View metadata, citation and similar papers at core.ac.uk

brought to you by CORE

provided by Purdue E-Pubs

Purdue University
Purdue e-Pubs

International High Performance Buildings Conference

School of Mechanical Engineering

2010

Large Eddy Simulation of Airflow in a Single Family House

Rashmin Damle
CTTC - UPC

Manel Soria
CTTC - UPC

Oriol Lehmkuhl
Termo Fluids

Guillem Colomer
Termo Fluids

Assensi Oliva
CTTC - UPC

Follow this and additional works at: http://docs.lib.purdue.edu/ihpbc

Damle, Rashmin; Soria, Manel; Lehmkuhl, Oriol; Colomer, Guillem; and Oliva, Assensi, "Large Eddy Simulation of Airflow in a Single Family House" (2010). *International High Performance Buildings Conference.* Paper 31.
http://docs.lib.purdue.edu/ihpbc/31

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at https://engineering.purdue.edu/Herrick/Events/orderlit.html

Large Eddy Simulation of Airflow in a Single Family House

Rashmin DAMLE[1], Manel, SORIA[1], Oriol LEHMKUHL[1,2], Guillem COLOMER[2], Assensi OLIVA[1]

[1]Centre Tecnològic de Transferència de Calor (CTTC),
Universitat Politècnica de Cayalunya (UPC),
Colon 11, 08222 Terrassa (Barcelona), Spain
Tel. +34-93-739.81.92; FAX: +34-93-739.89.20
E-mail: cttc@cttc.upc.edu http://www.cttc.upc.edu

[2]TermoFluids, S.L.
Magí Colet 8, 08204 Sabadell (Barcelona), Spain
E-mail: termofluids@termofluids.com

ABSTRACT

This work is addressed to contribute to the progress in the determination of indoor air flow in buildings. The air flow in such environments is caused by natural convection, stack and wind effects, infiltration of ambient air and mechanical ventilation. It is important to be able to predict the pressure, temperature and velocity distributions in order to maintain an adequate indoor environment and to minimize the energy demand at the same time. The problem is difficult due to the large and complex geometry involved, the changing boundary conditions, the mixture of free and forced convection and especially, because the flows are turbulent. The purpose of this work is to explore the feasibility of the use of Large Eddy Simulation (LES) models to carry out full scale simulations of airflow in buildings. A review of the previous works shows that the use of LES for this kind of problems has been restricted to idealized geometries and relatively low Re numbers. In this work, for code validation, a 3D cavity with mixed convection is simulated and the numerical results are compared with the experimental data of Blay, et al. (1992). Then, a single family house with a realistic geometry, Re number of 11834 and Rayleigh number of 2.4 x 10^{10} is solved using a Yoshizawa Smagorinsky LES model (Yoshizawa, et al. 2000). The code TermoFluids (Lehmkuhl, et al. 2007) has been used for the simulation and post-processing of the results.

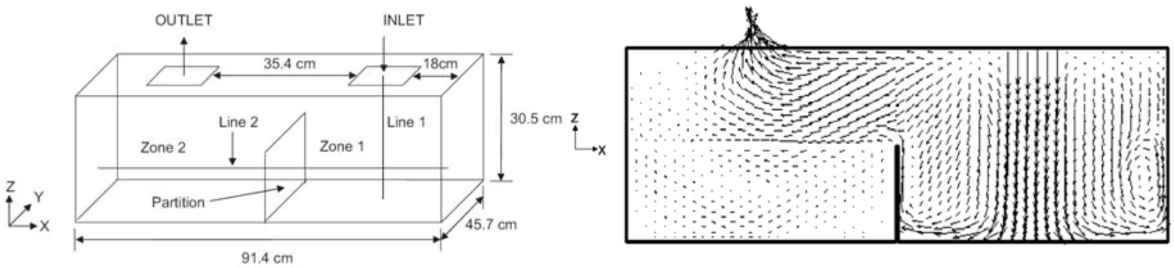
1. INTRODUCTION

As people spend 90% of their time in artificial environments (e.g. houses, offices, transport vehicles, etc.) (Awbi, 1991), it is important to have healthy and fresh indoors. Inadequate outdoor air supply results in the so called ``sick building syndrome" and other health problems caused by pollutants like odour and water vapour, concentrations of tobacco smoke, formaldehyde, particulate matter and radon (Awbi, 1991). Natural infiltration of air is not always sufficient to maintain the indoor air quality, and an excessively leaky or open building leads to additional and unnecessary energy demand that is required for conditioning the infiltrated air. Buildings represent 20-40 % of the total energy consumption ín developed countries (Pérez-Lombard, et al. 2008), and the energy needed for heating ventilating and air-conditioning (HVAC) applications can be around 50% of the energy consumed by the buildings. Thus, the potential for energy savings is significant. Therefore, controlled ventilation is necessary to provide thermal comfort, removal of pollutants, acceptable air quality and optimizing the energy demands. This is an important problem in different contexts, such as: (i) the design of energy efficient buildings based on a good passive behaviour, where it is important to know the air flow patterns due to external pressure differences, or the air circulation due to temperature differences in systems with natural convection; (ii) the air flow in buildings where mechanical ventilation devices are used, and where it is important to understand the air flow distribution, to find and avoid recirculations or areas where the velocity is too large. In both cases, it is important to be able to predict the pressure, temperature and velocity distributions. The problem is difficult due to the large and complex geometry involved, the changing boundary conditions (such as the solar radiation heat flux), the mixture of free and forced convection and specially, because the flows are turbulent. Airflow simulation models should provide information such as the fresh air supply needed, the suitable position of the air supplying devices, optimum air supply with temperature and velocity, the airflow patterns and the distribution of air velocities, contaminant concentrations, temperature and turbulence intensities in the room.

Many authors have studied the airflow in buildings due to natural and forced ventilation. A comprehensive study of multizone airflow modelling has been presented by Axley (2007). Chen (2009) has summarized the different methods for the ventilation performance for buildings like analytical models, empirical models, experimental models and numerical simulation models. Some of the well known models are: i) LBL model (ASHRAE, 1989) for single zone buildings. This model does not consider mechanical ventilation and calculates the infiltration taking in to account the wind and the stack effects depending on the leakage area of the building, wind speed and the ambient temperature; ii) CONTAM (Walton, 1977) is a multizone model which has been used to calculate the indoor air quality in buildings; iii) AIRGLAZE (Voeltzel, et al. 2001) is a model for modelling large highly-glazed spaces; iv) COMIS (Feustel, 1999) is a multizone model prepared by a group of experts of the IEA (International Energy Agency). Each zone is assumed to have a single pressure and temperature that are used to calculate the air flow between the zones. Flows due to large vertical openings, thermal buoyancy, single sided ventilation, fans, etc. are modelled with different expressions; v) COwZ (Stewart, et al. 2006) is an extension of the COMIS model that divides the zones into sub-zones. The mass flow between subzones is calculated depending on whether a subzone includes fluid jet, thermal buoyancy inducing elements like a heater, boundary layers, etc. Here also the room air volume is divided further into control volumes for better estimation of temperatures. Thus, more information of the room airflow distribution and temperatures is available.

Unlike the models described before which depend on experimental data, CFD (Computational Fluid Dynamics) models are expected to predict the room flow with greater accuracy. Some authors have used RANS (Reynolds Averaged Navier Stokes) for airflow movement. These works include particle dispersion in a ventilated room with objects inside with a zero equation model by Zhao and Guan (2007), cross and single side ventilation with a k-ε model by Visagavel, et al. (2009), numerical simulation models for the calculation of heat transfer coefficients by Blocken, et al. (2009), and CFD analysis of the transient flow development in a naturally ventilated room with a k-ε model by Kaye, et al. (2009). Also, authors like Stamou and Katsiris (2006), Kuznik, et al. (2007), Evola and Popov (2006), have simulated airflow using RANS for simple room configurations.

Other authors have used LES for simulating the airflow in indoor environments. Davidson and Nielson (1998) indicate problems with the RANS models and simulated Reynolds number up to 5000 (mesh: 1,310,720 control volumes) with a LES model for a geometry similar to a backward facing step. This configuration was chosen as it is similar to a ventilated room with the opposite wall removed. Zhang and Chen (2000) simulated natural convection (rectangular cavity with aspect ratio 5 with Ra=5 x 10^{10}, mesh: 46,128 control volumes), forced convection (Re=5000, mesh: 23,552 control volumes) and mixed convection (Ar=0.036, Re=678, mesh: 46,128 control volumes) for a simple parallelopiped geometry with a large eddy simulation (LES). Tian, et al. (2007) modelled forced convection flows with Re=1900 and Re=4830 with mesh of 3,96,720 control volumes for studying the dispersion of particles in the room interior for a rectangular geometry of dimensions 91.4cm x 45.7cm x 30.5cm, with a inlet and a outlet with two zones as shown in Figure 1. Cheng-Hu, et al. (2008) modelled unsteady cross ventilation using LES. The results agreed reasonably well with the experimental data in the above cases.



(a) (b)

Figure 1: Case simulated by Tian, et al. (2007): a) geometry b) velocity vectors.

In this work, the feasibility of LES in building applications is assessed for the typical Rayleigh and Reynolds numbers that exist in dwellings in normal working conditions. For the validation of the code TermoFluids (Lehmkuhl, et al. 2007), in the context of building ventilation problems, a 3D cavity with mixed convection (Ra=2.4 x 10^9 Ar=0.036) is simulated with the Yoshizawa Smagorinsky LES model (Yoshizawa, et al. 2000) and the numerical results are compared with experimental data of Blay, et al. (1992). Then the airflow in a single family house with real dimensions, unlike the simple geometries mentioned earlier, is simulated using TermoFluids (Lehmkuhl, et al. 2007).

2. TYPICAL RAYLEIGH AND REYNOLDS NUMBERS IN BUILDINGS

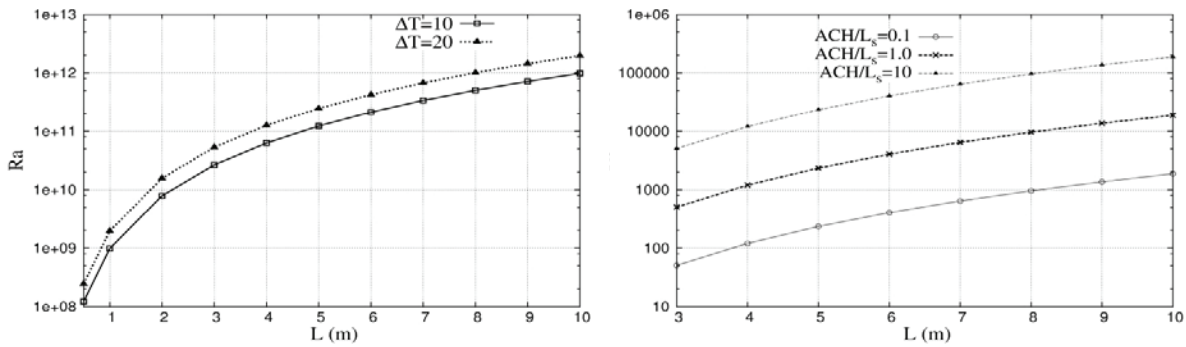
In building applications the heat transfer and fluid flow is caused by natural convection due to density differences, stack and wind effects, infiltration of outdoor air and artificial ventilation. The level of turbulence for natural convection and forced convection is governed by the Rayleigh number and the Reynolds number respectively. The Rayleigh number is given by the equation (1).

$$\text{Ra}=\frac{\rho g\beta\Delta T L^3}{\alpha\mu} \qquad (1)$$

The characteristic length in case of buildings is the height of the rooms L. The curves for Rayleigh number variation with L for temperature differences of 10°C and 20°C are shown in the Figure 2a. Respect to the Reynolds number, if a building is assumed to be an enclosure with a total volume of L^3 with one or more rooms in the enclosure connected to each other and that the air renovation in these rooms is ACH (Air Changes per Hour) times every hour through an aperture with a characteristic length of L_s, the Reynolds number is given by the equation (2) as:

$$\text{Re}=\frac{1}{3600}\frac{\rho}{\mu}\frac{ACH}{L_s}L^3 \qquad (2)$$

Figure 2b shows the values of the Reynolds number for different values of the fraction ACH/L_s between 0.1 $h^{-1}m^{-1}$ and 10 $h^{-1}m^{-1}$. Moreover, in multizone buildings there is a Rayleigh and a Reynolds number for each of the zones, and there exists a coupling between the fluid flow and the heat transfer from solid parts including thermal radiation. This makes the problem even more complicated and although Direct Numerical Simulations (DNS) for natural convection for Rayleigh numbers up to 10^{10} have been simulated (Trias, et al. 2010), the possibility of DNS in actual buildings is not feasible. However LES can be used in certain conditions in ventilation problems as will be seen in section 5 where a case with the Re=11834, Ra=2.4 x 10^{10} is solved.



(a) (b)

Figure 2: Rayleigh and Reynolds numbers in building applications

3. MATHEMATICAL MODEL

The Navier-Stokes equations are written for all the control volumes of the domain as:

$$M\boldsymbol{u}=0 \qquad (3a)$$

$$\frac{\partial\boldsymbol{u}}{\partial t}=-C(\boldsymbol{u})\boldsymbol{u}+\nu D\boldsymbol{u}-\rho^{-1}G\boldsymbol{p}+\boldsymbol{f} \qquad (3b)$$

$$\frac{\partial\boldsymbol{T}}{\partial t}=-C(\boldsymbol{u})\boldsymbol{T}+\lambda\rho^{-1}c_p^{-1}D\boldsymbol{T} \qquad (3c)$$

where vector velocity, pressure, temperature and buoyancy terms have a matrix form, $\boldsymbol{u}\in\mathbb{R}^{3N}$, $\boldsymbol{p}\in\mathbb{R}^{N}$, $\boldsymbol{T}\in\mathbb{R}^{N}$ and $\boldsymbol{f}=\beta(\boldsymbol{T}-T_o)\boldsymbol{g}\in\mathbb{R}^{3N}$ respectively. Convective and diffusive operators in the momentum equation for the velocity vector field are given by $C(\boldsymbol{u})\in\mathbb{R}^{3N\times 3N}$ and $D\in\mathbb{R}^{3N\times 3N}$ respectively. Similarly, in the energy equation, and considering the scalar temperature field, $C(\boldsymbol{u}) \in \mathbb{R}^{N\times N}$ and $D \in \mathbb{R}^{N\times N}$. Gradient and divergence operators are given by $G \in \mathbb{R}^{3N\times N}$ and $M \in \mathbb{R}^{N\times 3N}$ respectively. For the temporal discretization of the Navier-Stokes equation (3), a third-order Gear-like scheme (Fishpool, et al. 2009) is used for the derivative, convective and diffusive term. And for the pressure gradient, a first-order backward Euler scheme is used. Our spatial discretization schemes are conservative. These conservation properties are held if and only if the discrete convective operator is skew-symmetric ($C_c(\boldsymbol{u_c}) = -C_c^*(\boldsymbol{u_c})$), if the negative conjugate transpose of the discrete gradient operator is exactly equal to the divergence operator ($-\Omega_c G_c^* = M_c$), and if the diffusive operator D_c is symmetric and positive-definite. As Felten et al. (2006) had noted, the total contribution of the pressure gradient term to the evolution of the kinetic energy does not vanish when the fractional step method is used on a collocated arrangement,

$$\epsilon_{\mathbf{ke}} = (\tilde{p}_c)^* M_c (G_c - G_s)\, \tilde{p}_c \qquad (4)$$

where $\tilde{p} = \Delta t\, p$, G_s and G_c are gradient operators at the faces and at the cell centers, respectively. Since this term cannot be eliminated it is of interest to evaluate its scaling order. Felten, et al. (2006) had perform an analytical analysis of this term, and from their discussion, it is clear that the spatial term of the pressure error scales like $O(\Delta_x^2)$ and the temporal term scales like $O(\Delta_t)$. However, more recently Fishpool, et al. (2009), developed a third-order Gear-like temporal scheme for collocated meshes that allows both a reduction of the dissipation of this type of schemes and the increase of the numerical stability. This strategy is the one adopted in this work.

The number of control volumes required for resolving the Navier-Stokes equations and simulating all the scales of motion in turbulent flows is proportional to $Re^{9/4}$ and the computational cost to Re^3 (Piomelli, 1999). For this reason the Direct Numerical Simulation (DNS) is limited to simple geometries. Large eddy simulation (LES) on the other hand involves solving the large scales corresponding to the mesh size and modelling all the scales which are smaller than the mesh size. This is done by filtering Navier-Stokes equations in space which makes LES an unsteady computation unlike RANS (Reynolds Averaged Navier Stokes). The technique of modelling only the small eddies is justified in case of high Reynolds number flows where the dissipative eddies can be assumed to be homogeneous and universal. Moreover, as the mesh size increases the LES simulations approaches the DNS.

3.1 SMARGORINSKY LES MODEL

To obtain the governing equations for larger scales the Navier Stokes equations can be filtered spatially in LES. Doing so, the discrete filtered non-linear convective term must be modelled,

$$\boldsymbol{\Omega_c}\frac{\partial \overline{\boldsymbol{u}}_c}{\partial t} + C_c(\overline{\boldsymbol{u}}_c)\overline{\boldsymbol{u}}_c - \nu D_c \overline{\boldsymbol{u}}_c + \rho^{-1}\boldsymbol{\Omega_c} G_c \overline{\boldsymbol{p}}_c - \boldsymbol{\Omega_c}\overline{\boldsymbol{f}}_c = C_c(\overline{\boldsymbol{u}}_c)\overline{\boldsymbol{u}}_c - \overline{C_c(\boldsymbol{u}_c)\boldsymbol{u}_c} = -\mathcal{M}_c \boldsymbol{\mathcal{T}_c} \qquad (5)$$

where $\overline{\boldsymbol{u}}_c$ is the filtered velocity, $\mathcal{M}_c$ represents the divergence operator of a tensor, and $\boldsymbol{\mathcal{T}_c}$ is the SGS stress tensor, which is defined as,

$$\boldsymbol{\mathcal{T}_c} \approx -2\nu_{sgs}\overline{\boldsymbol{S}}_c + (\boldsymbol{\mathcal{T}_c}:\boldsymbol{I})\boldsymbol{I}/3 \qquad (6)$$

To close the formulation a suitable expression for the SGS-viscosity ν_{sgs} must be introduced. Smagorinsky supposed that LES eddy viscosity is proportional to a subgrid scale characteristic length $\Delta = c_s\ell$, and to a characteristic subgrid-scale velocity (Yoshizawa, et al. 2000). Then,

$$\nu_{sgs} = (c_s\ell)^2|\overline{\boldsymbol{S}}_c| \qquad (7a)$$

$$|\overline{\boldsymbol{S}}_c| = (2\overline{\boldsymbol{S}}_c:\overline{\boldsymbol{S}}_c)^{1/2} \qquad (7b)$$

$$\overline{\boldsymbol{S}}_c = \frac{1}{2}[G_c\overline{\boldsymbol{u}}_c + G_c^*\overline{\boldsymbol{u}}_c] \qquad (7c)$$

where ℓ is the grid cutting length and c_s is the Smagorinsky coefficient, which may be tuned so that the model reproduces the $\kappa^{-5/3}$ cascade (Pope, 2000) when simulating isotropic decay. As it is well known, the value of this coefficient lies in the range between 0.1 and 0.24, depending on the numerical method used to solve the equations. However, this model is not appropriate in the close vicinity of a solid wall subject to dominant molecular-viscosity effects. Here a non-equilibrium fixed-parameter SGS model derived by Yoshizawa is used which gives the SGS-viscosity as

$$\nu_{sgs} = c_{\nu s}\ell\|\overline{\boldsymbol{u}}_c - \overline{\overline{\boldsymbol{u}}}_c\|\left[1 - \exp[-(c_w\|\overline{\boldsymbol{u}}_c - \overline{\overline{\boldsymbol{u}}}_c\|/(|\overline{\boldsymbol{S}}_c|\ell))^2]\right] \qquad (8)$$

where $\overline{\overline{\boldsymbol{u}}}_c$ is the doubly filtered velocity and $c_{\nu s}$ and c_w are the two model parameters. These parameters are usually chosen as $c_{\nu s} = 0.03$ and $c_w = 21.0$. Of these two, the choice of $c_{\nu s}$ is more relevant to reproducing the logarithmic velocity law. Yoshizawa has shown that the model is not so sensitive to the c_w coefficient. In this model, the equilibrium of SGS fluctuation is not assumed, no use is made of wall-unit distance based on the friction velocity, and the near-wall asymptotic behaviour of the SGS viscosity is fulfilled.

4. MIXED CONVECTION CASE FOR CODE VALIDATION

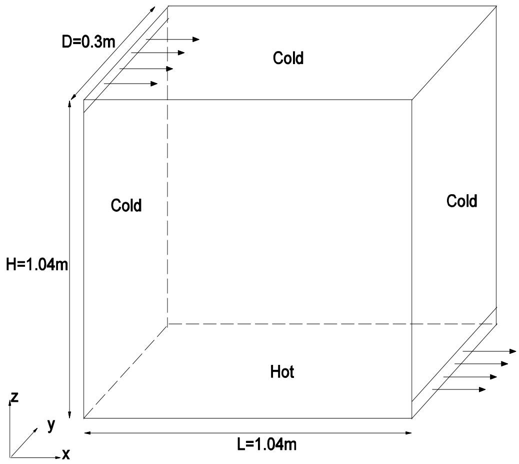


Figure 3: Schematic of the experimental setup.

Mixed convection airflow in a 3D cavity is considered for the validation of TermoFluids (Lehmkuhl, et al. 2007) code so as to use it further in building applications. The schematic of the test case is shown in the Figure 3. This case has been studied by Mergui (1993), Zhang and Chen (2000) and Ezzouhri, et al. (2009). Also, the experimental data for this case has been published by Blay, et al. (1992). The length (L) and the height (H) of the cavity are 1.04m each, and the depth (D) of the cavity is 0.3m. Air enters the cavity top through a slot of height *h*=0.018m along the cavity at 15°C and leaves though a slot of height *l*=0.024m at the bottom of opposite wall. The bottom surface of the cavity is at 35° C while the top and the lateral walls are at 15°C. The front and the rear walls are adiabatic. A parabolic inlet velocity distribution is imposed according to the measured velocity profile (Ezzouhri, et al. 2009).

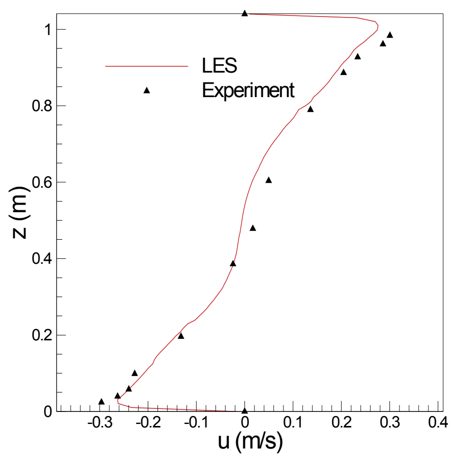


Figure 4a: Mean horizontal velocity (u) with z at y=0.15m and x=0.52m.

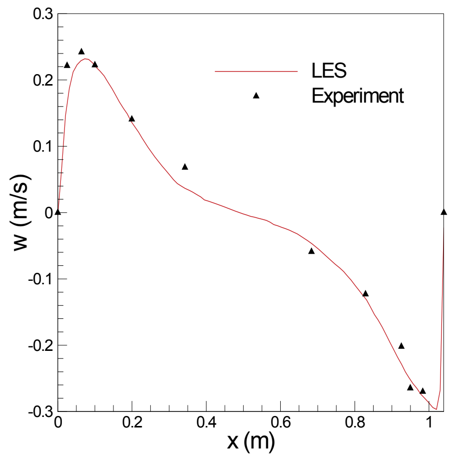


Figure 4b: Mean vertical velocity (w) with x at y=0.15m and z=0.52m

The Rayleigh number for this case with mean inlet velocity U_0=0.57 m/s is 2.4 x 10^9, while the Reynolds number and the Archimedes number ($Ar=\frac{g\beta h\Delta T}{U_0^2}$) are 684 and 0.036 respectively. A mesh with 100x120x32 control volumes has been used for the simulation of this case with 104 processors with an integration time of 500secs. Numerically obtained mean velocities are compared with the experimental data of Blay, et al. (1992) in Figure 4a and Figure 4b. It can be seen that the numerical and experimental values of the horizontal and vertical velocities are in good agreement, and the TermoFluids (Lehmkuhl, et al. 2007) code is able to reproduce the physics of the flow to a good extent.

5. AIRFLOW IN A SINGLE FAMILY HOUSE

An illustrative simulation of airflow in a real house geometry with dimensions 8.3mx13.8mx3.5m, computed with LES, is presented here. Air is introduced from a rectangular inlet in the ceiling of living room through an area of 0.25m^2 with a velocity of 0.4 m/s which corresponds to one air change per hour of the house air volume. The value ACH/L_s in this case is 2 $h^{-1}m^{-1}$. Two windows are kept open to the ambient where a condition of pressure=0 is applied. The house walls are considered adiabatic as the objective here is to simulate the turbulent airflow with the LES model to see the air movement in a real house geometry, much more complex than the studies previously mentioned, and also to show the feasibility of the LES simulation which can give detailed information of the flow within the different zones (rooms) of the house. Further studies will include realistic wall boundary conditions such as walls contributing to natural convection and thermal radiation, air infiltration from ambient, solar radiation over exterior surfaces, etc.

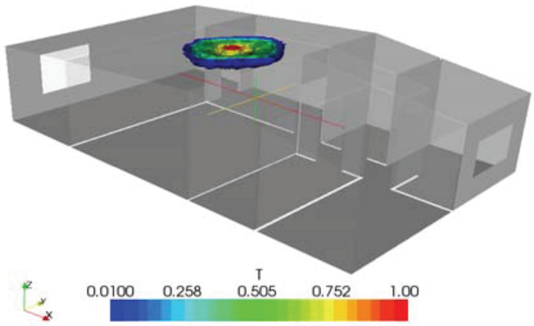


Figure 5a: Temperature contours after 10 seconds

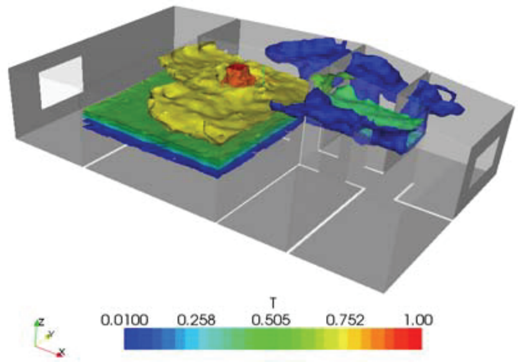


Figure 5b: Temperature contours after 5 minutes

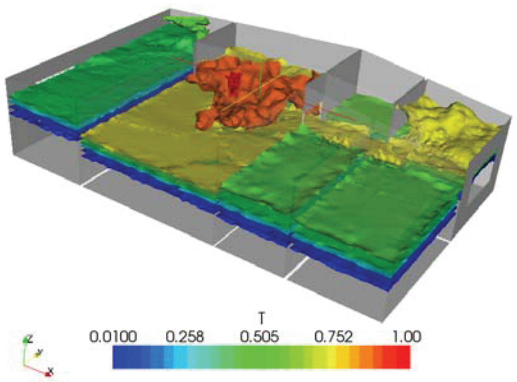


Figure 5c: Temperature contours after 30 minutes

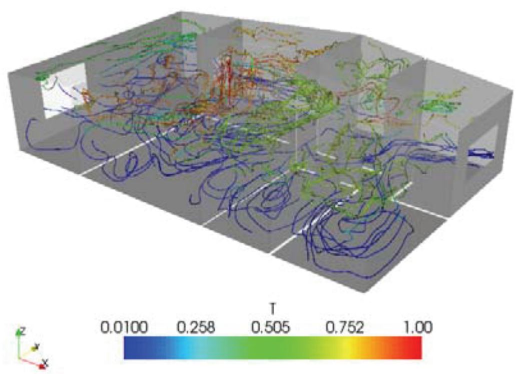


Figure 5d: Streamtracers after 30 minutes

The airflow simulation in the house is done with the code TermoFluids (Lehmkuhl, et al. 2007). An unstructured mesh of 2.12 million control volumes is used and the aforementioned Yoshizawa Smagorinsky model (Yoshizawa, et al. 2000) is applied. The Rayleigh number for this case is 2.7 x 10^{10} and the Reynolds number is 11834. The Archimedes number (Ar) for mixed convection is 0.79. The simulation is started with the entire temperature map in the house set to a dimensionless temperature of zero and the inlet jet enters at a dimensionless temperature of 1. This is done for a better visualization of the flow and its evolution with time into the different rooms. The simulation was run with the aforementioned boundary conditions for a physical time of 30 minutes. With 32 processors, the computational time was about 10 days. Figures 5a-5d show the evolution of the airflow with time. The hot air inlet jet does not come down immediately because of the buoyant forces as in Figure 5a. The hot air flows first through the room with open window on the right as in Figure 5b. The hot air layers slowly descend into the rooms with time and the Figure 5c shows the stratification of the air in the entire house after 30 minutes. The streamtracers in Figure 5d show the complex air circulations in different rooms. In further studies, for reducing the simulation time, the central room having the inlet jet can be modelled with LES, while the other rooms can be modelled by using simplified models.

7. CONCLUSIONS

Large eddy simulations of the air flow are beginning to be feasible for the typical Rayleigh and Reynolds numbers that occur in building applications if enough computational resources and appropriate parallel CFD codes are used. Airflow in a 3D cavity with mixed convection (Rayleigh number =2.4 x 10^9, Reynolds number=684, Archimedes number=0.036) has been simulated with the code TermoFluids (Lehmkuhl, et al. 2007) and the numerical results have been compared with the experimental data available in the literature. Good agreement between the numerical and experimental data has been observed. An illustrative airflow simulation of a real house geometry with Ra=2.4 x 10^{10}, Re=11834 and Ar=0.79 has been solved using the Yoshizawa Smagorinksy model (Yoshizawa, et al.2000) has also been presented.

NOMENCLATURE

Ra	Rayleigh number	Re	Reynolds number
Ar	Archimedes number	ACH	Air changes per hour (1/h)
L	characteristic length (m)	L_s	characteristic length of aperture (m)
ΔT	temperature difference	g	gravitational acceleration 9.81 (m/s^2)
Δ	volume of a control volume (m^3)	c_p	specific heat (J / (Kg K))
−	filtered variable	=	double filtered variable
‖ ‖	euclidean norm	I	identity matrix

Greek symbols

ρ	density (kg/m^3)	μ	viscosity (N-s/m^2)
β	thermal expansion coefficient (1/K)	λ	conductivity (W/(mK))
α	diffusivity (m^2/s)	κ	wavenumber (1/m)
Ω	volume (m^3)	ϵ	error

Superscripts		Subscripts	
*	skew symmetric matrix	c	collocated mesh arrangement
ke	kinetic energy	o	reference quantity

REFERENCES

1. Awbi, H.B., 1991, Ventilation of buildings. E&FN SPON: An imprint of Chapman & Hall. Axley, J., 2007, Multizone airflow modeling in buildings: history and theory. HVAC & R Research, 13, pp. 907– 928.
2. ASHRAE, 1989, Handbook of Fundamentals. Atlanta.
3. Blay, D., Mergui, S., Nicolae, C., 1992, Confined turbulent mixed convection in the presence of a horizontal buoyant wall jet. Fundamentals of Mixed Convection, HTD 213, pp. 65-72.
4. Blocken, B., Defraeye, T., Derome, D., Carmeliet, J., 2009, High-resolution CFD simulations for forced convection heat transfer coefficients at the facade of a low-rise building. Building and Environment, 44(12), pp.2396-2412.
5. Chen, Q., 2009, Ventilation performance of buildings: a method overview and recent applications. Building and Environment, 44, pp. 848–858.
6. Cheng-Hu, H., Ohba, M., Yoshie, R., 2008, CFD modelling of unsteady cross ventilation flow using LES. Journal of Wind engineering and Industrial Aerodynamics, 96, pp. 1692–1706.

7. Davidson, L., Nielson, P. V., 1998, A study of low-Reynolds number effects in backward-facing step using large eddy simulations. ROOMVENT'98 6th Int. Conf. On air distribution in rooms, 1, pp. 125–128.
8. Evola, G., Popov, V., 2006, Computational analysis of wind driven natural ventilation in buildings. Energy and Buildings, 38, pp. 491–501.
9. Ezzouhri, R., Joubert, P., Penot, F., Mergui, S., 2009, Large eddy simulation of turbulent convection in a 3D ventilated cavity: comparison with existing data. International Journal of Thermal Sciences, 48, pp. 2017-2024.
10. Felten, F.N., Lund, T. S., 2006, Kinetic energy conservation issues associated with the collocated mesh scheme for incompressible flow. Journal of Computational Physics, 215, pp. 465–484.
11. Feustel, H.E., 1999, COMIS-an international multizone air-flow and contaminant transport model. Energy and Buildings, 30, pp. 3–18.
12. Fishpool, G.M., Leschziner, M.A., 2009, Stability for fractional-step schemes for the Navier-Stokes equations at high Reynolds number. Computers & Fluids, 38, pp. 1289–1298.
13. Fureby, C., Grinstein, F.F., 1999, Monotonically Integrated large eddy simulation of free shear flows. AIAA Journal, 37, pp. 544-556.
14. Kaye, N.B., Ji, Y., Cook, M.J., 2009, Numerical simulation of transient flow development in an naturally ventilated room. Building and Environment, 44, pp. 889–897.
15. Kuznik, F., Rusaouën, G., Brau, J., 2007, Experimental and numerical study of a full scale ventilated enclosure: Comparison of for two-equation closure turbulence models. Building and Environment, 42, pp. 1043–1053.
16. Lehmkuhl, O., Borrell, R., Pérez-Segarra, C.D., Soria, M., Oliva, A., 2007, A new parallel unstructured CFD code for the simulation of turbulent industrial problems on low cost PC Cluster. Proceedings of PCFD, Turkey.
17. Mergui, S., 1993, Caracterisation experimentale des ecoulements d'air de convection naturelle et mixte dans une cavite fermee. Thèse de l'Universite de Poitiers, France.
18. Pérez-Lombard, L., Ortiz, J., Pout, C., 2008, A review on buildings energy consumption information. Energy and Buildings, 40, pp. 394–398.
19. Piomelli, U., 1999, Large-eddy simulation: achievements and challenges. Progress in Aerospace Sciences, 35, pp. 335–362.
20. Pope, S.B., 2000, Turbulent flows, Cambridge University Press.
21. Stamou, A., Katsiris, I., 2006, Verification of a CFD model for indoor airflow and heat transfer. Building and Environment, 41, pp. 1171–1181.
22. Stewart, J., Ren, Z., 2006, COwZ:-a subzonal indoor airflow, temperature and contaminant dispersion model. Building and Environment, 41, pp. 1631–1648.
23. Tian, Z.F., Tu, J.Y., Yeoh, G.H, Yuen, R.K.K., 2007, Numerical studies of indoor airflow and particle dispersion by large eddy simulation. Building and Environment, 42, pp. 3483–3492.
24. Trias, F.X., Gorobets, A., Soria, M., Oliva, A., 2010, Direct numerical simulation of a differentially heated cavity of aspect ratio 4 with Ra -number up to 10^{11} - Part II: Heat transfer and flow dynamics. International Journal of Heat and Mass Transfer, 53, pp. 674-683.
25. Visagavel, K., Srinivasan, P.S.S., 2009, Analysis of single side ventilated and cross ventilated rooms by varying the width of the window opening. Solar Energy, 83, pp. 2–5.
26. Voeltzel, A., Carrié, F.R., Guarracino, G., 2001, Thermal and ventilation modelling of large highly-glazed spaces. Energy and Buildings, 33, pp. 121–132.
27. Walton, G.N., 1977, Contam 96 users manual. nistir 6055, USA. National Institute of Standard and Technology.
28. Wilcox, D.C., 1998, Turbulence Modeling for CFD. DCW Industries, Inc. CA.
29. Yoshizawa, A., Kobayashi, K., Kobayashi, T., Taniguchi, N., 2000, A non-equilibrium fixed parameter subgrid-scale model obeying the near wall asymptotic constraint. Physics of Fluids, 12(9),pp. 2338
30. Zhang, W., Chen, Q., 2000, Large eddy simulation of indoor air flow with a filtered dynamic subgrid scale Model. International Journal of Heat and Mass transfer, 43, pp. 3219–3231.
31. Zhao, B., Guan, P., 2007, Modelling particle dispersion in personalized ventilated room. Building and Environment, 42, pp.1099–1109.

ACKNOWLEDGEMENT

This work has been partially financed by the Ministerio de Ciencia e Inovación, Secretaría de Estado de Investigación, Spain (ref. ENE 2009-09496).