Wind effects on atria fires

Robert N. Meroney [a]

[a] *Wind Engineering Software, 6218 Eagle Ridge Court, Fort Collins, CO 80525, USA*,
Phone: (970) 482-9252, Email: 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. In some cases aggressive fire "whirls" form which can enhance fire strength, enclosure mixing, and exposure. Results are compared qualitatively with similar physical model experiments.

KEYWORDS: Building ventilation, atria, atria fires, computational fluid dynamics,
WORD COUNT: 4200, 12 Figures

1 INTRODUCTION

The movement of fire-generated smoke in buildings is of considerable practical importance since smoke exposure is the major killer in fire situations. Indeed, fire kills more Americans than all natural disasters combined. Between 1999 and 2008, there was an average of 1,634,150 fires resulting in an estimated $11,634,800,000 in direct dollar loss each year. An average of 3,625 Americans lost their lives and another 18,765 were injured annually as the result of fire. Residential structure fires represented 27.8 percent of all fires and 78.3 percent of all structure fires. Nonresidential structures include educational, institutional, public assembly, stores and offices, industry, utility, defense, storage, and special structures. (U.S. Fire Administration, 2008) Given the severity of fire hazards considerable effort has been to mitigate fires though proper building ventilation, evacuation routes, sprinkler and smoke control systems. Today there are extensive building and fire codes which control occupant exposure based on physical experiments, analytic algorithms, and even computational fluid dynamics. (International Code Council, 2006). Nonetheless, most experience and attention is focused on conventional single or multiple floor buildings with limited ventilation connections to the external environment. In particular large nonresidential structures such as sports arena, airline and rail terminals, plane hangers, churches, warehouses, and agricultural storage buildings (barns, stables and silos) produce opportunities for the interaction of fire plumes with cross-flow ventilation produced by exterior winds through large exterior openings.

Typically, the wind field outside a building is not considered when specifying fire hazard systems. Yet 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. Poreh and Trebukov (2000) have proposed relations which suggest threshold wind speeds at which internal smoke layers in atriums would be modified by enhanced flow through elevated exhaust vents. They predicted that the probability of exceeding threshold wind speeds in case situations in Israel or Scotland can range from 1 to 25% of the time depending on atrium height and fire heat output. Higher wind speeds were calculated to reduce smoke visibility substantially. Yet even their calculations did not address the impact of inlet winds directly changing the growth, position and character of the fire plume itself which is the concern here. Popular fire programs such as the CFAST zone model incorporate simple algorithms based on the adjusted Cp 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. Satoh and Yang (2000) and Zhou and Wu (2007) examined the generation of fire whirls by multiple fires using physical and numerical models. Satoh and Yang found that different fire configurations (Eg. simultaneous burning of furniture at different room locations) can produce “severe fire hazards in the form of horizontal fire whirls located near the lower parts of the room walls with a high destructive power.” Fortholfer et al. (2009) numerically demonstrated how strong fire whirls can be produced by geometrically asymmetric fires (Eg. fires burning over an L-shaped region). They found wind orientation significantly affected fire growth; hence, ambient winds, fire size, shape and location within an atrium may change the hazard of a given fire substantially. Yet current codes presume that these factors are secondary.

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).

A second series of calculations were performed with each 1 m x 3 m door opening offset 4 m from the centerline such that a counterclockwise rotation was induced when observed from the top of the atrium. 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.

All calculations were performed using the FLUENT 6.1 finite-volume CFD code. Unsteady realizable kappa-epsilon with standard wall functions and/or large-eddy turbulence simulation models were used. (Since no significant differences were seen in the results and qualitative flow behavior was emphasized, no conclusions are drawn here concerning the turbulence models.) Air was treated as a constant property incompressible ideal gas. The solution methods used were pressure-based segregated solutions of the momentum and energy equations, Time steps were set at a constant rate of 0.1 seconds. Inlet flows imposed were permitted to be uniform velocity and

---

[1] 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)

constant turbulent intensity (10% turbulent intensity and a 10m turbulent length scale), and no effort was made to impose a particular boundary-layer profile. More realistic boundary-layer profiles may induce quantitatively different results, but the general flow structure around and within the simple box atrium should be similar.

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#projects under a link to Recent Research. Accessed July 2010.)

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 over further by the approach wind, but the internal flows seem to remain similar at speeds above U = 0.5 m/sec.

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: Symmetric external doors*

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 Fig. 10 for U = 0 m/sec and Fig. 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. Chow and Han (2009) performed model experiments for a fire at the base of a vertical shaft (height 145 cm and cross section 34 x 35 cm) in which symmetric ventilation openings were provided at the bottom of the shaft (door heights, h, and width, w, of 2.5 x 27.5 cm and 7 x 7 cm; Tests C1 and C2). During these experiments ambient winds were zero which resulted in modest flame heights with no observable swirl. Their results were thus qualitatively similar to the numerical calculations discussed above.

3.3 *Wind Effects on 3-Dimensional Generic Atria: Asymmetric external doors*

Asymmetric external doors (4 m offset from wall centerline) quickly produced counter clockwise fire whirl rotation for ambient wind speeds, U = 0 and 2 m/sec. In both cases the flow developed a strong vortical flow that extended to the atrium ceiling. When the vortex impacted the ceiling it spread rapidly radially producing impact on the side walls and gusts which mixed downward filling the upper space with heated air. Meanwhile the plume vortex tended to precess around the building floor frequently wandering outside the heated 2 m sq central simulated fire source region.

The primary affect of the ambient winds were to change the average vertical orientation of the rising fire plume inside the atrium. For U = 0 m/sec the plume was unsteady, but it remained centered at the primary room axis. For U = 2.5 m/sec (Fig. 12) the plume tended to lean toward the inside surface of the upwind building wall resulting in enhanced contact of the rising plume against that wall. This was likely a result of the higher inlet velocities occurring due to flow stagnation on the upwind wall door opening, the lower velocities entering through the side and downwind door openings as a result of lower external pressures in side separated flow regions, and the resulting low pressure region which occurred to the side of the inlet door jet which drew the rising plume toward the upwind wall.

The behavior of the 3-dimensional cube flow is in strong contrast to the 2-dimensional study in which ambient winds tended to strongly deflect the fire plume against the downwind atria wall. The difference in flow patterns can be attributed to the cross-plane circulations present during 3 dimensional flows.

Chow and Han did not examine experimental situations in which bottom openings were placed asymmetrically; however, they did examine the effect of a vertical corner gap with and without the presence of bottom openings. They considered vertical corner gaps of various widths, heights, and vertical location. They found that fire whirls occurred only for gap widths exceeding a minimum value, gaps which began at floor level and gaps which exceeded a minimum height. Fire whirls did not occur when the corner gap was placed significantly above the ground level, and the presence of central bottom openings seemed to inhibit corner gap flows which otherwise produced fire whirls. They conjectured that supplying air from the bottom of an atrium might actually inhibit fire whirls. For centrally and symmetrically placed openings at the base of an atrium their experiments confirm the numerical results provided here in Section 3.2; however, for door openings placed asymmetrically about the base the numerical calculations suggest that fire whirls can be produced as effectively as with single corner gaps of appropriate width and height.

4.0 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.

5.0 ROLE OF THE FUNDING SOURCE:

This article was not supported by separate contractor or government funding agency, but was prepared under the author's sole and independent support. The use of the Academic version of FLUENT 6.3 under a user agreement with Colorado State University is gratefully acknowledged.

6 REFERENCES:

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

Forthofer, J., Shannon, K. and Butler, B., 2009, Investigating Causes of Large Scale Fire Whirls Using Numerical Simulation, *Microscale/Coupled Modeling Conference*, American Meteorological Society, Oct 12, 2009, 8 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. (http://www.engr.colostate.edu/~meroney/CFD-papers.htm link to item number 61. Accessed 7/2010)

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. (http://www.engr.colostate.edu/~meroney/PapersPDF/RNM-reviewed_papers.htm link to item number 210. Accessed 7/2010)

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. (http://www.engr.colostate.edu/~meroney/PapersPDF/RNM-reviewed_papers.htm link to item number 205. Accessed 7/2010)

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. (http://www.engr.colostate.edu/~meroney/CFD-papers.htm link to item number 74. Accessed 7/2010)

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. (http://www.engr.colostate.edu/~meroney/CFD-papers.htm link to item number 77. Accessed 7/2010)

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. (http://www.engr.colostate.edu/~meroney/CFD-papers.htm link to item number 79. Accessed 7/2010)

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. (http://www.engr.colostate.edu/~meroney/PapersPDF/RNM-reviewed_papers.htm link to item number 204. Accessed 7/2010)

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. (http://www.engr.colostate.edu/~meroney/PapersPDF/RNM-reviewed_papers.htm link to item number 159. Accessed 7/2010)

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.

Poreh, M. and Trebukov, S., 2000, Wind effects on smoke motion in buildings, *Fire Safety Journal*, Vol. 35, 257-273.

Satoh, K. and Yang, K.T., 2000, A horizontal fire-whirl design scenario for engineering performance-based fire-code applications, *Int. J. on Eng. Performance-Based Fire Codes*, Vol. 2, No. 2, 48-57.

U.S. Fire Administration (2008), The Overall Fire Picture – 2008, Quick Statistics, http://www.usfa.dhs.gov/statistics/quickstats/index.shtm and http://www.usfa.dhs.gov/statistics/reports/ , FEMA (accessed 7/2010).

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*, Vol. , 1-15 pp.

Zhou, R. and Wu, Z.N., 2007, Fire whirls due to surrounding flame sources and the influence of the rotation speed on the flame height, *J. Fluid Mechanics*, Vol. 583, 313-345.

FIGURE CAPTIONS:

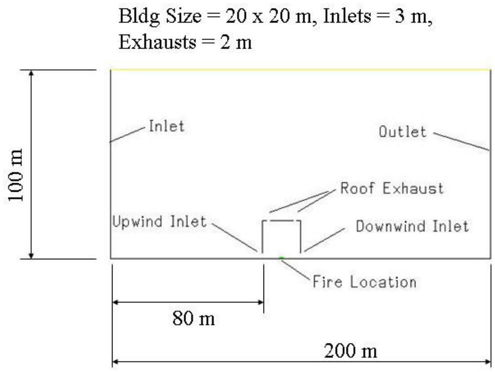

Figure 1. Schematic 2-d generic building atrium.

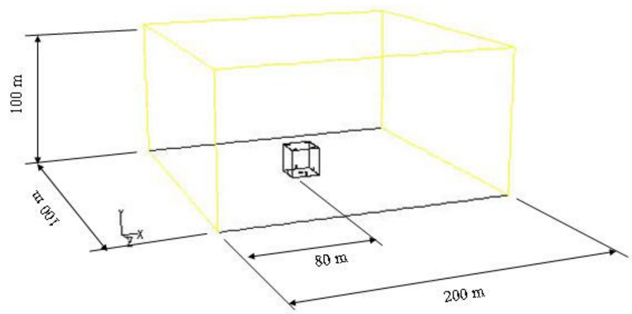

Figure 2. Schematic 3-d generic building atrium, symmetric door openings.

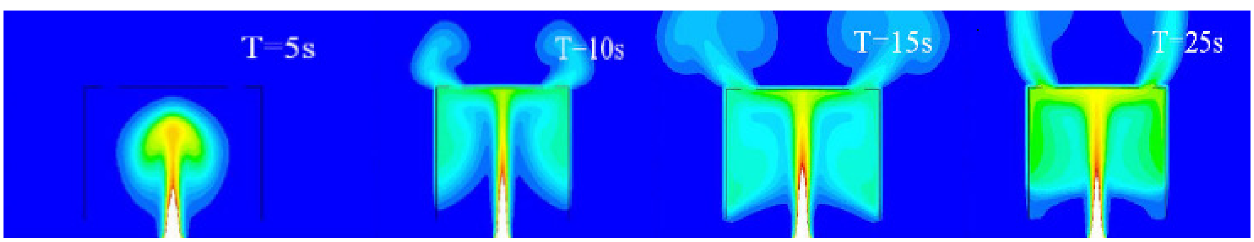

Figure 3. Temperature Contours 2-d Building, Up and downwind door openings, U = 0 m/s, t = 5, 10, 15 & 25 sec.

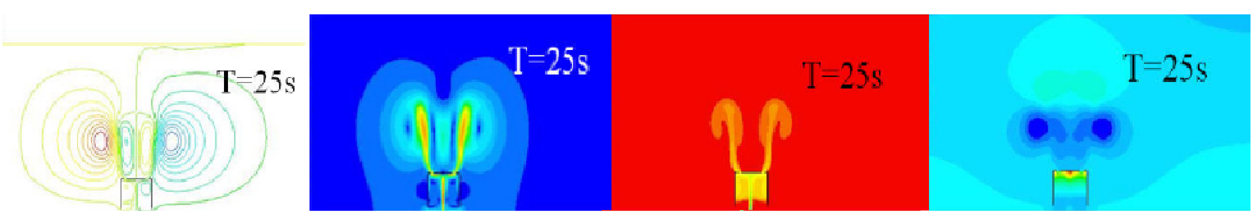

Figure 4. Stream lines, velocity magnitude, density and static pressure contours, 2-d Building, Up and downwind door openings, U = 0 m/s, t = 25 sec.

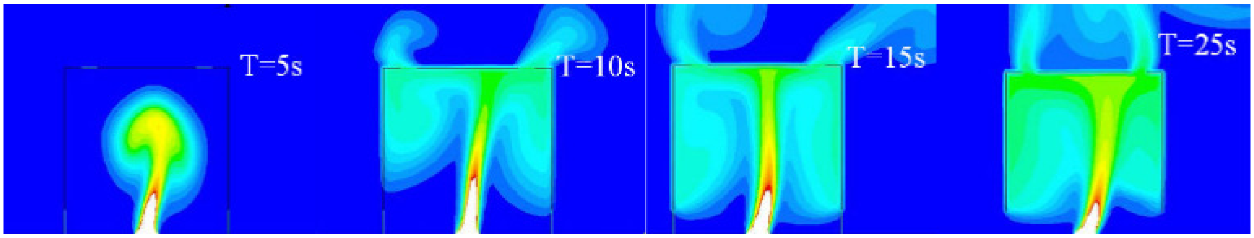

Figure 5. Temperature Contours 2-d Building, Up and downwind door openings, U = 0.5 m/s, t = 5, 10, 15, and 25 sec.

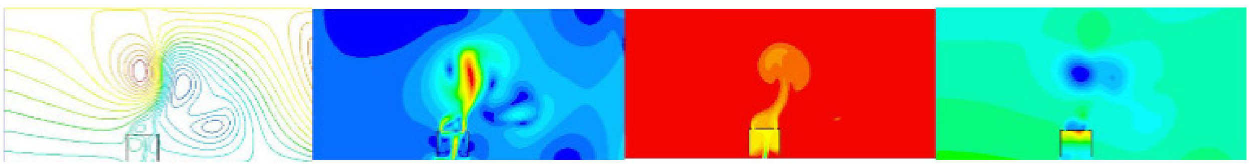

Figure 6. Stream lines, velocity magnitude, density and static pressure contours, 2-d Building, Up and downwind door openings, U = 0.5 m/s, t = 35 sec.

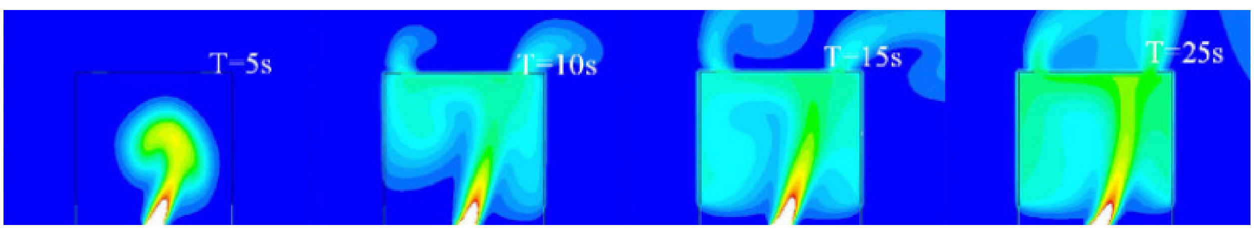

Figure 7. Temperature Contours 2-d Building, Up and downwind door openings, U = 1 m/s, t = 5, 10, 15, and 25 sec.

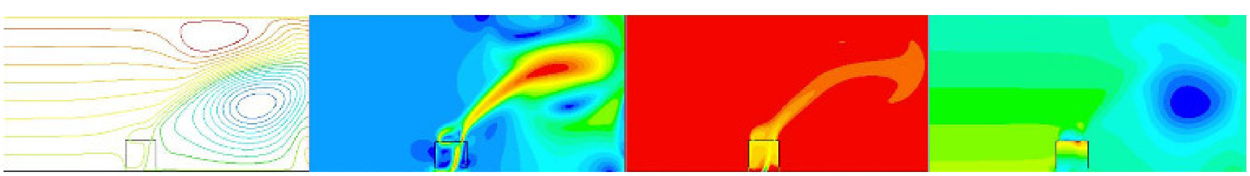

Figure 8. Stream lines, velocity magnitude, density and static pressure contours, 2-d Building, Up and downwind door openings, U = 1 m/s, t = 35 sec.

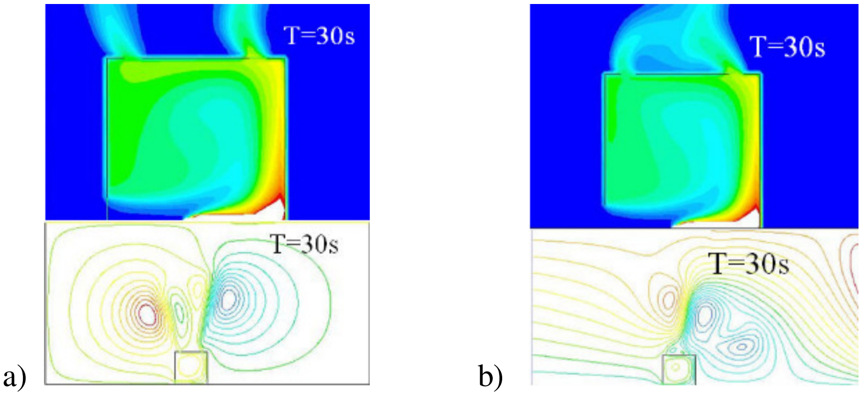

Figure 9. 2-d Building, Upwind door openings, Temperature contours and Stream lines, t = 30 sec.
a) U = 0.0 m/sec b) U = 0.5 m/sec

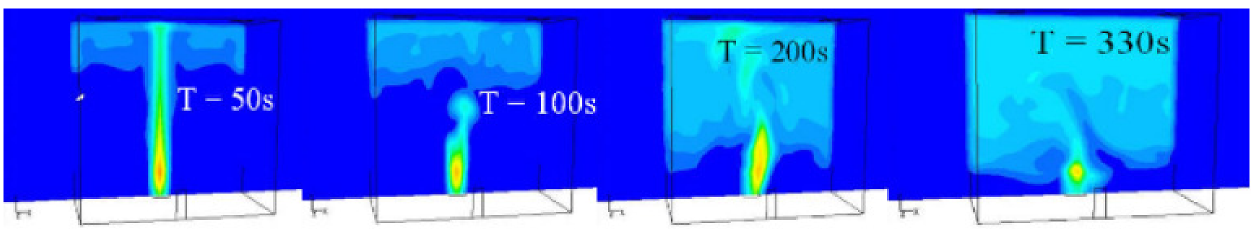

Figure 10. Temperature Contours 3-d Building, U = 0 m/sec. t = 50, 100, 200 and 330 sec. Symmetric door openings.

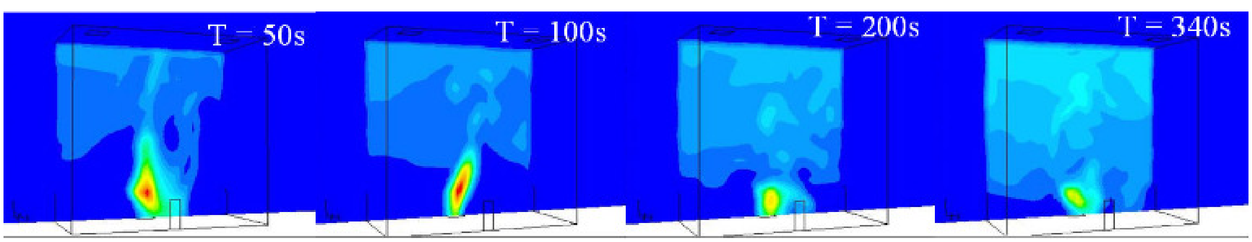

Figure 11. Temperature Contours 3-d Building, U = 2 m/sec. t = 50, 100, 200 and 340 sec. Symmetric door openings.

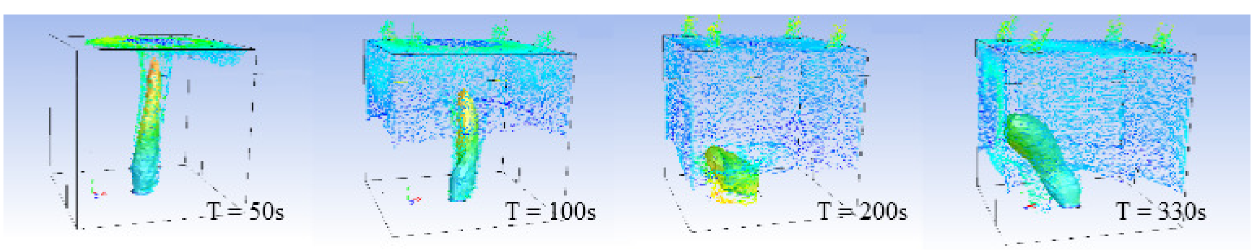

Figure 12. Temperature Contours 3-d Building, U = 2 m/sec. t = 50, 100, 200 and 330 sec. Asymmetric door openings.