCMP contribution to the Supercomputing96 Conference, we used FAST3D in a live demonstration to compute a contaminant's flow over the Pentagon. Figure 2 shows a computation performed faster than real time and whose graphical results were communicated over the Internet to the conference site. The figure shows the contaminant cloud on a horizontal slice 3 meters above the ground in the center of the Pentagon. The slice cuts through the elevated roadway of US Route 395, whose tunnels just south of the Pentagon are shown. Thirty minutes of real time required 26 minutes to compute on a 128-node Intel iPSC-860 parallel computer.

More recent applications requiring some of the largest parallel processors in the DoD include a computation of the airflow over a 4 km × 2 km section of a city resembling Washington, DC (see Figure 3). The blue and green regions are low flow speeds below one meter per second, mostly confined to recirculation regions in the wake of buildings or groups of buildings. The yellow–orange–red regions in the figure are air speeds in excess of one meter per second. The combined effect of all these dynamic recirculation zones makes urban airflow qualitatively different from the generally wind-aligned flows modeled by existing plume models.

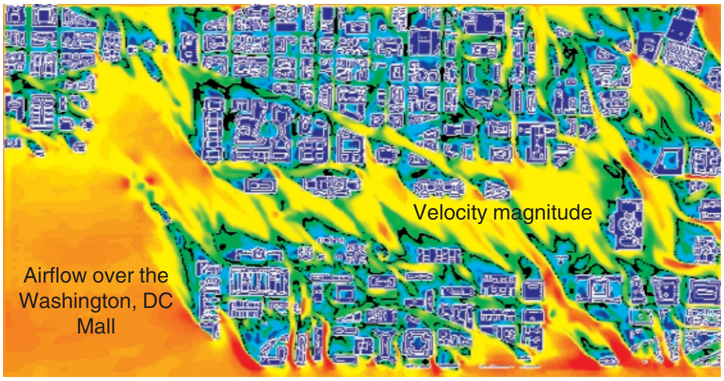


Figure 3. Color contour plot of the velocity magnitude for air flowing across the Washington, DC mall.

In its current form, the FAST3D-CT numerical model solves the high Reynolds number Navier-Stokes equations in a time-dependent, 3D, large eddy simulation formulation.[6,7] The underlying mathematical model in FAST3D-CT is a set of conservation equations for mass, momentum, potential temperature, and contaminant species. The time integration is second-order accurate and has been adapted for fast execution with very complex geometry. The CFD algorithms solve for slow but compressible flow with arbitrary buoyancy using scalable parallel processing implementations. Most of the algorithms, their coupling procedures, and their implementation in FAST3D-CT are described elsewhere.[7]

The model's 3D flow solver is based on the scalable, low dissipation *flux-corrected transport* algorithm.[8,9] FCT is a high-order, monotone, positivity-preserving method for solving generalized continuity equations with source terms. We achieve the required monotonicity by introducing a diffusive flux and later correcting the

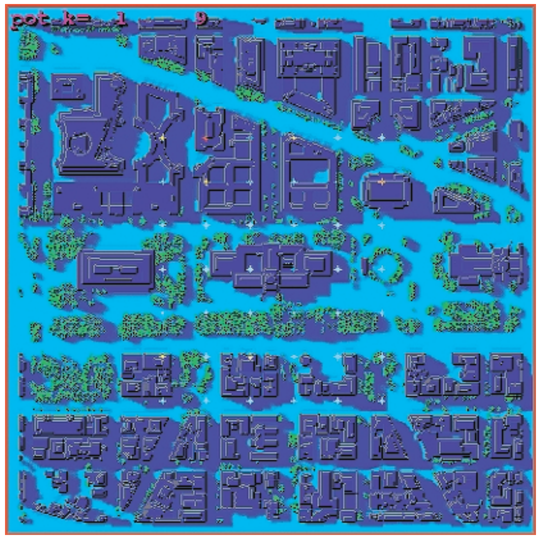


Figure 4. A geometry database for an urban area.

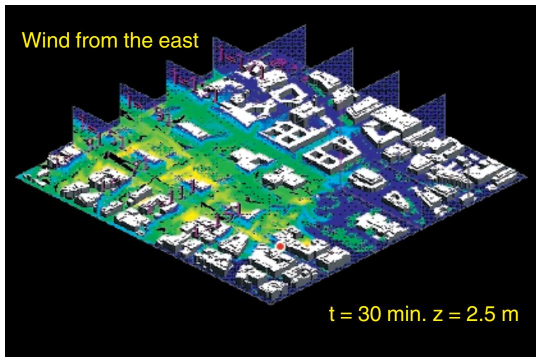


Figure 5. Three-dimensional visualization of a contaminant cloud computed using FAST3D-CT.

calculated results with an antidiffusive flux modified by a flux limiter. This convection algorithm converges to the solution of the underlying equations.[10] (The specific version of the convection algorithm implemented in FAST3D-CT is documented elsewhere.[11]) A Monotone Integrated Large Eddy Subgrid (MILES) turbulence model is automatically built in when you compute turbulent flows with an FCT algorithm.[12,13] This fact is the subject of the extensive testing Christer Fureby and Fernando Grinstein have recently performed.[14,15] They show that the MILES subgrid model in FAST3D-CT is correctly tensorial in nature and is accurate for both free-shear and wall-bounded flows. These are the conditions of most importance for CB transport in cities.

Virtual cell embedding algorithms were developed specifically for HPC parallel processing and they can treat the complex geometry of buildings and cities.[2–4] VCE is a method for representing and computing the flow around bodies of arbitrary shape on a Cartesian grid without sacrificing computational speed or memory. Although the grid remains orthogonal, the VCE method effectively increases the number of mesh points in the vicinity of complex geometric shapes, thus eliminating "staircase" effects. In addition to its intrinsic scalability, VCE yields a simple and efficient grid generation scheme.

For urban-scale simulations, we need a geometric database for the city. Ideally, we can specify the buildings, trees, and terrain elevation so that we can accommodate additional information about the surfaces (such as grass, dirt, streets, and so forth). Figure 3 uses the outlines of the buildings to visualize the flow, and Figure 4 shows an enlarged cross-section of the computational domain created from such a database for a 1.3 km × 1.3 km portion of a modeled city. The geometric database is arranged in a tile format for very fast CFD grid generation and to simplify the addition or deletion of features (buildings, structures, terrain, and so on). This tile format also lets us run the grid generator with the flow solver for better computational efficiency and faster simulation turnaround.

Figure 4 actually plots the air temperature just above the ground 5 seconds after the beginning of a run. Dark blue represents the ambient temperature, 293.15 degrees Kelvin. Light blue indicates temperatures elevated a few tenths of a degree above ambient conditions. The horizontal cross-section cuts through all the buildings as in Figure 3; trees are included in the figure along with their appropriate effects on the airflow in the simulation itself. Note that the sun casts shadows as dictated by the geometry of the buildings and trees. This causes differential heating and has important effects on the flow and transport of contaminants in a large enough computational domain.

Figure 5 shows a 3D visualization of a contaminant cloud propagating through Figure 4's grid when the wind is from the east. The red-filled circle in Figure 5 identifies the source location for this figure. The display and diagnosis of simulation results is an important adjunct to rapidly comprehending the results. Using the precomputed 2D cross-sections from FAST3D-CT and additional geometric data taken from the tile database, a simple rendering program can postprocess the sequence of stored cross-sections into 3D isometric plots. A simplified graphical technique minimizes computational processing time to a couple of seconds, making it possible to determine exactly where a particular level of contaminant is using the cross-sectional plane frames as location references.

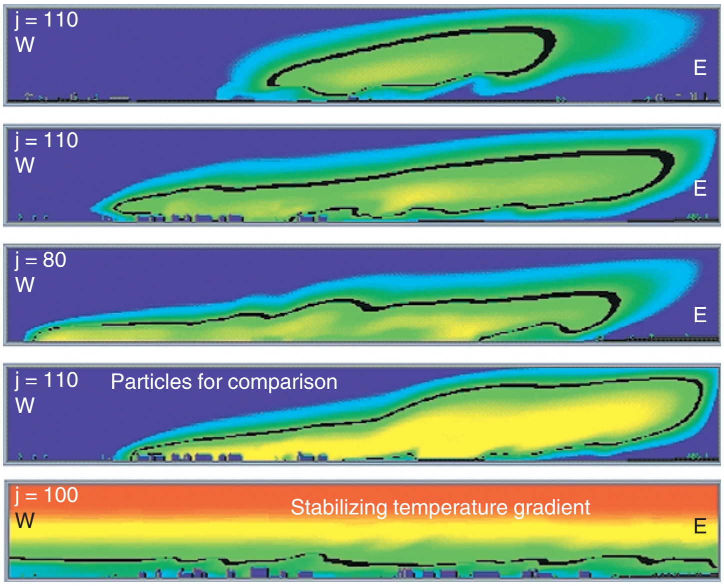


Figure 6. Vertical cross-sections of a multigroup evaporating droplet cloud and a multigroup particle cloud through an urban landscape.

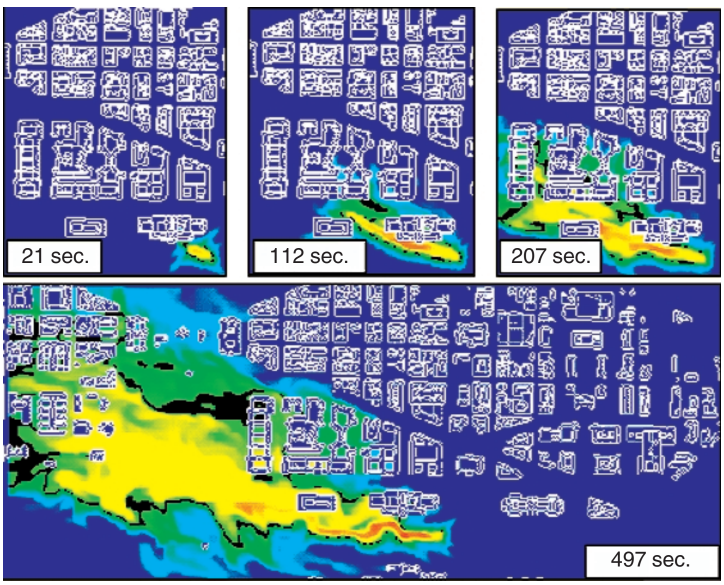


Figure 7. Evolution of a ground-level vapor source with time-varying winds from COAMPS.

The FAST3D-CT model is an adaptation of FAST3D for containment transport. FAST3D, designed as a reactive flow code, already had many of the validated algorithms and numerical techniques needed for atmospheric CB transport. We adapted FAST3D for CB agent transport by including the necessary multiphase flow physics for several coexisting size distributions of droplets or particles. Each size distribution has its own velocity field and continuity equation, including deposition on surfaces, evaporation physics, and turbulent transport to surfaces.[16] Figure 6 shows vertical cross-sections from a simulation using these capabilities. The top three panels show the summed mass density of the evaporated vapor and droplets of five different sizes on three east–west cuts through the computation. The next panel shows the corresponding cross-section through the summed mass density of much smaller particles for another run. The bottom panel shows the east–west and vertical variation of the potential temperature through the center of the computational domain. The higher temperature above the ground, shown as orange and red in Figure 6, stabilizes the vertical flow in the atmosphere, but the buildings' mixing and destabilizing effects are evident. The interplay between these processes and the computational domain's solar heating determines the correct urban boundary layer and fluctuation spectra that we should input as the simulation's upwind and top boundary conditions.

We are conducting extensive work on the boundary condition algorithms to allow shifting and fluctuating winds to smoothly change an inflow boundary to an outflow boundary and back as a result of the computed and imposed meteorological flows. One aspect of the work on FAST3D-CT is the connection of FAST3D-CT simulations to the predictions of COAMPS (Coupled Ocean-Atmospheric Mesoscale Prediction System), the Navy's operational mesoscale weather prediction model.[17] We used COAMPS's time-dependent atmospheric profiles as evolving boundary conditions for the simulation in Figure 7. Here a contaminant drifts north and west across the central portion of the contaminated domain as the COAMPS-predicted winds fluctuate due to vortex shedding from the buildings while changing direction and vertical profiles continuously.

The NRL team added and tested additional models in FAST3D-CT for stochastic backscatter in the turbulence representation and for turbulent transport of heat and momentum between air and body surfaces. We compared these capabilities with experiments on ventilation-driven flows inside a large room with heated walls.[18] We also applied FAST3D-CT to coupled interior/exterior modeling where ventilation flows inside a building, leaks through the building shell and between the rooms, and external wind pressure on the building combine to define a contaminant's movement inside and outside the building. Bohdan Cybyk, Theodore Young, and I conducted these simulations as part of a validation study and model comparison with experiments conducted at the Dugway Proving Ground.[16,19,20]

Calibration and validation studies

As with many engineering-grade models, HPC applications of the FAST3D-CT contaminant transport model generally fall into three categories: directed applications, sensitivity studies, and calibration–validation studies. Directed applications are solutions for specific important scenarios in which the answers are of immediate programmatic concern and where there is little directly relevant data or where modifying a system or procedure is contemplated. Directed applications can also demonstrate significant new capabilities, design new systems of components, or establish operational rules of thumb for counterterrorism procedures and preparations.

Sensitivity studies are generally conducted with the model to assess the relative importance of various processes or to study the effects of changing a parameter or a set of conditions. These "what if" computations often entail parameter variations about the best guess input data for a particular class of scenarios. This helps determine the importance of particular physical assumptions, modeling approximations, or input uncertainties. Sensitivity studies are often performed in a regime where little or no corroborating data are available but where trends and relative scales of importance can be as valuable as hard numbers. From a practical viewpoint, we can also use these detailed simulations to establish a database for developing and calibrating simpler models.

We primarily use calibration–validation studies for model development and accuracy assessment. These studies rely on the existence of accurate and reliable experimental data. The key distinction between calibration and validation is tied to the ability to measure and control inflow and boundary flow characteristics and to repeat the experiments. Validation is a stronger statement than calibration because it entails a thorough characterization of the inflow and initial conditions for the experiments against which the computations are compared. When it is not possible to say in detail exactly what problem a computational model should solve, it is equally difficult to assess how good the model actually is.

The NRL team has regularly undertaken FAST3D calibration and validation studies over the more than two decades of the model's development and use. Figure 8 shows a horizontal cross-section through an HPC computation of the airflow over the Washington, DC mall. The wind was steady at 3 meters per second from the south. The figure shows the cross-section 20 meters above the ground level following the terrain. The variable plotted is the north–south (Y) component of velocity with orange faster than yellow and yellow faster than green. Blue corresponds to regions of negative Y-velocity indicating building recirculation zones. The Washington monument's wake is clearly visible as a sinusoidal region of momentum deficit (green in the figure). The overlaid panel on the left side of Figure 8 shows an enlargement of the region around and just north of the Washington monument from which we determined the vortex-shedding wavelength. Because the monument is square, tall, and quite isolated, this simulation provides a convenient and rigorous test of the model's fidelity with respect to vortex shedding. Fluid dynamics researchers have collected much experimental data concerning the shedding of vortices from square cylinders.

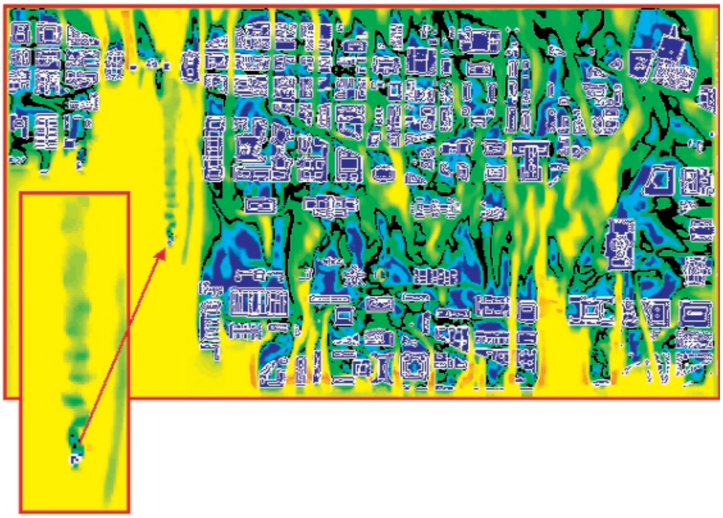
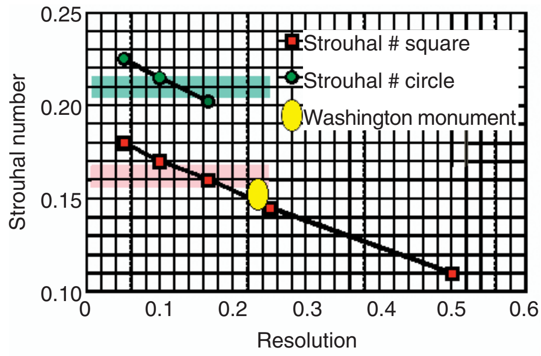


Figure 8. Vortex shedding from the Washington monument.

Figure 9 summarizes several computations of vortex shedding by FAST3D-CT and compares them with the experimentally measured values for both square and circular cylinders. The figure plots the nondimensional Strouhal number for the shedding versus the structure's computational resolution. The Strouhal number is the ratio of the characteristic wavelength in the wake

Figure 9. Variation of nondimension al vortex shedding Strouhal number computed with FAST3D.



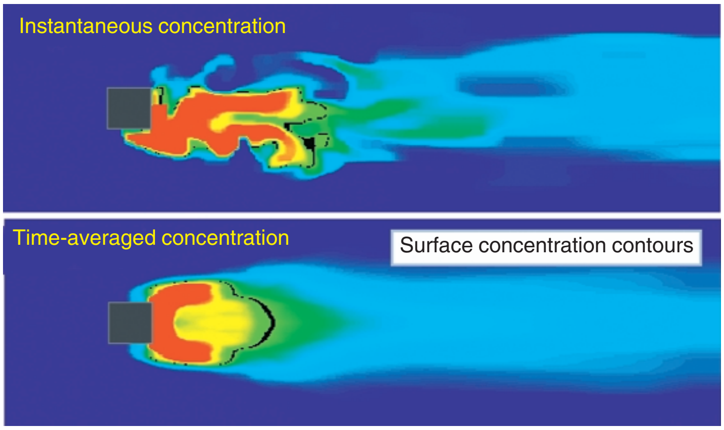


Figure 10. Dye tracer circulation and dispersion in the wake of a surface-mounted cube.

to the size of the 2D obstacle (thickness or diameter). A Strouhal number of 0.2 means the wavelength is five times the object's size; a resolution of 0.2 means that there are five computational cells resolving the object. We ran the model for ideal squares and circles at a wide range of resolutions; the horizontal bands in the figure show the range of experimental values measured for squares (red) and circles (green). The three-dimensional Washington monument data point is indicated as a yellow ellipse in Figure 9 and is only in error by about 5 percent with five-meter spatial resolution. The ellipse axes indicate uncertainties in measuring the wavelength and resolution from Figure 8. Figure 9 shows that the model will still shed vortices from the Washington monument even at 10-meter resolution, but the frequency is 30 to 40 percent too low.

Fluid dynamics becomes arbitrarily complex in even simple flow geometries when turbulence is present. Furthermore, the representation of the turbulence can play an important role in determining how rapidly a CB contaminant plume spreads. Sally Cheatham and I conducted a FAST3D-CT validation study of surface-mounted cubical obstacles by using HPC resources. This helped us investigate the effects of base-plate boundary conditions and the inflow velocity profile on a dye tracer's flow and dispersion patterns.[21,22] Because the cube is a single parameter representation (or size) that approximates many buildings, it is an excellent baseline problem for a building-block approach to urban modeling development. It is also a configuration well suited to validation in wind tunnels, where we can control flow parameters.[23–30]

Figure 10 shows the dye tracer circulation and shedding in the wake behind the cube. The cross-sections show the dye density in the layer of computational cells adjacent to the surface in the boundary layer where the fluid velocity is small. The upper panel shows an instantaneous snapshot of the tracer distribution while the lower panel shows the distribution's time average.[22] The horseshoe-shaped vortex is clearly visible in the time-averaged result, although the dye tracer, which is injected one cube width downwind on the centerline, never migrates to the upwind side of the cube on the left. Comparing these panels and the detailed analyses of several HPC runs makes it clear that the tracer's lateral dispersion is dominated by the intermittent, nonperiodic shedding of vortices and vortex pairs. The actual wake is about twice as wide as can be accounted for by diffusion models superimposed on steady-state solutions. Therefore, modeling the dynamic atmospheric wind fluctuations in the urban boundary layer to accurately represent the inflow conditions is important to urban CB transport and dispersion.[30]

Although research on simple structures such as the cube can illuminate this issue, the extensibility of the simulation results to realistic situations depends on the model's ability to represent more complex (and realistic) configurations easily and faithfully. In a calibration–validation study conducted around Building 170 at Lawrence Livermore National Laboratory,[31,32] the effects of inflow wind profiles and flow patterns around a complex multistory building were investigated using FAST3D-CT. The geometrical representation of the building and its surroundings is based on architectural plans, GPS coordinates, and an aerial photo, and it includes a large row of trees and a small rectangular building located just east of the main building (see Figure 11). We ran a time-dependent simulation with a constant 3 m/s wind flowing from the southwest. We modeled the north–south row of Eucalyptus trees using a geometrically complex, fractal-like momentum-deficit representation based on a previously published model.[33] We measured time-averaged and fluctuating winds experimentally and used them as input to the CFD model.

Figure 11 shows several cross-section snapshots of the simulated velocity. The two vertical cross-sections at the bottom of the figure appear in the top views as dotted lines. The dotted outlines in the figure indicate the relative location of bodies not actually in the plane of the cross-section. The velocity deficit in and behind the trees is clearly visible. The comparison of computed and experimental results generally shows that averaging unsteady simulations in time matches the time-averaged experimental results with the largest relative errors occurring where the averages are near zero at dead zones. The Livermore experiments also provided several relatively high-frequency wind fluctuation data sets that proved instrumental in the initial stages of constructing FAST3D-CT's parameterized fluctuating wind boundary condition model.

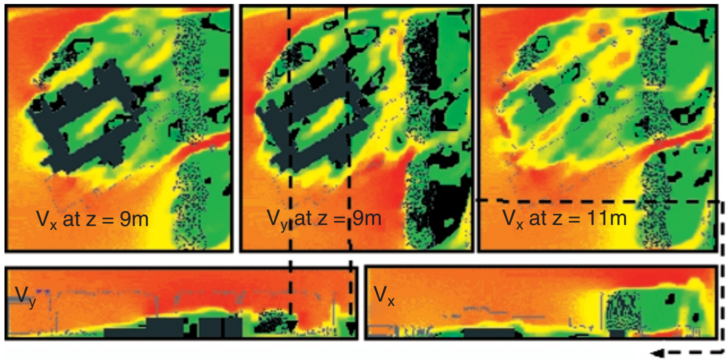


Figure 11. Calibration and validation studies using wind measurements at Livermore Building 170. The wind is flowing from 225 degrees at 3 m/s.

HPC urban demonstration computations and sensitivity studies

Large-scale FAST3D-CT simulations of city areas covering several square kilometers during the last four years demonstrate the feasibility of using a scalable CFD model to simulate realistic CB release scenarios in complex environments. In fact, for relatively small areas of a city (such as one or two square kilometers), we can now do contaminant transport computations on desktop workstations and laptops. These simulations incorporate the full geometry of the buildings, trees, and terrain and use fluctuating wind fields guided by outputs from mesoscale weather models or in situ measurements.

City-scale field studies conducted in the past used data that are nominally available for calibrating and validating a range of modeling tools. We are using several of these data sets for our ongoing FAST3D-CT validation efforts. However, the data's variable quality and incompleteness in comparison to a wind tunnel often raise as many legitimate questions as they answer. There are obvious political difficulties associated with releasing even harmless but detectable tracers that can adequately characterize flow conditions in cities. Furthermore, choosing what simulated quantities to compare, which data sets to focus on, and which numerical parameters to adjust for best comparison leaves any statement that we can validate a model this way subject to question. The fact that agencies in the US, Europe, and Asia yearly submit major budget requests to conduct further urban-scale experiments for model validation suggests a general concurrence that validation efforts to date are inadequate.

We can classify most of the FAST3D-CT applications used in the last three years as sensitivity studies, problems for which there is no corresponding experimental data set but where comparisons of several simulations differing in only one aspect yield useful information. About 300 such runs have been performed at NRL around a few baseline cases. Subjects studied include

- urban boundary layer structure and evolution as air flows over a city
- effect of buildings on airflow unsteadiness and contaminant dispersion
- effect of wind fluctuations on contaminant dispersion
- action of trees on contaminant transport and dispersion
- differences between vapor, droplet, and particulate transport and dispersion
- local effects of solar heating on urban canyon circulation and atmospheric turbulence
- effects of turbulent stochastic backscatter on lofting contaminants near the ground
- variations on contaminant decay caused by wind direction, strength, and geometry
- street-level flows induced by traffic
- numerical convergence with increasing spatial resolution and various boundary representations

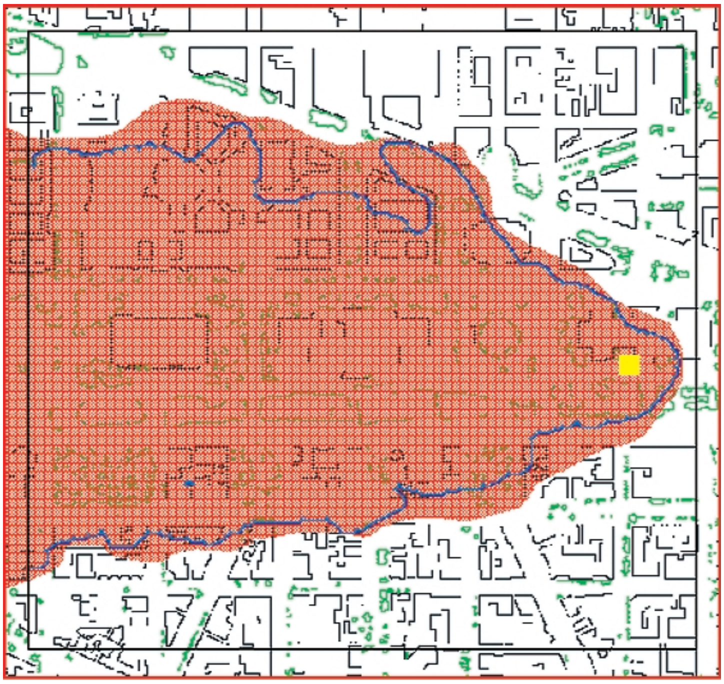


Figure 12. Example of a dispersion nomograph for wind from the east.

As confidence in accurate HPC models such as FAST3D-CT grows, these sensitivity studies should be conducted instead of costly experimental programs, particularly in situations where the experiments are limited by physics, cost, or law.

Dispersion nomographs

Decision makers responsible for executing operational and crisis management procedures during a CB attack cannot use high-fidelity CFD directly in its present form because the computer requirements are still too great. But there is another way to use these simulations of airborne CB contaminants to develop and calibrate much faster models in operational conditions when fast, informed responses are needed. The FAST3D-CT model has enabled a new, high-fidelity zero-latency approach called dispersion nomographs, which present a practical solution to this problem. Dispersion nomographs also provide a fast approach to automated sensor integration for protecting buildings or facilities in high-density urban environments. This new nomograph technology relies on a special kind of map overlay that is precomputed for exterior threat scenarios and includes the complexity or real geometry, realistic urban boundary layer profiles, and time-varying winds. In its current state, dispersion nomograph technology is specifically aimed at exterior contaminant propagation in any kind of terrain and complex geometry. Using this new data compression and recall technology, a matrix of detailed 3D simulations can provide the basis for informed operational decisions in real-time crisis management, even when the precise parameters of a release are unknown.

Two key benefits of the dispersion nomograph approach are its speed and accuracy. The limited dispersion nomograph database, processed and compressed from a number of HPC simulations, gives a fast (on the order of milliseconds) way to predict the consequences of a set of sensor readings from a suspected release location. This zero-latency information can orient line-of-sight sensors, activate building air supply defenses, plan escape routes, and backtrack to possible source locations. Figure 12 shows an example of a dispersion nomograph for a release at the location indicated as a yellow square in the middle right of the figure. The shaded area is the estimated contaminated footprint superimposed on a map of the city. Although the wind is from the east, the city geometry and street layout has clearly induced some asymmetries about the wind direction. If the indicated location is a single sensor reading, the shaded area is interpreted as the region the contaminant could reach. The fusion of additional sensor readings as they are received could then further define the contaminated region.

Current efforts center on how to compute nomograph databases automatically from matrices of FAST3D-CT simulations and how to quantify and bound the errors and uncertainties inherent in the nomograph approximation. There are really two types of errors to consider: those inherent in the 3D database used to derive the nomograph tables and those arising from the data compression and data recall algorithms used by the nomographs. The blue line superimposed on Figure 12 is the contamination boundary of a CB cloud computed by FAST3D-CT for the indicated source location. The agreement is good, and the nomograph gives a conservative estimate of the contaminated area.

Automated application areas for dispersion nomographs are also being investigated at NRL. One such area involves CB sensor network development. Knowing in advance, based on geometry, wind direction, and tree foliage, just where a cloud would have to appear to threaten a particular installation or event will greatly enhance the potential effectiveness of sensor systems. Knowing the expected evolution of a contaminant cloud faster than real time, based on only a few sensor readings, can save lives.

Thanks to the resources and foresight of the DoD HPCMP, we now have a good head start on harvesting the power of modern, large-scale parallel computers to advance counterterrorism planning and operational capabilities. The nation's increased focus on developing effective procedures to respond to CB attacks deserves a wide range of prediction techniques and models. High-fidelity, time-dependent CFD models such as the FAST3D-CT simulation model can predict a CB agent's movement in a complex urban environment and thus provide the strongest technical and scientific foundation for the nation's more broadly based simulation and modeling effort.

Detailed CFD modeling is being extended to CB transport and dispersion threat scenarios ranging from flow through a building's rooms, corridors, and auditoriums to the tunnels, stations, and entrances of subway systems, and across the buildings, trees, and complex open terrain of urban scenarios. As we wait for the computer advances required to incorporate real-time CFD directly into operational consequence management, we are hurrying to mature the dispersion nomograph technology into a suitable tool for a wide range of operational CB applications.

Acknowledgments

Several sponsors have supported the development of the FAST3D capability from its infancy to its present form. These include the Office of Naval Research (ONR), the US Naval Research Laboratory (NRL), the Defense Threat Reduction Agency (DTRA, then DNA and DASA), the Defense Advanced Research Projects Agency (DARPA), the Department of Energy (DOE), and the 911-BIO ACTD program. The project reported here got its inception and initial push by the DoD High Performance Computing Modernization Program (HPCMP) in 1996 and subsequently received partial support as a demonstration problem in the FAST3D CHSSI program that ran from 1996 to 1999.

Many people have contributed their ingenuity and hard work to bring the FAST3D capability to its current level. Alexandra Landsberg and Theodore Young, Jr., were instrumental in developing the VCE complex geometry capability and the scalable version of FAST3D implemented for the HPCM program. Bohdan Cybyk, Charles Lind, Mark Emery, Carey Cox, and Almadena Chechlkanova all contributed to the FAST3D CHSSI project and several other projects to develop the contaminant transport (CT) version of the model while at NRL. I also thank my colleagues at NRL, Sally Cheatham, Fernando Grinstein, Kevin Harris, John Iselin, K. Kailasanath, Keith Obenschain, Gopal Patnaik, Robert Scott, and Ted Young, for their daily contributions to the FAST3D-CT projects and for figures and analysis from their published and ongoing work for this article. Finally, I acknowledge the efforts of Jamy Bond and Bill Gabor of the HPCMP Office for their help and encouragement in pulling this article together.

References

1. J. Boris et al., "Simulation of Fluid Dynamics around Complex Urban Geometries," *Proc. 39th Aerospace Sciences Meeting*, American Inst. Aeronautics and Astronautics, Reston, Va., 2001.
2. A.M. Landsberg et al., "Naval Ship Superstructure Design: Complex Three-Dimensional Flows Using an Efficient, Parallel Method," *Proc. Soc. for Computer Simulations Multiconference*, SCS, San Diego, 1993, pp. 15–20.
3. A.M. Landsberg et al., "Analysis of the Nonlinear Coupling Effects of a Helicopter Downwash with an Unsteady Ship Airwake," *Proc. 33rd Aerospace Sciences Meeting*, American Inst. Aeronautics and Astronautics, Reston, Va., 1995, pp. 1–16.
4. T.R. Young, A.M. Landsberg, and J.P. Boris, "Implementation of the Full 3D FAST3D (FCT) Code Including Complex Geometry on the Intel iPSC/860 Parallel Computer," *Proc. 1993 Soc. for Computer Simulations Multiconference*, SCS, San Diego, 1993, pp. 143–148.
5. A.M. Landsberg and W.C. Sandberg, *DDG-51 FLT-IIA Airwake Study Part 3: Temperature Field Analysis for Baseline and Upgrade Configurations*, tech. report NRL/MR/6401-00-8432, US Naval Research Laboratory, Washington, D.C., 2000.
6. W. Rodi et al., "Status of Large Eddy Simulation: Results of a Workshop," *J. Fluids Eng.*, vol. 119, June 1997, pp. 248–262.
7. E.S. Oran and J.P. Boris, *Numerical Simulation of Reactive Flow*, Cambridge Univ. Press, New York, 2001.
8. J.P. Boris and D.L. Book, "Flux-Corrected Transport I: SHASTA, A Fluid Transport Algorithm that Works," *J. Computational Physics*, vol. 11, Nov. 1973, pp. 38–69.
9. J.P. Boris and D.L. Book, "Solution of the Continuity Equation by the Method of Flux-Corrected Transport," *Methods in Computational Physics*, vol. 16, 1976, pp. 85–129.
10. T. Ikeda and T. Nakagawa, "On the SHASTA FCT Algorithm," *Mathematics of Computation*, vol. 33, no. 148, 1979, pp. 1157–1169.
11. J.P. Boris et al., *LCPFCT: A Flux-Corrected Transport Algorithm for Solving Generalized Continuity Equations*, tech. report NRL/MR/6410-93-7192, US Naval Research Laboratory, Washington, D.C., 1993.
12. J.P. Boris, "On Large Eddy Simulation Using Subgrid Turbulence Models, Whither Turbulence?" *Turbulence at the Crossroads: Lecture Notes in Physics 357*, J.L. Lumley, ed., Springer-Verlag, New York, 1989, pp. 344–353.

13. J.P. Boris et al., "New Insights into Large Eddy Simulation," *J. Fluid Dynamics Research*, vol. 10, nos. 4–6, 1992, pp. 199–228.
14. C. Fureby and F.F. Grinstein, "Monotonically Integrated Large Eddy Simulation of Free Shear Flows," *AIAA J.*, vol. 37, no. 5, May, 1999, pp. 544–556.
15. C. Fureby and F.F. Grinstein, "Large Eddy Simulation of High Reynolds-Number Free and Wall Bounded Flows," *Proc. Fluids 2000*, American Inst. Aeronautics and Astronautics, Reston, Va., 2000.
16. B.Z. Cybyk et al., "A Detailed Contaminant Transport Model for Facility Hazard Assessment in Urban Areas," *Proc. 14th Fluid Dynamics Conf.*, American Inst. Aeronautics and Astronautics, Reston, Va., 1999.
17. C.F. Cox et al., "Coupled Microscale-Mesoscale Modeling of Contaminant Transport in Urban Environments," *Proc. 3rd Symp. Urban Environment*, Am. Meteorological Soc., Boston, 2000, pp. 66–67.
18. F.F. Grinstein and J.P. Boris, "Numerical Simulation of Unsteady Buoyant Flows in Buildings," *Proc. 2001 ASME Fluids Eng. Division Summer Meeting*, American Soc. Mechanical Eng., 2001.
19. B.Z. Cybyk, J.P. Boris, and T.R.Young, Jr., *Influence of Facility Characterization and External Winds on Contaminant Release Modeling*, tech report NRL/MR/6410-99-8362, US Naval Research Laboratory, Washington, D.C., 1999
20. B.Z. Cybyk, J.P. Boris, and T.R. Young, "Coupling of External Winds and Recirculations with Interior Contaminant Release Modeling," *Proc. 3rd Symp. Urban Environment*, Am. Meteorological Soc., Boston, 2000, pp. 58–59.
21. S.A. Cheatham, B.Z. Cybyk, and J.P. Boris, "Simulation of Fluid Dynamics around a Cubical Building," *Proc. 3rd Symp. Urban Environment*, Am. Meteorological Soc., Boston, 2000, pp. 46–47.
22. S.A. Cheatham, B.Z. Cybyk, and J.P. Boris, "Simulation of Flow and Dispersion around a Cubical Building," submitted, Oct. 2001.
23. I.P. Castro and A.G. Robins, "The Flow around a Surface-Mounted Cube in Uniform and Turbulent Streams," *J. Fluid Mechanics*, vol. 79, no. 2, Feb. 1977, pp. 307–335.
24. J.C.R. Hunt et al., "Kinematical Studies of the Flows around Free or Surface-Mounted Obstacles: Applying Topology to Flow Visualization," *J. Fluid Mechanics*, vol. 86, no. 1, May 1978, pp. 179–200.
25. D.A. Paterson and C.J. Apelt, "Simulation of Flow Past a Cube in a Turbulent Boundary Layer," *J. Wind Eng. and Industrial Aerodynamics*, vol. 35, nos. 1–3, Sept. 1990, pp. 149–176.
26. R. Martinuzzi and C. Tropea, "Flow Around Surface-Mounted, Prismatic Obstacles Placed in a Fully Developed Channel Flow," *J. Fluids Eng.*, vol. 115, no. 1, Mar. 1993, pp. 85–92.
27. W.H. Snyder, "Some Observations of the Influence of Stratification in Building Wakes," *Stably Stratified Flows: Flow and Dispersion over Topography*, I.P. Castro and N.J. Rockliff, eds., Oxford Univ. Press, Cambridge, Mass., 1994, pp. 301–324.
28. Y.Q. Zhang, S.P. Arya, and W.H. Snyder, "A Comparison of Numerical and Physical Modeling of Stable Atmospheric Flow and Dispersion around a Cubical Building," *Atmospheric Environment*, vol. 30, no. 8, 1996, pp. 1327–1345.
29. D. Lakehal and W. Rodi, "Calculation of the Flow Past a Surface-Mounted Cube with Two-Layer Turbulence Models," *J. Wind Eng. and Industrial Aerodynamics*, vol. 8, nos. 67–68, 1997, pp. 65–78.
30. K.B. Shah, *Large Eddy Simulation of Flow Past a Cubic Obstacle*, doctoral dissertation, Dept. of Mechanical Eng., Stanford Univ., Palo Alto, Calif., 1998.
31. S.T. Chan, D.E. Stevens, and R.L. Lee, "A Model for Flow and Dispersion around Buildings and its Validation Using Laboratory Measurements," *Proc. 3rd Symp. Urban Environment*, Am. Meteorological Soc., Boston, 2000, pp. 56–57.
32. M.A. Emery et al., "Detailed Modeling of Air Flow Around a Complex Building," *Proc. 3rd Symp. Urban Environment*, Am. Meteorological Soc., Boston, 2000, pp. 48–49.
33. M.J. Dwyer, E.G. Patton, and R.H. Shaw, "Turbulent Kinetic Energy Budgets from a Large-Eddy Simulation of Airflow above and within a Forest Canopy," *Boundary-Layer Meteorology*, vol. 84, no. 1, July 1997, pp. 23–43.

Jay Boris is the NRL Chair of Science in Computational Physics and the chief scientist and director of the Laboratory for Computational Physics and Fluid Dynamics at the US Naval Research Laboratory. His research interests include fluid dynamics and turbulence, computational physics, numerical analysis, parallel processing algorithms, and high-performance computing technology. He received a BA in physics and an MA and PhD in astrophysical sciences, all from Princeton University. He is a member of the APS, American Institute of Aeronautics and Astronautics, American Astronomical Society, International Combustion Institute, Sigma Xi, and the Washington Academy of Sciences. Contact him at the NRL, Code 6400, Overlook Ave., Washington, DC 20375; boris@lcp.nrl.navy.mil.

For more information on this or any other computing topic, please visit our Digital Library at http://computer.org/publications/dlib.