Wind effects on atria fires

Robert N. Meroney [a]

[a]*Colorado State University, Fort Collins, CO, USA, Robert.Meroney@ColoState.Edu*

ABSTRACT: A series of unsteady atria fire calculations are performed using a finite-volume CFD program about two and three dimensional generic buildings immersed in simulated atmospheric boundary layers. The model results reveal that external winds can modify the infiltration and exfiltration of air through external doors and windows, distort thermal and smoke columns rising above test fires in the atria, cause the plumes to impact directly against atria walls, and modify the resultant filling of elevated atria spaces.

1 INTRODUCTION

Typically, the wind field outside a building is not considered when specifying fire hazard systems. Moderate winds can produce sufficient pressures on a building envelope to overwhelm the expected buoyancy driven internal flow patterns. While this has been acknowledged in smoke control literature, and is tacitly incorporated into some state building code (e.g. Ohio Building Code Section 909.4.3 reads "The design shall consider the adverse effects of wind.."), no formal method of performing an analysis of these wind effects is prescribed (Klote, 1995).

One simple technique sometimes applied involves the estimation of the mean pressure coefficient (Cp) at the building's openings based on either typical coefficient of pressure, Cp, patterns for buildings of simple shape, free of upwind or downwind obstructions or a specific simulation of the unique building geometry and its environment (Meroney et al., 1995, 2004). Combined with knowledge of the local winds, wind pressures expected on the building openings can be estimated. Popular fire programs such as the CFAST zone model incorporate simple algorithms based on this approach to account for wind direction, building orientation, and opening heights on interior air movements (Peacock et al., 2008). Unfortunately, although this approach adjusts inflow and exhaust rates for wind effects, it does not actually modify the internal plume growth rate, plume rise or trajectory.

The primary concern for a natural or forced ventilation exhaust is that the wind may push the smoke back into the atria or excessive wind driven makeup air will augment the fire combustion rate (Klote and Evans, 2007). On the other hand if the smoke outlets are located in regions expected to experience wind induced suction forces, smoke removal may even be enhanced (Meroney, 2002). To characterize these processes a series of numerical calculations have been performed about simple generic 2- and 3-dimensional rectangular building atria with external doors at ground level and exhaust openings positioned about the roof.

1.1 *Natural and Forced Ventilation in Building Interiors*

The growth and behavior of fires in building atriums are affected by a combination of fire plume kinematics and the character of airflow within the atrium as driven by natural ventilation through exterior openings and forced convection due to mechanical ventilation and the presence of exterior winds about the building (NFPA, 2005). Although the joint behavior may be nonlinear, it is instructive to look at building airflow in the absence of fire. Reinforced by extensive full scale and physical model measurements specialized CFD programs are now routinely used to evaluate building ventilation (Eg. Airpak (Fluent) , Flair (Phoenics), FloVENT (Flomerics), etc.). Such programs may be used alone or in combination with physical modeling to establish exterior building flows and pressure distributions and resultant internal circulations (van Hoff and Blocken, 2009; Meroney, 2009; Phillips, 2008; Meroney et al., 1995).

Meroney (2009) used CFD to predict wind driven natural ventilation within the model structure studied by Karava (2008) in a boundary layer wind tunnel. Meroney successfully used both coupled external /internal flows and domain decomposition to replicate the wind tunnel results. Flows inside a structure are determined by the external pressure distributions produced by the external wind field, the size location and contraction coefficients associated with building openings, and the internal geometries of the building. Depending on door/window/vent openings the internal circulations can be individual clockwise or counterclockwise vortices, groups of circulation cells, or contain cross-room short circuiting between inlet and exhaust openings.

1.2 *Potential in Atria for Fire Tornadoes and Fires Deflected to Walls*

Initially fire and smoke predictions were developed presuming that external wind effects were modest and internal flows driven by fire driven natural convection or mechanical ventilation would dominate fire and smoke behavior (Klute, 1995). This led to the development of algebraic formula which predicted room zone behavior divided between those spaces dominated by the fire plume itself, ceiling spaces contaminated by smoke and heat, and lower regions still uncontaminated by smoke or heat (Peacock et al., 2008). These models (Eg. CFAST) work remarkably well when building shapes are simplistic and external influences (like wind driven cross flows) are absent. The models even incorporate a first-order estimate of the effect of external wind pressures on flow through inlet and exhaust ventilators, but they do not directly predict wind driven fire plume modifications.

Subsequently, the fire community introduced what they termed “field” models, which are essentially finite-volume CFD codes to incorporate fire dynamics, combustion dynamics, exterior wind effects, heat transfer (radiation, conduction), mitigation systems (sprinklers), and even pyrolysis of the burning objects. Again there are a number of model choices (Fire Dynamic Simulator (NIST-FDS), Smartfire (FSEG), CFD2000 (Adaptive Research), etc.). These models are capable of simulating the erratic and unsteady behavior of fires influenced by cross-wind flows. Critical in such situations is the likelihood that fire plumes will deflect against room boundaries igniting additional flammable materials and blocking balcony and stairway evacuation routes. It is even possible for asymmetric inlet flows to cause the rising thermal plume above the fire to become vortical producing a “fire tornado”. Such tornadoes tend to consume combustibles at a rapid rate, produce additional smoke and fumes, and exaggerate fire danger. Meroney (2002, 2004b, 2007) and Meroney and Banks (2004) predicted such behavior for several atria configurations where floor level door placement and elevated vents were positioned in initially unfortunate locations.

It is clear that tools now exist to predict fire and smoke behavior in complex external and interior building configurations. Unfortunately, realistic architectural arrangements are often so complex that even the examination of many case studies does not always offer basic insight into likely fire plume behavior. Hence, this paper will present some very simple building space and building opening configurations that can, hopefully, establish a preliminary fire plume/ wind effect conceptual base.

2.0 CFD SIMULATION OF WIND EFFECTS ON GENERIC ATRIA

Calculations were performed for both 2- and 3-dimensional domains. The 2-d calculations were used to examine initial impressions concerning fire plume behavior under windy conditions. A rectangular domain 200 m long and 100 m tall contained a 20 m x 20 m rectangular building located 80 m from the entrance with 3 m high door openings on the front and back surfaces and two 2 m wide exhausts located at the ceiling corners (Fig. 1). The fire was set to produce 2 MW/m heat release over a 2 m wide region at the center of the building floor. Since this was a concept test, unsteady flow calculations with the standard k-epsilon turbulence model was used with uniform inlet flow set to 0, 0.5 or 1 m/s. The grid contained 80,000 hexagonal cells.

Next a 3-d atrium was studied contained within a rectangular domain 200 m long, 100 m wide and 100 m tall. In this case 1 m x 3 m door openings were placed at the center of all four building sides, and four 2 m x 2 m exhaust vents were placed near all four corners of the ceiling (Fig. 2). The fire was set to 2.5 MW over a 2 sq m region at atrium center.[1] A large eddy simulation (LES) turbulence model was specified with uniform inlet flow of 0 and 2 m/s. The grid contained 442,000 hexagonal cells.

3.0 RESULTS OF NUMERICAL CALCULATION

The primary results from these calculations are the visualizations of fire plume behavior for different wall openings and various approach wind conditions. Video clips (*.avi format) have been archived on a Colorado State web site under the category of Recent Research Presentations, where they can be downloaded on demand. http://www.engr.colostate.edu/~meroney/index.html) The behavior of the various test cases are summarized below in prose statements that describe the primary behavior of the atrium fire plumes.

3.1 *Wind Effects on 2-Dimensional Generic Atria*

As we examine the 2-dimensional results we must acknowledge that the configuration proscribes flow around the building or internal flow around the fire source; hence, effective blockage is larger both inside and outside the atrium for a 2-dimensional than for a 3-dimensional building. 2-dimensional inlet and exhaust vents are also effectively larger; hence, the flows are not directly comparable to the 3-d situation.

Figures 3 to 8 display results for 2-dimensional conditions with both up and downwind doors open, and Figure 9 displays results when only the upwind door is open. Figures 3 and 4 present temperature, streamline, velocity magnitude, density and pressure contours for zero wind conditions. The flow field is entirely symmetric about the building centerline. The fire plume impacts the ceiling at a central point between the two ceiling exhausts. Warm gases exhaust through the ceiling vents, but the upper atrium zone mixes downward quickly until fumes fill most of the atrium above door level, and the flow reaches an equilibrium condition after about t = 25 sec. Outside the building two symmetric thermal plumes rise above the ceiling vents until they are deflected laterally by the upper symmetry condition set at the top of the domain. (One might imagine the top is an effective inversion level in the atmosphere.)

At higher approach wind speeds the fire plume inside the atrium is deflected toward the rear wall, but inflow through the rear door balances this tendency somewhat and the plume does not impact against the rear wall (Figs 5 and 6 for U = 0.5 m/s and Figs 7 and 8 for U = 1.0 m/sec.) As the wind speed increases the thermal plumes exhausting through the ceiling vents are progressively bent further over by the approach wind, but the internal flows seem to remain similar at speeds above U = 0.5 m/sec.

---

[1] **Error! Main Document Only.**Design fires for atria are usually selected to be 2 to 5 MW for atria with restricted fuel and atria with combustibles, respectively (Klote and Milke, 2002; International Code Council, 2006)

When the rear door is closed the fire plume inside the atrium is deflected strongly toward the rear wall even for zero external wind conditions (Fig. 9a), and at finite wind speeds the fire plume is driven completely to the horizontal and gases circulate counter-clockwise within the atrium bathing the rear wall in hot gases (Fig. 9b).

3.2 *Wind Effects on 3-Dimensional Generic Atria*

The 2 and 3-dimensional atria display qualitatively similar flow behavior, but the time scales and effective buoyancy fluxes are significantly different. The inlet and exhaust openings are effectively smaller for the 3-dimensional cases, the pressure coefficients on the faces of the building are less since flow can deviate around the building, and the effective strength of the fire is also less.

As noted in both Figure 10 for U = 0 m/sec and Figure 11 for U = 2 m/sec the centerline sections of temperature contours are very similar despite higher external wind speeds. The ceiling thermal (smoke) layer reaches door opening heights at about the same time (~200 sec). Since ground level openings exist on all four sides of the building there does not appear to be any primary redirection of the central fire plume toward a wall. Thermal plumes rising above the ceiling exhausts are deflected by the approach wind but appear to raise out of the rooftop separation bubbles and do not re-enter the wake cavity in this simulation.

4 SUMMARY

These CFD calculations do reveal the nature of how external wind can deflect and change the character of fire plumes growing within a building atrium. Care must be taken, however, in projecting 2-dimensional impressions to a 3-dimensional configuration because the inherent nature of the 2-dimensional flow field is different in important respects. One must also be careful in how one characterizes the dimensionless parameters when comparing 2- and 3- dimensional configurations.

These calculations and configurations are limited and preliminary, and it would be worthwhile to extend the set of building geometries, building orientations, fire sizes, and openings selected to a greater parameter set. The tests are also limited to cases driven by natural convection and do not consider fan-driven forced ventilation.

5 REFERENCES:

International Code Council, 2006, *International Fire Code*, 434 pp.

Karava, Panagiota ,2008, Airflow Prediction in Buildings for Natural Ventilation Design: Wind Tunnel Measurements and Simulation, *Ph.D. Thesis, Dept. of Building, Civil and Environmental Engineering*, Concordia University, Montreal Quebec, 242 pp.

Klote, J.H.,1995, Smoke Control, Chapter 12 in Section 4 of NFPA SFPE 95, *SFPE Handbook of Fire Protection Engineering*, National Fire Protection Assoc., Quincy, MA, DiNenno, P. J. et al. eds.,, Section 4: 230_245.

Klote, J.H. and Milke, J., 2002, *Principles of Smoke Management*, ASHARAE/SFPE, Atlanta, GA.

Klote, J.H. and Evans, D.H., 2007, *A Guide to Smoke Control in 2006 IBC*, Int. Code Council, Country Club Hills, IL, USA, 214 pp.

Meroney, R.N., 2002, Containment of Fire and Smoke in Building Atria: Examination of Virtual Hazards, *Proceedings Workshop on Virtual HVAC on CD-R*, Centre de Rechere en Calcul Applique, 67 slides.

Meroney, R.N., 2004a, Wind tunnel and numerical simulation of pollution dispersion: a hybrid approach. Working Paper. *Croucher Advanced Study Institute on Wind Tunnel Modeling*, Hong Kong University of Science and Technology, 6–10 December, 2004, 60 pp.

Meroney, R.N., 2004b, Fires in Porous Media: Natural and Urban Canopies, *NATO Advanced Study Institute on Flow and Transport Processes in Complex Obstructed Geometries: from cities and vegetative canopies to industrial problems*, Kiev, Ukraine, 4-15 May 2004, 10 pp.

Meroney, R.N., 2007, Numerical Prediction of Fire Propagation in Idealized Wildland and Urban Canopies, *Proceedings of 12th International Conference on Wind Engineering (ICWE12)*, 1-6 July 2007, Cairns, North Queensland, Australia, 8 pp.

Meroney, R.N., 2008, Virtual Fires via Computers, *1st American Association for Wind Engineering Workshop*, Session 2: CWE and Analytical, 21-22 August 2008, Vail, CO, 8 pp. CEP08-09-1

Meroney, R.N., 2009, CFD Prediction of Airflow in Buildings for Natural Ventilation, *Proceedings 11th Americas Conference on Wind Engineering*, San Juan, Puerto Rico, June 22-26, 2009, 11 pp.

Meroney, R.N. and Banks, 2004, D., Smoke and Fire in Building Atria, *1st Int. COE Symposium on Wind Effects on Buildings and Urban Environment*, Yokohama, Japan, 8-9 March 2004, 15 pp.

Meroney, R.N., Neff, D.E., and Birdsall, J.B., 1995, Wind-tunnel Simulation of Infiltration Across Permeable Building Envelopes: Energy and Air Pollution Exchange Rates, *Proceedings of the 7th International Symposium on Measurement and Modeling of Environmental Flows*, San Francisco, CA, 8 pp.

NFPA, 2005, *Standard for Smoke Management Systems in Malls*, *Atria and Large Spaces*, NFPA 92B, Quincy, MA, 57 pp.

Peacock, R.D., Jones, W.W., Reneke, P.A., and Forney, G., 2008, *CFAST – Consolidated Model of Fire Growth and Smoke Transport (Version 6): User's Guide*, NIST Special Publication 1041, 110 pp.

Phillips, D.A., 2008, Who Can See The Wind: Designing Natural Ventilation, *Canadian Consulting Engineer*, August/September 2008, Vol. 49, No. 5, pp. 34-40.

Van Hooff, T. and Blocken, B., 2009, Coupled urban wind flow and indoor natural ventilation modelling on a high resolution grid: A case study for the Amsterdam ArenA stadium, *Environmental Modelling & Software*, 1-15 pp.

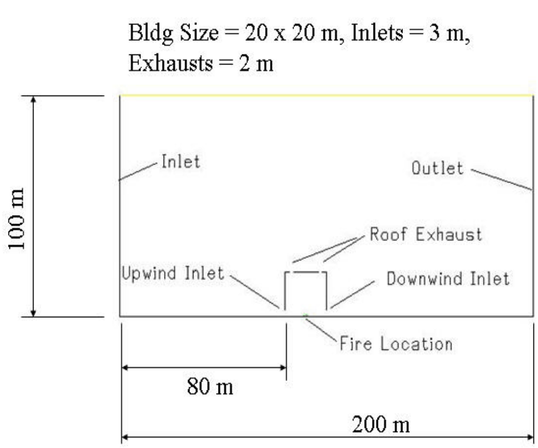


Figure 1. Schematic 2-d generic building atrium domain.

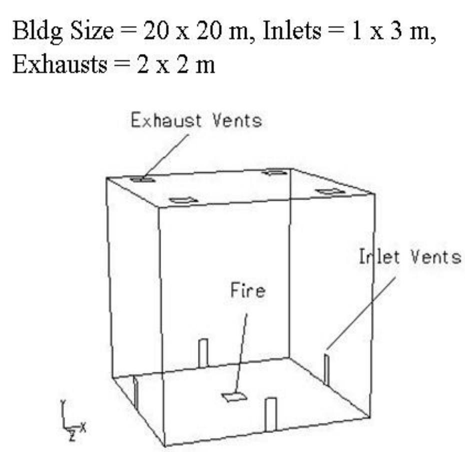


Figure 2. Schematic 3-d generic building atrium.

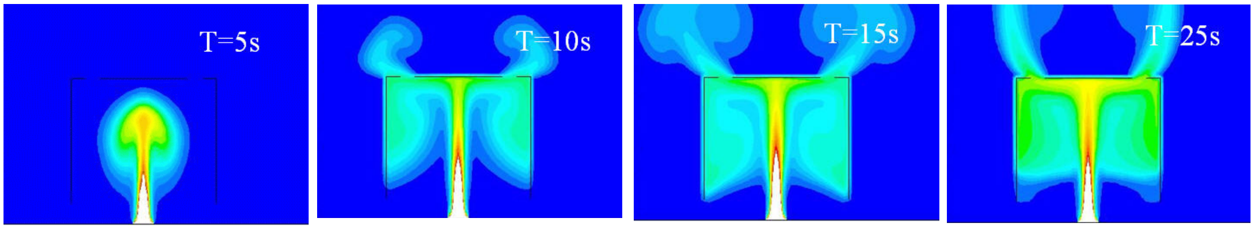


Figure 3. Temperature Contours 2-d Building, Up and downwind door openings, U = 0 m/s, t = 5, 10, 15 & 25 sec.

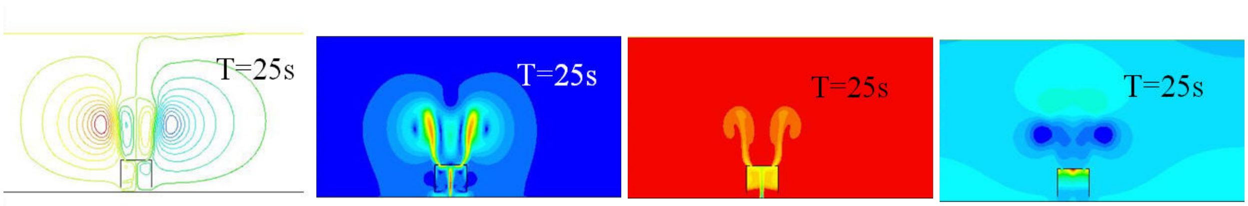


Figure 4. Stream lines, velocity magnitude, density and static pressure contours, 2-d Building, Up and downwind door openings, U = 0 m/s, t = 25 sec.

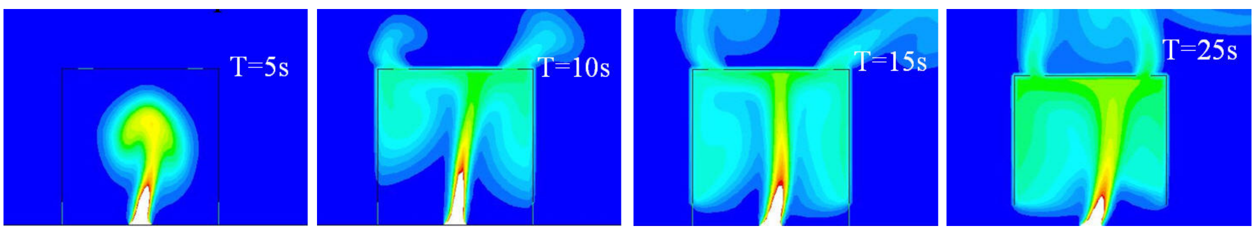


Figure 5. Temperature Contours 2-d Building, Up & downwind door openings, U = 0.5 m/s, t = 5, 10, 15, and 25 sec.

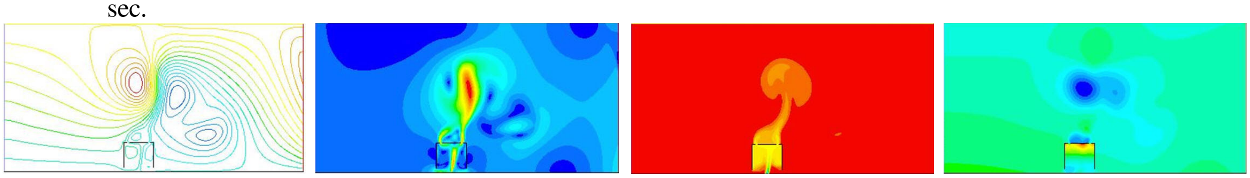


Figure 6. Stream lines, velocity magnitude, density and static pressure contours, 2-d Building, Up and downwind door openings, U = 0.5 m/s, t = 35 sec.

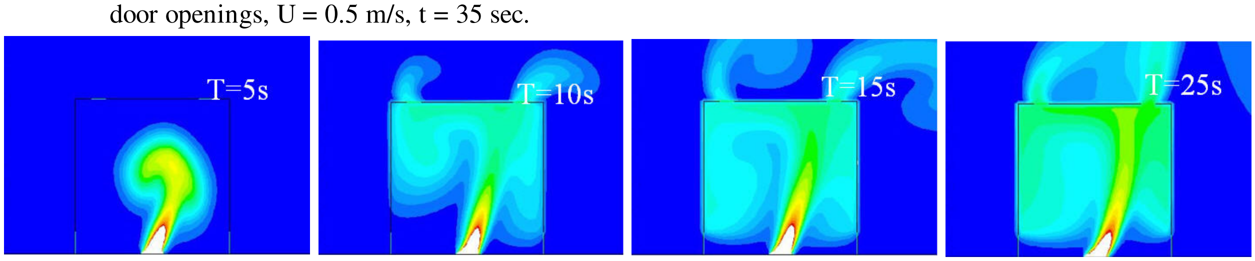


Figure 7. Temperature Contours 2-d Building, Up and downwind door openings, U = 1 m/s, t = 5, 10, 15, and 25 sec.

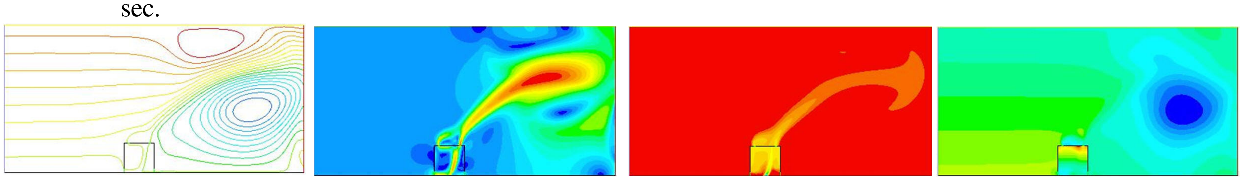
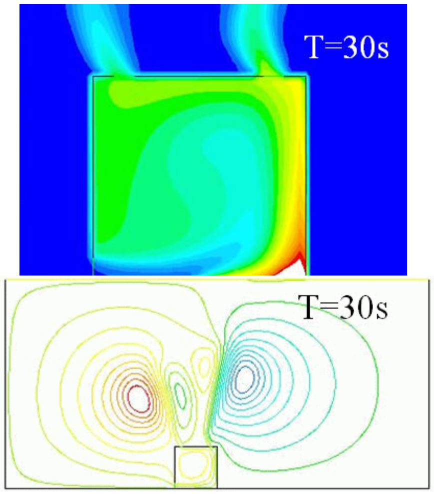


Figure 8. Stream lines, velocity magnitude, density and static pressure contours, 2-d Building, Up and downwind door openings, U = 1 m/s, t = 35 sec.

a)



b)

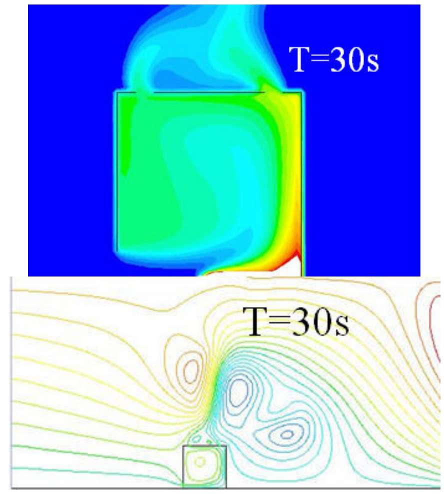


Figure 9. 2-d Building, Upwind door openings, Temperature contours and Stream lines, t = 35 sec.
a) U = 0.0 m/sec, b) U = 0.5 m/sec

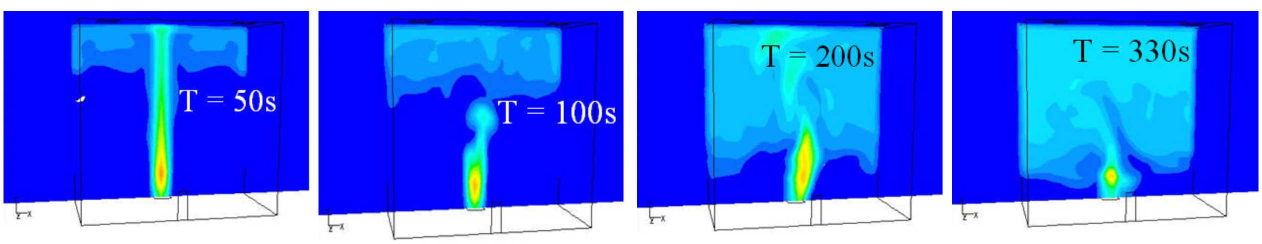


Figure 10. Temperature Contours 3-d Building, U = 0 m/sec. t = 50, 100, 200 and 330 sec.

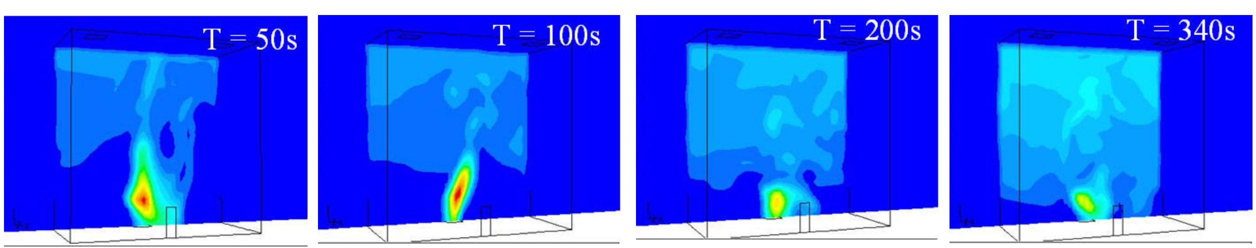


Figure 11. Temperature Contours 3-d Building, U = 2 m/sec. t = 50, 100, 200 and 330 sec.