2014

High-Resolution Performance Analysis of a Large Building with Linear Dispersion Ductwork System

Anthony Fontanini
Iowa State University

Alberto Passalacqua
Iowa State University, albertop@iastate.edu

Umesh Vaidya
Iowa State University, ugvaidya@iastate.edu

Michael Olsen
Iowa State University, mgolsen@iastate.edu

Baskar Ganapathysubramanian
Iowa State University, baskarg@iastate.edu

High-Resolution Performance Analysis of a Large Building with Linear Dispersion Ductwork System

Abstract
Buildings consume 40% of the U.S. annual energy usage. The energy usage of building can drastically be reduced by proper management of the thermal load in large interior spaces. A well designed space should include systems that balance both occupant comfort and efficiency. Recently linear dispersion ductwork systems have been shown to be a promising solution for creating both comfortable and efficient spaces. We investigate long-term thermal behavior of a 10,000 ft2 building that utilizes a linear air dispersion system. We optimize the operating point of the HVAC system by utilizing three dimensional Computational Fluid Dynamic (CFD) simulations. The long-term air flow, comfort, and energy metrics of the building are analyzed for a number of different flow rates. The steady state simulations consist of 86 million degrees of freedom and resolve physics for length scales that span three orders of magnitude. The simulations utilize High Performance Computing (HPC) and use hundreds of processors to perform the simulations. Finally, challenges associated with the analysis and visualization of the extremely large data sets is discussed.

Disciplines
Acoustics, Dynamics, and Controls | Computer-Aided Engineering and Design | Mechanical Engineering

Comments
This proceeding is published as Fontanini, Anthony. "High-Resolution Performance Analysis of a Large Building with Linear Dispersion Ductwork System." ASHRAE Transactions 120 (2014): S1. Posted with permission.

High-Resolution Performance Analysis of a Large Building with Linear Dispersion Ductwork System

Anthony Fontanini **Alberto Passalacqua, PhD** **Umesh Vaidya, PhD**

Michael Olsen, PhD **Baskar Ganapathysubramanian, PhD**

ABSTRACT

Buildings consume 40% of the U.S. annual energy usage. The energy usage of building can drastically be reduced by proper management of the thermal load in large interior spaces. A well designed space should include systems that balance both occupant comfort and efficiency. Recently linear dispersion ductwork systems have been shown to be a promising solution for creating both comfortable and efficient spaces. We investigate long-term thermal behavior of a 10,000 ft² building that utilizes a linear air dispersion system. We optimize the operating point of the HVAC system by utilizing three dimensional Computational Fluid Dynamic (CFD) simulations. The long-term air flow, comfort, and energy metrics of the building are analyzed for a number of different flow rates. The steady state simulations consist of 86 million degrees of freedom and resolve physics for length scales that span three orders of magnitude. The simulations utilize High Performance Computing (HPC) and use hundreds of processors to perform the simulations. Finally, challenges associated with the analysis and visualization of the extremely large data sets is discussed.

INTRODUCTION

Buildings currently use about 40% of the U.S. annual energy usage [1]. Because of the large energy usage, an enormous effort has been invested into reducing the energy usage in buildings. A couple of strategies to improve the building's performance are to increase the efficiency of HVAC systems and air distribution systems, and to reduce the peak thermal loads on the building's envelope through cost effective insulation. Often these approaches are addressed separately, which can lead to less optimal solution in terms of the sizing of the HVAC system and/or an overinvestment in the amount of insulation in the building's envelope. The current study combines these strategies and investigates how a linear dispersion ductwork system of a large commercial building is affected by the building's envelope.

Recently linear air dispersion ductwork system has been shown to be a promising solution to developing a comfort and efficient environment. Linear air dispersion systems distribute air evenly along a length of ductwork, which helps to decrease hot and cold areas and provide more uniform fresh air in the building. In our previous work we have shown that

A. Fontanini is PhD student in the Department of Mechanical Engineering, Iowa State University, Ames, IA. A. Passalacqua is a professor in the Department of Mechanical Engineering, Iowa State University, Ames, IA. U. Vaidya is a professor in the Department of Electrical and Computer Engineering, Iowa State University, Ames, IA. M. Olsen is a professor in the Department of Mechanical Engineering, Iowa State University, Ames, IA. B. Ganapathysubramanian is a professor in the Department of Mechanical Engineering, Iowa State University, Ames, IA.

fabric based linear dispersion ductwork systems heat smaller spaces relatively quickly in a very uniform, comfortable, and efficient manner [2]. Fabric linear dispersion ductwork systems have become more common in buildings due to light weight design, ease of installation, and cleanliness. Because of these advantages linear dispersion systems have also been a topic of resent research, [3-5]. Although some research exists on improving the design of these air distribution systems a limited amount of research has been completed on how these system are affected by the building's envelope.

The effect of the building's envelope on the HVAC system was performed using high resolution Computational Fluid Dynamics (CFD) simulations. CFD simulations allow for visualization of the air flow structures in buildings, the ability to calculate both comfort and efficiency metrics, and estimate how changes in the air distribution system directly affect the HVAC system. These capabilities among others have helped make CFD analysis a powerful non-intrusive virtual tool for design in the ventilation community.

Our contributions in this paper are the following: 1) We perform high resolution simulations to evaluate the air flow physics in large commercial buildings with linear dispersion ductwork systems, 2) Present strategies to properly manage extremely large data sets, 3) Show that changing the insulation has a dramatic effect on operation point of the HVAC system and thermal comfort of the occupied space.

PROBLEM DEFINITION

The problem the present work seeks to address is to analyze the effect of the building's envelope on linear dispersion air distribution systems for large commercial buildings. The physical space being studied represents a large commercial warehouse or indoor athletic field located in Des Moines, Iowa. The simulations performed represent the steady state flow characteristics produced by the designed ductwork system during peak cooling conditions. The simulations provide valuable information about the level of comfort in the building and how efficiently the ductwork system is performing. The metrics investigated for comfort and efficiency includes the air distribution performance index (ADPI), mean draft temperature, and cooling capacity needed by the HVAC system to overcome the energy loads from the building's envelope.

The design of the building's envelope is based on the requirements in ASHRAE 90.1 [6]. The interior of the building has a floor area of 10,000 [ft^2] or 929 [m^2] (100 [ft] x 100 [ft] or 30.48 [m] x 30.48 [m]) and a height of 25 [ft] (7.62 [m]), **as shown in Figure 1**. The linear dispersion ductwork system is placed 20 [ft] above the floor. The ductwork design is based on an actual ductwork product which is 45 [ft] 8 [in] (13.92 [m]) long, a diameter of 28 [in] (0.71 [m]). Eighty larger diameter holes (2.25 [in] or 5.7 [cm]) are located at 3:40 and 8:20 on the cross-section of the ductwork. One hundred and two smaller diameter holes (1 [in] or 2.54 [cm]) are located at 5:00 and 7:00 on the cross-section of the ductwork. The air return for the building is placed above the origin on the ceiling and has dimensions 6 [ft] x 6 [ft] (2 [m] x 2[m]).

SIMULATION METHODOLOGY

Turbulence Modeling

The small holes in the linear dispersion based ductwork system present unique challenges when modeling the ventilation dynamics of a space. One of the challenges is the result of relatively high Reynolds numbers when the air is exiting the dispersion tube combined with lower air flow in the center of the space. The inlet Reynolds number for the air flow rates investigated is 3000 – 16000, with the inlet velocity and characteristic length based on the air speed and diameter of the smallest hole in the ductwork.

An appropriate choice of the turbulence model is essential to accurately representing the air flow characteristics. The compressible form of the renormalized group (RNG) $k - \varepsilon$ model was chosen for its simplicity, relative low computational cost, and also because it has been shown to have a large range of applicability [7-12]. The RNG $k - \varepsilon$ model is RANS model. The model assumes that the air flow is composed of a mean and fluctuating component [13]. The governing equations are conservation of mass, conservation of linear momentum, conservation of energy and the two transport equations for k and ε. For brevity, we do not give the governing equations, but the equations can be found in [14].

Boundary Conditions

In order to solve the governing equations, boundary conditions are needed for all the solution variables at the boundaries of the domain. For brevity, only some details on the calculation of the boundary conditions are given, while a detailed discussion of boundary conditions is deferred to a subsequent publication. The air flow enters the domain normal to the inlet holes in the ductwork. The temperature of the inlet air was 21.1 [C] (70 [F]). The velocity of the air at each of the holes is based on an area weighted velocity that satisfies the desired air flow rate. The boundary conditions for the turbulent parameters k and ε follow turbulent theory of jets and boundary layers given by G. Papadopoulus [15] and S.B. Pope [13] respectively. The temperature of the walls, ceiling, and floor were calculated by the Thermal Analysis Research Program (TARP) algorithm originally presented by G.N. Walton [16] and discussed at length in the EnergyPlus Engineering Reference [17]. The interest of the study is the peak loads during the summer months. The temperature data for the high temperatures were taken from WeatherSpark. The exterior temperatures are based on the upper 90%-ile in July in the location of Des Moines Iowa. The solar loads needed for the analysis is taken from the "Solar radiation Data Manual for Buildings" [18]. The insulation of the walls, floor and ceiling were taken from ASHRAE 90.1 [6] for a building construction of a metal/steel framed building.

Domain Discretization

The numerical approach to solving the equation begins with a discretization of the domain into a finite volume representation of the building. The complex geometry of the industrial based linear dispersion fabric ducting presents discretization challenges. The length scales of the problem span 100 [ft] to 0.083 [ft], the length of one side of the building and the diameter of the small hole set respectively. The large difference in length scales potentially leads to large velocity gradients in some parts of the domain (near the inlet holes and the boundaries). A non-uniform discretization scheme that selectively refined regions of large gradients was used to generate the complex geometry, **as shown in Figure 1**. The symmetry of the building was taken into account, and one quarter of the building was discretized. The large number elements were created in parallel, and different levels of refinement were given to surfaces and edges to adequately represent the desired geometry, boundary layers, and free shear flows.

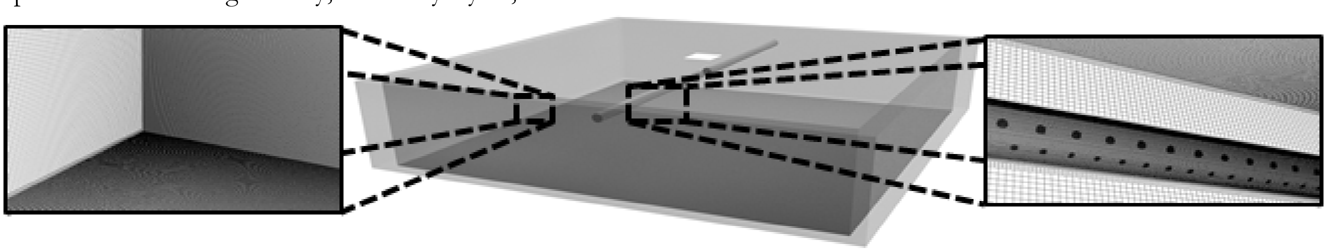

Figure 1: Designed building and selected views of the 12.4 million element discretization used in the CFD simulations.

Solution Procedure

The open source CFD software OpenFOAM [19] was used to solve the coupled set of governing equations. The SIMPLE algorithm was used to link the pressure and velocity equations for the steady state simulations. The linear solvers used for the pressure equation was the Generalized Algebraic Multi Grid (GAMG) while the other linearized equations used a Gauss-Seidel smooth solver. Numerical method used for the divergence terms was a 1st order upwind method. The LaPlacian and gradient terms were solver with a 2nd order Gaussian integration technique. The interpolation scheme from cell center to cell face was linearly interpolation. Non-orthogonal surface normal gradients were corrected. The steady state simulations were solved to a residual tolerance of at least 1e-04 for all the solution variables.

Parallelization and High Performance Computing

The domain was discretized into roughly 1.1 million, 3.2 million, 6.9 million and 12.4 million elements to check for numerical convergence. For each element, the solution variables x-velocity, y-velocity, z-velocity, density, temperature, turbulent kinetic energy, and turbulent dissipation were computed. The number of degrees of freedom (DOF) in the convergence study spanned 7.7 million – 86.8 million. The high performance computing (HPC) cluster at National Institute of Computational Sciences (NICS) was used to solve the large simulations. Each simulation used 12,000 computational hours and 48 hours of wall time on 256 processors.

Verification and Validation

We perform both verification of the numerical methods and validation of the CDF model to ensure the accuracy of our simulations in three steps. First, the CFD model is checked with experimental results to explicitly show that the model accurately reproduces turbulent jet characteristics in the near and far field regions of the jet. Second, we argue that the RNG k – ε model has been successfully applied to accurately model the physics responsible for buoyant based flows and heat transfer in interior environments. Third, we perform a comprehensive convergence analysis at four different mesh densities/resolutions that ensure accuracy of the CFD results.

1. The turbulent jets may occupy a small portion of the domain and away from the occupied zone, but the jets bring in most of the turbulent kinetic energy into the domain. Our previous work has studied the evaluation of RANS models in the near, intermediate, and far wake regions of turbulent jets. The experiments carried out by our previous publications with Liu et al. [7] and Feng [8] et al. investigate the performance of the k – ε model against PIV and PLIF experiments for the propagation of turbulent jets. The results from Liu et al. [7] and Feng et al. [8], show that the k – ε model properly predicts and propagates the streamwise velocity, turbulent kinetic energy and turbulent dissipation of a turbulent jet.
2. Some of the first applications of the RNG k – ε model to interior air flow and ventilation was presented by Linden in 1990 [9]. In this paper, three benchmark problems for buoyancy dominant flow are presented along with comparisons showing that the RNG k – ε model out performs the Standard k – ε model. This comparison is also iterated by Chen [10] a few years later, and more recently by Zhang et al. [11]. Zhang et al. also shows that the RNG k – ε model performs very well in approximating the mean velocities and temperature fields amongst the other RANS models, Large Eddy Simulations (LES), and Detached Eddy Simulations (DES) for both natural and forced convection in interior spaces [11]. H. Sun et al. has also recently compared the accuracy of the RNG k – ε model to PIV data of non-isothermal turbulent flow in a full-scale test room [12]. The results of the comparison concluded that the RNG k – ε model performs fairly well by accurately describing the air flow characteristics in 70% of the space, and also recommends using the RNG k – ε model for Reynolds numbers near 4000 and above [12].
3. A strict mesh dependent convergence analysis was performed based on the four discretizations discussed above. The convergence analysis was performed at the largest flow rate, with ensured that the discretization for lower flow rates were also converged. Table 1 shows the relative error associated with three quantities of interest. The error is reported in terms of the relative L_1 norm $\left(L_1(\boldsymbol{x}) = \sum_{i=1}^{n} \left|\boldsymbol{x}_i - \boldsymbol{x}_i^{(ideal)}\right| / \sum_{i=1}^{n} \left|\boldsymbol{x}_i^{(ideal)}\right|\right)$ where $\boldsymbol{x}$ is a quantity of interest: Air Distribution Performance Index (ADPI), mean temperature of the occupied zone, and mean air speed of the occupied zone. Table 1 shows that for the simulations the largest error is less than 1%.

Quantification of Comfort and Efficiency

In order to quantify the performance of the ductwork system, a set of metrics are utilized to represent the notion of both comfort and efficiency. The comfort metrics used are ADPI, mean temperature in the occupied zone, and mean air speed in the occupied zone. The efficiency metric used in this study is the energy is energy entering through the building's envelope compared to the cooling capacity needed by the HVAC system to return the air at the outlet back to the inlet temperature. The energy entering the building through the envelope is dependent on the insulation in the boundaries, surface area of the boundaries and the temperature difference between the outside and inside.

$$Q_{env} = A_s(T_{ext} - \varphi_T)/R_{eff} \tag{1}$$

The cooling capacity needed by the HVAC system is dependent on the air flow rate and the temperature difference between the inlet and a discrete area weighted sum of the temperature at the outlet.

$$E_{cc} = \rho c_p \dot{V}\left(\frac{1}{A_{out}}\sum_{i=1}^{n_o} T_i A_i - T_{inlet}\right) \tag{2}$$

The ratio of these two quantities, Q_{env}/E_{cc}, gives the input/output relationship on how energy is entering and leaving the building system. The ratio also gives insight to whether at the operating air flow rate is the HVAC system supplying enough cooling to overcome the load created by building's envelope.

Table 1. Convergence analysis and error quantification

Degrees of Freedom	Error in Air Distribution Performance Index	Error in Mean Temperature In Occupied Zone	Error in Mean Air Speed In Occupied Zone
7,730,352	1.30e-01	2.20e-05	1.16e-01
22,837,360	6.33e-02	7.43e-05	4.06e-02
48,554,513	4.99e-03	5.88e-05	9.90e-03

Visualization of Results

Using three dimensional CFD simulations to analyze the air flow characteristics result in extremely large data sets. A single simulation for this study resulted in a data file of 12 GB! Such large data sets are difficult to manipulate and visualize because of limited virtual memory on local machines. Often a single solution variable with cell center and connectivity information cannot be loaded onto a single processor or will crash the virtual memory requirements of commercial data visualization software. After loading the data, color contoured slices, vectors, streamlines, iso-contour surfaces, and any other visualization technique requires additional memory on top of system requirements.

We overcame these challenges with the following strategies. First, the data was downsampled by interpolating the results onto a coarser grid (this was only used for visualization purposes). Next, CFD simulations were decomposed into processors sections and were analyzed separately. Then the iso-contour surfaces, vector plots, and contoured slices from all of the decomposed sections of the domain were stitched into a single lower memory file. Finally, parallel visualization tools were used, such as ParaView [20], to utilize the entire memory capabilities of a local machine or to run a script/macro on a computer cluster that loads the data onto a specified number of computing nodes that have the memory requirements for the large data set.

RESULTS AND DISCUSSION

The results section is broken into two parts. The first section discusses the qualitative flow characteristics and structures seen in the CFD simulations. The second section uses the quantitative metrics discussed above to evaluate the system's performance in terms of comfort and efficiency. In the current study, a total of 15 simulations were completed. Five different air flow rates (2000, 4000, 6000, 8000, and 10000 [CFM] or 0.94, 1.89, 2.83, 3.87, 4.72 [m^3/s]) to determine the ideal flow rate necessary to overcome the thermal loads brought into the building by the building's envelope and keeping a comfortable space. The same air flow rates were then repeated for different effective R-values (11.6, 23.2, and 46.3 which correspond to 1/2, 1 and 2 times the ASHRAE standard for R values for buildings) of the building's envelope.

Air flow patterns are extremely important in indoor air quality and the removal of heat from the conditioned space.

Fresh air needs to be continually replaced in the occupied zone. Two different air flow structures were found in simulations. The first structure is observed at the higher flow rates of 10000 - 6000 [CFM] (4.72 – 2.83 [m^3/s]) and for all R-values. Air first enters the space and moves downward and away from the linear diffuser. This behavior creates a recirculation zone directly under the linear dispersion ductwork system, **as shown in Figure 2**. This behavior disappears at the lower flow rates 4000 and 2000 [CFM] (1.89 and 0.94 [m^3/s]) and for all R-values. The air entering the space immediately drops down to the floor before continuing along the floor. This characteristic may be more desirable in terms of indoor air quality, since pollutants could potentially get trapped in the recirculation zone created by the higher flow rates.

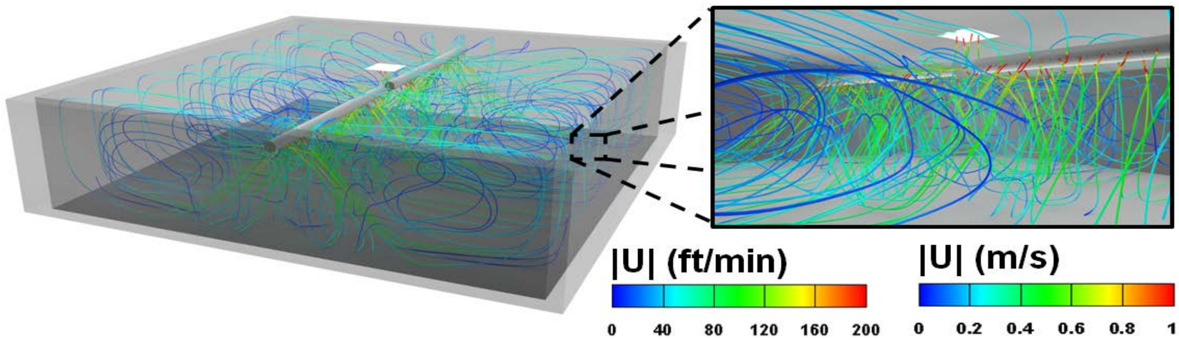

Figure 2: Steady state stream lines contoured with air speed at 10,000 [CFM] (4.72 [m^3/s]).

By placing the ductwork at the center of the space, the cooler fresh air is able to first drop down into the occupied zone and move along the floor to the walls of the building. When the air reaches the walls the air moves up the wall, aided by buoyancy, to the ceiling. This strategy has several benefits. Cold air is able to condition the occupied zone first before mixing with the warmer air at the walls and the ceiling of the building. The ventilation system is able to create a fairly uniform temperature in the occupied zone while the walls and the ceiling experience higher thermal gradients, **as shown in Figure 3**. The movement of the air is aided by the natural buoyant forces that exists in near the boundaries and keeps the warmer air up out of the occupied zone.

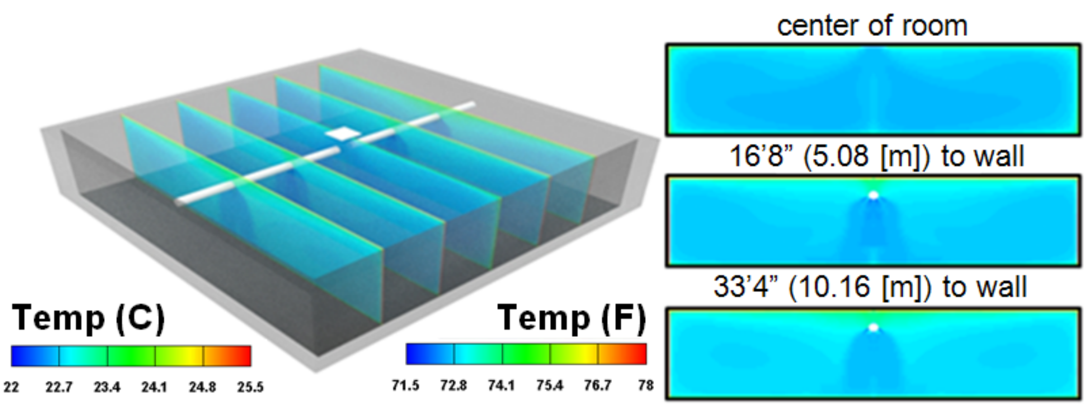

Figure 3: Steady state slices of the building's temperature profile at 10,000 [CFM] (4.72 [m^3/s]).

Further evaluation of the comfort of the occupied zone was done by investigating how the ADPI, mean temperature, and mean air speed changes for the different flow rates and effective R-values. ADPI for the building drastically changes as a function of air flow rate, and also shows some sensitivity to how well the building is insulated, **as shown in Figure 4a**. Generally at low flow rates the building is not very comfortable, and at higher flow rates the ADPI seems to decrease. **Figure 4a** shows that the nonlinearity of the response of the ADPI curve to air flow rate decreases as the effective R-value is decreased. The peak of the ADPI curve increases and shifts to lower flow rates as the effective R-value increases. Increasing the insulation on the building will then decrease the power needed by the fan in the HVAC system, but makes the comfort of the space highly sensitive to air flow rate.

In order to determine the cause of the drastic changes in the ADPI curves, the parameters of the draft temperature are investigated. The draft temperature is a function of the local temperature difference from the mean temperature of the occupied zone and the local air speed difference from 30 [ft/min] (0.15 [m/s]). The mean temperature of the occupied zone is more sensitive to changes in the building's insulation than the mean air speed of the occupied zone, **as shown in Figure 4b**. The tightly grouped mean air speed curve could be due to the large circulation zone, **as shown in Figure 2**. A large amount of energy is contained in the large circulation zone, and changes in the insulation of the building do not change the air flow structures in the building. The shifts in the increase of the ADPI curve are a response to the change in the mean temperature of the space.

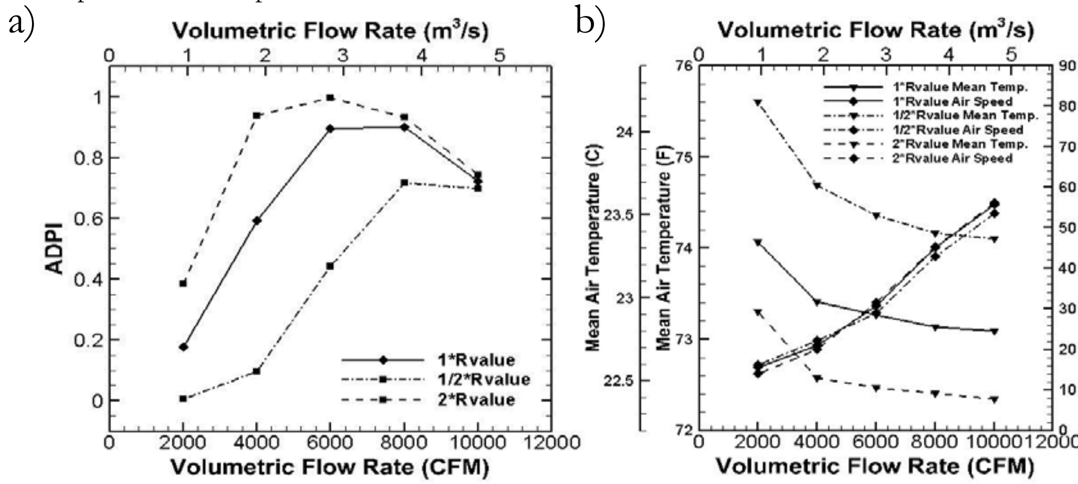

Figure 4: (a) ADPI of occupied zone for multiple flow rates and effective R-vales. (b) Mean temperature and mean air speed in occupied zone for multiple flow rates and effective R-values.

In terms of efficiency the performance of the ductwork system is evaluated by the amount of the energy entering through the building's envelope that the HVAC system can remove at a given flow rate. The critical operating flow rate is determined when the ratio $Q_{evn}/E_{cc} = 1$. At this critical point, the cooling capacity needed by the HVAC system is equal to the heating of the space by the building's envelope. The critical operating point decreases as the effective R-vale of the building increases, **as shown in Figure 5**. By doubling the insulation from the recommended R-values in ASHRAE 90.1, the cooling capacity by the HVAC system is cut by 25.4%-24.7% for the range of flow rates investigated. If the building is older, and is poorly insulated with R-values half of the current recommended R-values, retrofitting the building could cut the cooling capacity seen by the HVAC system by 26%-30%.

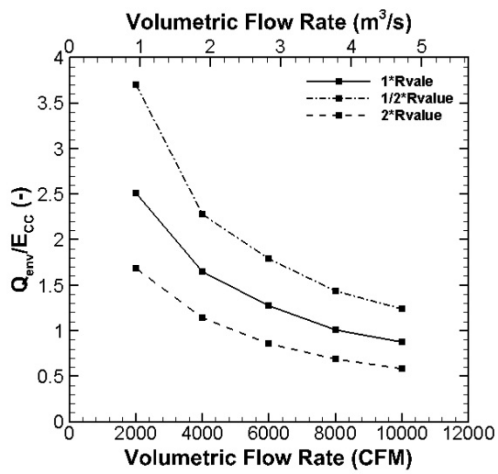

Figure 5: Ratio of the energy entering through the building's envelope and the cooling capacity.

CONCLUSION

We have used high resolution CFD simulations to investigate the performance and sensitivity of a linear dispersion ductwork system in a 10,000 [ft²] (929 [m²]) building. We have shown two different air flow structures exist for the linear dispersion system. The ADPI is extremely sensitive to air flow rate and effective insulation in the building's envelope. The shift of the ADPI curves is due to the shift in mean temperature of the occupied zone. We have established a calculation method for determining the air flow rate necessary remove the energy entering the through the building's envelope. Finally we have discovered that by doubling the insulation on the building gives roughly a 25%-30% savings in cooling capacity seen by the HVAC system.

NOMENCLATURE

ρ	=	Density of air
φ_T	=	Target air temperature
A_i	=	Area of face i attached to outlet
A_s	=	Surface area of domain
A_{out}	=	Area of the outlet
c_p	=	Specific heat of air
E_{cc}	=	Cooling capacity of HVAC system
n_o	=	Number of elements attached to the outlet
n	=	Number of elements in the occupied zone
Q_{env}	=	Energy entering through envelope
R_{eff}	=	Effective R-value of building envelope
T	=	Temperature of air
T_{inlet}	=	Inlet air temperature
T_{ext}	=	Temperature of outside air
$\dot{V}$	=	Volumetric air flow rate

REFERENCES

[1] U.S. Department of Energy. 2011. Buildings Energy Data Book.

[2] Fontanini, A., Olsen, M., and B. Ganapathysubramanian. 2011. Thermal comparison between ceiling diffusers and fabric ductwork diffusers for green buildings. Energy and Buildings. 43: 2973-2987.

[3] Chen, F., Chen, H., Xie, J., Shu, Z., and J. Mao, 2011. Air distribution in room ventilated by fabric air dispersion system. Building and Environment. 46: 2121-2129.

[4] Li, H., Huang, C., Wang, X., Zou, Z., and F. Wang. 2011. Experimental Study on the Thermal Environment of Fabric Duct System Application in a Large Space. ICETCE Conference. 1258-1261.

[5] Nielsen, P.V., Topp, C., Sonnichsen, M., and H. Andersen. 2005. Air distribution in rooms generated by a textile thermal – comparison of mixing and displacement ventilation. ASHRAE Transactions. 733-739.

[6] ASHRAE 90.1. 2007. Energy Standard for Buildings, Except Low-Rise Residential Buildings. Atlanta: American Society of Heating Refrigeration and Air Conditioning Engineers, Inc.

[7] Liu, Y., Feng, H., Olsen, M., Fox, R., and J. Hill. 2006. Turbulent mixing in a confined rectangular wake, Chemical Engineering Science. 61: 6946-6962.

[8] Feng, H., Olsen, M., Liu, Y., Fox, R., and J. Hill. 2005. Investigation of turbulent mixing in a confined planar-jet reactor, AIChE Journal. 51: 2649-2664.

[9] Linden, P.F., Lane-Serff, G.F., and D.A. Smeed. 1990. Emptying filling boxes: the fluid mechanics of natural ventilation. Journal of Fluild Mechanics. 212: 309-335.

[10] Chen. Q. 1995. Comparison of different k-ε models for indoor air flow computations. Numerical Heat Transfer. 28: 353-369.

[11] Zhang, Z., Zhang, W., Zhai, Z.J., and Q.Y. Chen. (2007). Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 2-Comparison with Experimental Data from Literature. HVAC&R Research. 13: 871-886.

[12] Sun. H., Zhao, L., and Y. Zhang. 2007. Evaluating RNG k-ε models and LES an non-isothermal models for indoor airflow using PIV measurement data. ASABE Transactions. 50: 621-631.

[13] Pope, S.B. 2008. Turbulent Flows. Boston: Cambridge University Press.

[14] Tannehill, J.C., Anderson, D.A., R.H. Pletcher. 1997. Computational Fluid Mechanics and Heat Transfer. Philadelphia: Taylor & Francis.

[15] Papadopoulos, G., and W.M. Pitts. 1999. A Generic Centerline Velocity Decay Curve for Initially Turbulent Axisymmetric Jets. Transactions of the ASME. 121: 80-85.

[16] Walton, G.N. 1983. Thermal Analysis Research Program Reference Manual.

[17] EnergyPlus. 1996. Engineering Reference. Ernest Orlando Lawrence Berkeley National Laboratory, US Department of Energy (DOE).

[18] Marion, W., and S. Wilcox. 1995. Solar Radiation Data Manual for Buildings. National Renewable Energy Laboratory, US Department of Energy (DOE).

[19] OpenFOAM webpage [online]. 2013. Available: http://www.openfoam.com.

[20] ParaView webpage [online]. 2013. http://www.paraview.org.