2012

Parallelization of the coupling between CFD models for airflow and building energy simulation with an object-oriented infrastructure

Rashmin Damle
oriol@termofluids.com

Oriol Lehmkuhl

Joan López

Joaquim Rigola

Assensi Oliva

Damle, Rashmin; Lehmkuhl, Oriol; López, Joan; Rigola, Joaquim; and Oliva, Assensi, "Parallelization of the coupling between CFD models for airflow and building energy simulation with an object-oriented infrastructure" (2012). *International High Performance Buildings Conference.* Paper 73.
http://docs.lib.purdue.edu/ihpbc/73

Parallelization of the coupling between CFD models for airflow and building energy simulation with an object-oriented infrastructure

Rashmin DAMLE[1], Oriol LEHMKUHL[1,2], Joan LOPEZ[2], Joaquim RIGOLA[2], Assensi OLIVA[2]

[1]Termo Fluids S.L,
Magí Colet 8, 08204, Sabadell. Spain.
E-mail: termofluids@termofluids.com

[2]Heat and Mass Transfer Technological Center (CTTC),
ETSEIAT Colom 11, 08223, Terrassa, Spain.
Phone:+34937398192, Fax: +34 93 7398020
E-mail: cttc@cttc.upc.edu

ABSTRACT

Integrating CFD & HT models with the general building program raises the computational time of building simulations as these simulations are usually performed over a period of one year. Within this context, our aim is to couple a object-oriented modular building program with CFD & HT for airflow and parallelize the simulation with numerous processors for reducing computational time. Also the modular nature of the code will allow to resolve selective critical zones with CFD & HT models while employing simple models for airflow in less critical zones. Thus, there are different levels of modelling different rooms/elements of the building system depending on the requirements of a specific case.

1. INTRODUCTION

As people spend a lot of energy and time in buildings, it is important to have energy efficient buildings along with adequate air supply for maintaining acceptable indoor conditions. Airflow in buildings is complicates due to the large and complex geometry involved, changing ambient conditions, heat and moisture flows through the building envelope including heat transfer due to solar and thermal radiation, airflows due to natural convection, stack and wind effects, infiltration of ambient air and mechanical ventilation, and the mixture of free and forced convection flows which are often are turbulent. Building energy simulations can give vital information of the peak loads during the heating and cooling season, room temperatures and velocity distributions for maintaining an adequate indoor environment, and overall energy demands during a year.

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). Some of the well known models are: LBL model (ASHRAE, 1989) for single zone buildings; CONTAM (Walton and Dols, 2005) and COMIS (Feustel, 1999) for multizone modelling; and AIRGLAZE (Voeltzel et al., 2001) for modelling large highly-glazed spaces. COwZ (Stewart and Ren, 2006) is an extension of COMIS (Feustel, 1999) that divides the zones into sub-zones. The mass flow between subzones is calculated taking into account the special subzone cells with fluid jet, thermal buoyancy inducing elements like a heater, or thermal boundary layers. Thus, more information of the room airflow and temperature distribution is available. Chen (2009) has summarized the different methods for the ventilation performance for buildings and suggests that the use of subzonal models is not easy because of the special cells (with boundary layers, jets, etc.) and that they are not much superior to coarse grid CFD & HT simulations.

CFD & HT with building simulations has been addressed by several authors to have more information of the room airflow. One of the earlier attempts of CFD integration with building simulations was presented by Negrao (1998). Musser (2001) and Zhai and Chen (2003) also made studies in this direction. More recently, Wang and Chen (2007) studied three possibilities of coupling methods between CFD & HT and multizone network program. Airflow simulations with LES models by Damle et al., (2011) show that the overall behaviour of room airflow patterns and temperature distributions can be predicted with coarse meshes. So, in that sense the aim of the article is to couple CFD & HT simulations with the general building program NEST for better prediction of the temperature and velocity patterns of zones where the single well-mixed zone assumption is not applicable. Moreover the simulation will be run with numerous processors due to the parallel infrastructure for reducing the computational time. Also, the modular and object-oriented nature of the NEST program will allow the resolution of selective critical zones with CFD & HT while employing simple models for airflow in less important zones. Thus, there are different levels of modelling different rooms/elements of the building system in the same simulation. The modular methodology, global resolution algorithm, parallel framework and preliminary results of an illustrative case of coupling between simple models and CFD &HT models run on numerous processors is presented in this work.

2. MODULAR FRAMEWORK

In this work a building or a structure is modelled as a collection of basic elements (walls, rooms, windows, outdoor, etc.) as shown in the Figure 1a and Figure 1b. A summary of the different elements (objects from C++ programming point of view) which constitute the NEST program are given in Table 1. The terms object and element are used interchangeably in this work.

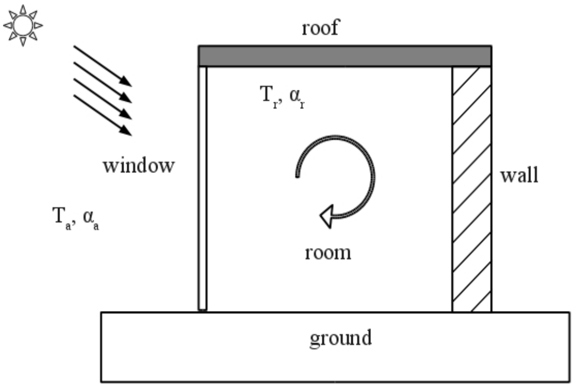

Figure 1a: A simple building.

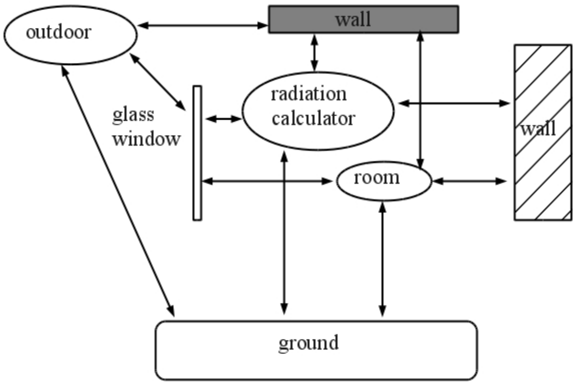

Figure 1b: The building as a group of elements.

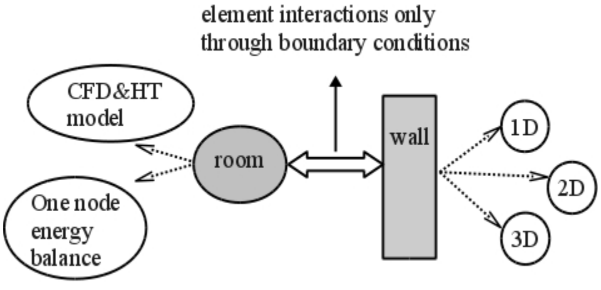

Figure 1c: Flexibility in choosing individual element models.

MPI (Message Passing Interface) is used for the parallelization of the software and a given system is split into partitions (groups of elements) to be run on different processors. These elements/objects are capable of solving themselves when subjected to boundary conditions which are taken from the neighbouring elements. At each iteration, inputs (e.g., pressure, temperature, etc.) are taken from the neighbours, governing equations of the element are solved and the outputs (e.g., pressure, temperature, etc.) are set as boundary conditions for the resolution of the neighbour elements. Iterations continue until convergence is reached at a given time step and then the next time step calculation starts after updating the variables. The global resolution algorithm is shown in Figure 2. The advantage of such a modular approach, as can be seen from Figure 1c, is that each element can be represented in any form as long as it can exchange the necessary boundary information from the rest of the elements in the system. For instance, in a building with many rooms some rooms can be modelled with a detailed CFD & HT calculation while other rooms could be modelled using a global single CV energy balance. Also, new models can be implemented for a given element without changing the entire program.

Table 1. Elements (objects) developed in the NEST program.

Elements	Description
Calendar	Keeps time in two different modes.
Global Position	Stores the geographical position (latitude, longitude, etc.).
Local Position	Stores the orientation and local position of exterior walls.
Outdoor	Read/calculates outdoor data like temperature, wind, solar radiation, etc.
Time function	Retrieves/interpolates values of variables at a given time.
Ground	Resolves three dimensional heat conduction and surface convection.
Wall	Single material layer with one dimensional heat conduction, and heat convection, thermal radiation and solar gains at surfaces.
Composite wall	Multiple material layers with one dimensional heat conduction, and heat convection, thermal radiation and solar gains at surfaces.
Glass wall/window	Glass layer transparent to solar radiation with one dimensional heat conduction, and heat convection and radiation at surfaces.
Room	One node control volume with mass and energy balance.
Radiation calculator	Calculates view factors for thermal radiation calculation.
Solar distributor	Distributes the transmitted solar radiation uniformly to room walls.
Air change calculator	Calculates air infiltration.
HVAC	Controls heating and cooling of rooms.
Opening	Air movement between rooms and outdoor due to pressure differences.
Door	Air movement between rooms connected with large openings
Internal volumes	Volumes with surfaces at a given temperature or with heat flux
Flow network	Calculates the flows across a network of rooms with a Newton-Raphson method by taking into account the mass residuals of the entire network.
CFD object	Detailed three dimensional resolution of fluid flow and heat transfer with or without turbulence models

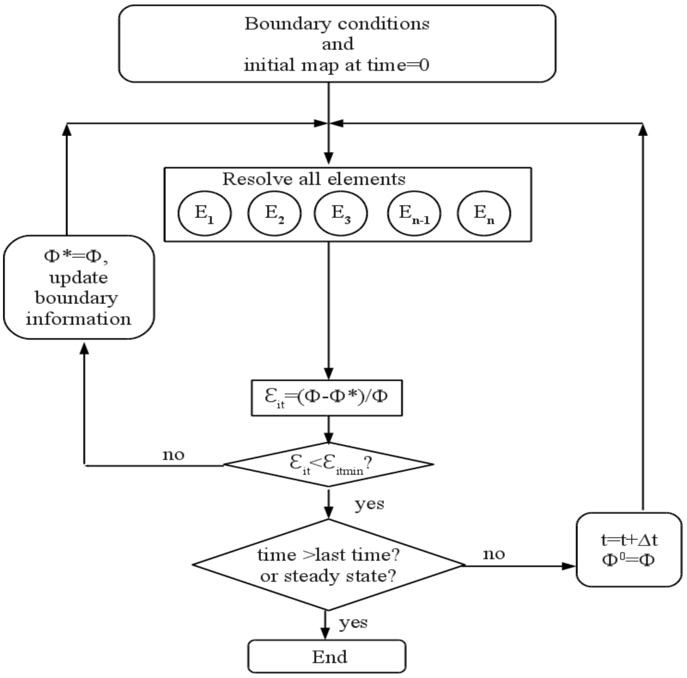

Figure 2: Global resolution algorithm.

3. VERIFICATION AND VALIDATION

3.1 Mixed convection case for the validation of the CFD object used for room air movement

In this section a mixed convection airflow in a three dimensional cavity is considered for the validation of TermoFluids (Lehmkuhl et al., 2007) code. This code will be used further for simulating room air in the CFD object for building applications. The schematic of the test case is shown in the Figure 3. The experimental data for this case has been published by Blay et al. (1992). Also, this case, which is a representative of the airflow in building, has been studied by Mergui (1993), Zhang and Chen (2000) and Ezzouhri et al. (2009). The length (L) and the height (H) of the cavity are 1.04m, and the depth (D) of the cavity is 0.3m. Air enters the cavity top at 15^{o}C with a mean inlet velocity of 0.57 m/s through a slot of height h=0.018m along the depth of the cavity and leaves through a similar slot of height l=0.024m at the bottom of opposite wall. The bottom surface of the cavity is at 35^{o} C while the top and the lateral walls are at 15^{o}C. The front and the rear walls are adiabatic. The Rayleigh number for this case is 2.23x10^{9}, while the Reynolds number and the Archimedes number are 684 and 0.036 respectively. A parabolic inlet velocity distribution is imposed according to the measured velocity profile (Ezzouhri et al., 2009).

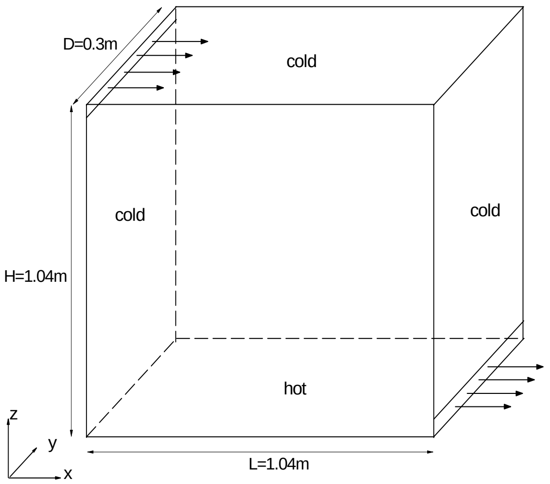

Figure 3: Schematic of the experimental setup.

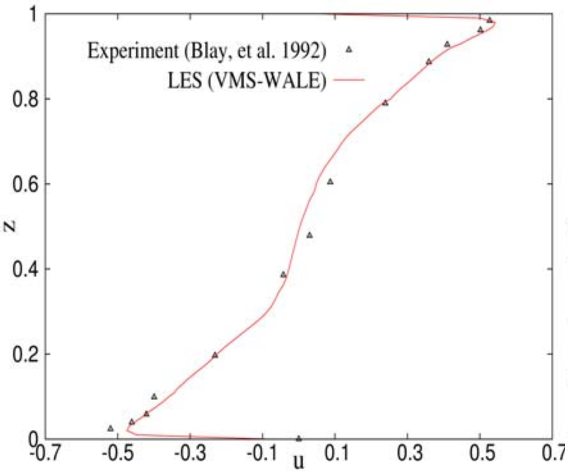

Figure 4a: Mean horizontal velocity (u) with z at y=0.15m and x=0.52m.

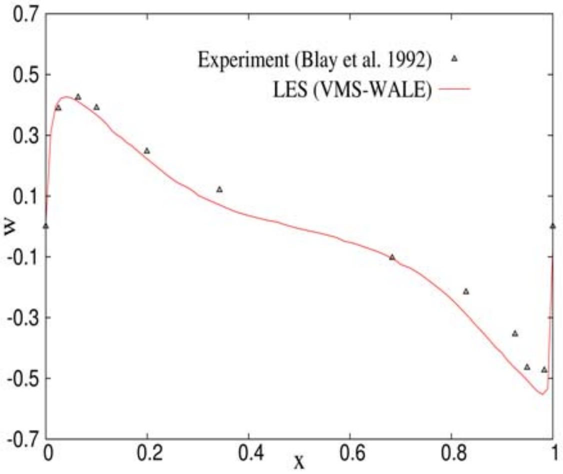

Figure 4b: Mean vertical velocity (w) with x at y=0.15m and z=0.52m.

A mesh with 100x120x32 control volumes has been used for the simulation of this case with 104 processors with an integration time of 500 seconds. VMS-WALE LES model is used for the simulation of this case (Hughes et al, 2000, Nicoud and Ducros, 1999). Numerically obtained mean velocities are compared with the experimental data by 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.

3.2 Verification of NEST with BESTEST cases

For the verification of element models like walls, glass windows, outdoor and room as a single well mixed zone BESTEST qualification cases (600FF and 900FF) published by Judkoff and Neymark (1995) were simulated in the free-floating mode for program validation. Figure 5a and Figure 5b show the hourly values of indoor temperature calculated by NEST for 4th of January for lightweight and heavyweight building structures respectively. It can be seen that the values agree reasonably well with other building programs.

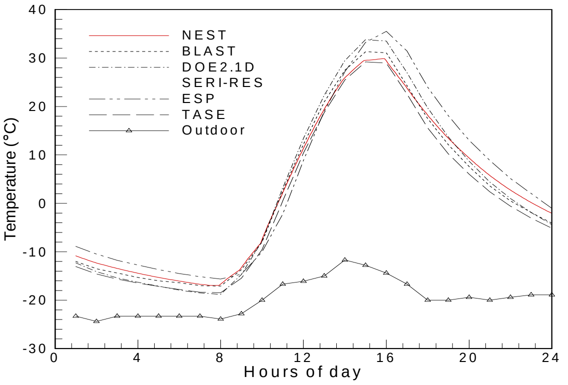

Figure 5a: Indoor temperature variation on 4th January for a lightweight structure.

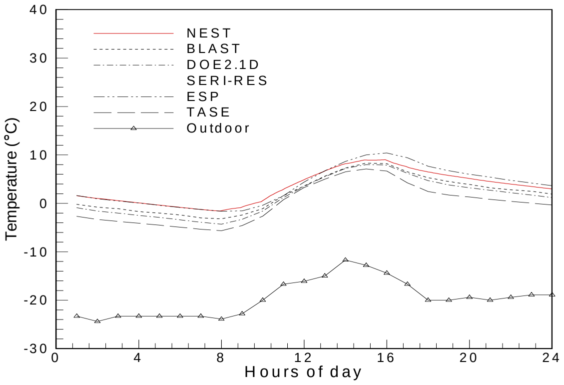

Figure 5b: Indoor temperature variation on 4th January for a heavyweight structure.

4. RESULTS

4.1 Illustrative case

The schematic of the illustrative test case is shown in Figure 6a. Here, the channel is resolved with a CFD&HT model while the room is modeled with a global single well-mixed zone model. The channel which needs more computational resources for CFD&HT analysis is resolved with processors 1, 2 and 3, while the other objects like openings and the room are resolved on processor 4 as shown in Figure 6b.

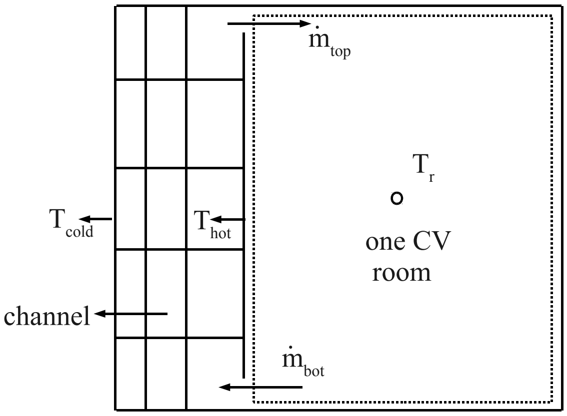

Figure 6a: Schematic of the illustrative case

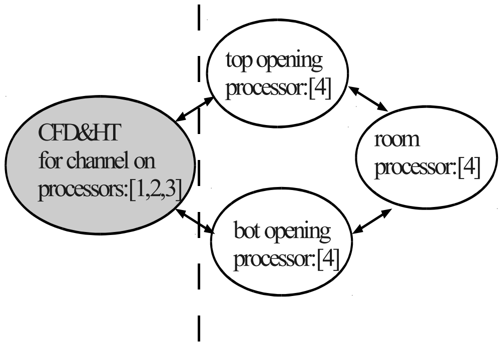

Figure 6b: Partition of elements on different processors.

The channel is exposed to a hot side T_{hot}=301K and a cold side T_{cold}=296K while the rest of the surfaces are adiabatic. The Rayleigh number for this case is about 1.04 $x10^{10}$ and the Prandtl number is 0.71. The air at the hot surface rises creating a suction effect at the bottom of the channel which in turn draw air from the room. The inflow at the bottom of the channel and the outflow at the top of the channel is put as the boundary condition for the room which adjusts itself to both the mass flow rates. The room model is a global model for heat and mass balances assuming a single well mixed zone. A compressible flow large eddy simulation (LES) model (Chiva et al., 2011) is used for the air flow movement in the channel.

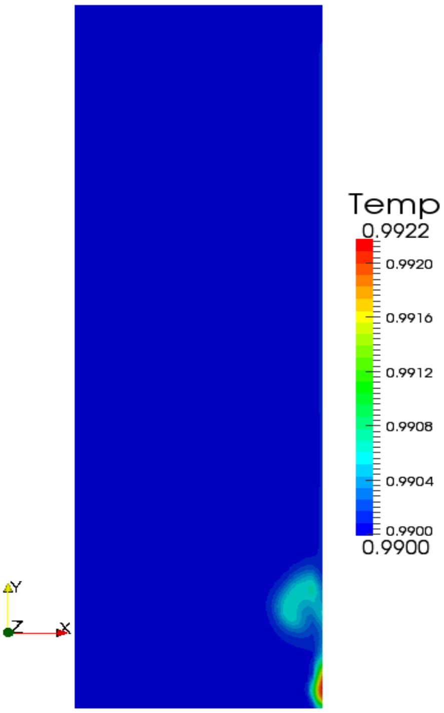

Figure 7a: Temperature map after 40 seconds

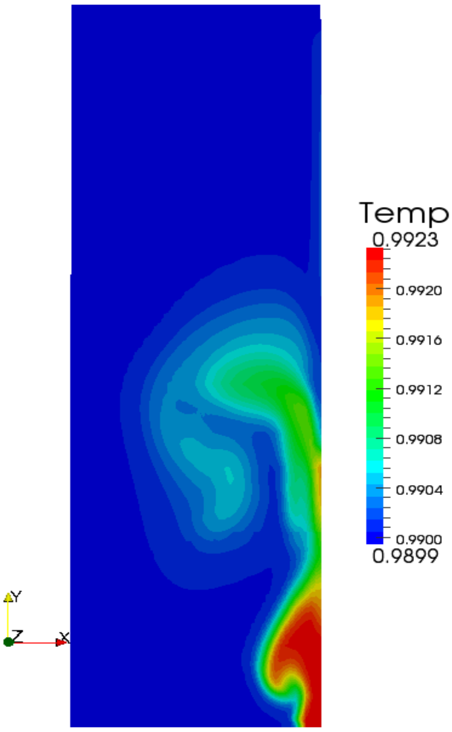

Figure 7b: Temperature map after 80 seconds.

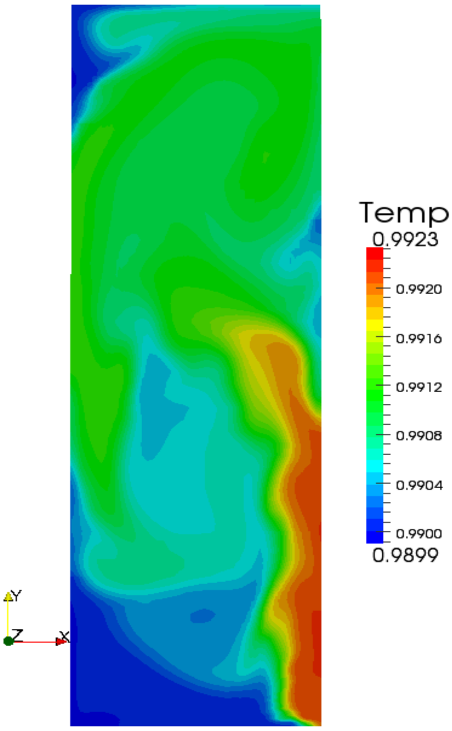

Figure 7c: Temperature map after 120 seconds.

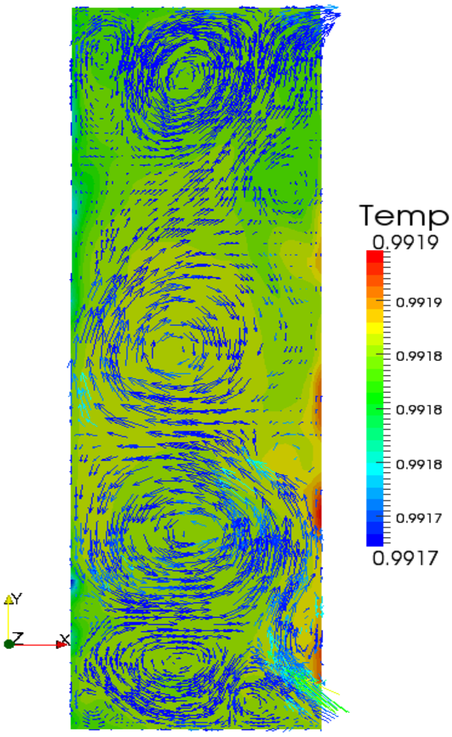

Figure 7d: Temperature map after 300 seconds with velocity vectors.

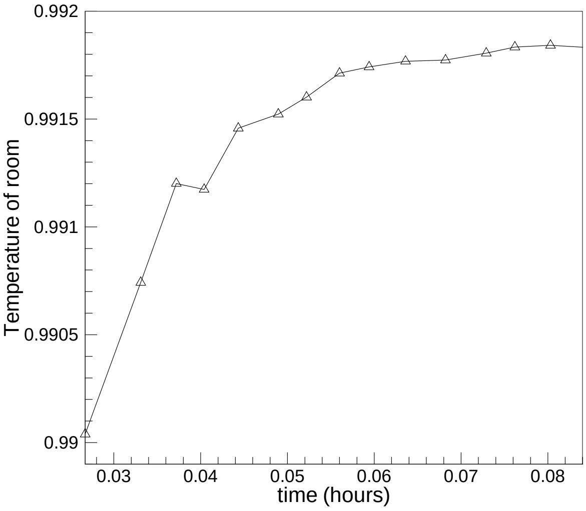

Figure 8: Temperature of the room modelled with single CV.

A rectangular Cartesian mesh with 5200 CVs is used for this case. Figures 7a-7d show preliminary results of the evolution of the temperature field with time. It can be seen how the air near the hot wall starts rising and the flow develops over the entire channel with time. Figure 8 shows the evolution of the temperature in the room modeled as a single well-mixed zone with one control volume. All the temperature values are non-dimensionalized with respect to the highest temperature 301K. The mesh used in this study is coarse, but the main objective of this work is to couple CFD&HT model with global models with a parallel infrastructure. In the future denser meshes and more elements like walls, outdoor, etc. will be incorporated.

5. CONCLUSIONS

A modular object-oriented tool with parallel infrastructure has been presented for the simulation of buildings which also has the possibility to integrate CFD & HT models for required elements of a building system. This is possible because a building or a thermal system is modelled as a collection of basic elements like walls, rooms, glass windows, outdoor, etc. Different models are implemented for different elements allowing different levels of complexity in the same simulation. These elements are programmed only once in a general way and only interact through boundary conditions. Not only there is no need to program specific system configurations as a whole but also, different configurations are made by linking the same basic elements. This results in time savings and adds flexibility to the software the code is also reusable, easy to debug and maintainable. Moreover, the simulation is done on a parallel framework which allows splitting the system in a number of parts which can be resolved on different groups of processors. This will lead to reduction in computational time not only for the cases with CFD & HT models employed for critical zones/elements but also for cases with large number of interlinked elements.

The validation of the CFD & HT object is presented along with the benchmark BESTEST cases for elements like walls, windows, outdoor, and room with single well-mixed zone model. The numerical results obtained are in good agreement with the experimental and benchmark solutions in both the cases. Finally, as an illustrative example, preliminary results for a channel resolved with CFD & HT model is coupled with a room that is resolved with global single well-mixed zone model on 4 processors using the parallel infrastructure.

NOMENCLATURE

CFD & HT: computational fluid dynamics and heat transfer
CV: control volume
E: element of system
T : temperature [^{o}C]
t: time [s]
Δt: time step [s]
x,y,z: cartesian coordinates

Subscripts:

1, 2, ...n: enumeration of the concerned quantity
a: ambient temperature
r: room

Greek letters:

α : heat transfer coefficient [W/ m^2 K]

ϵ_{min} : minimum accuracy demanded
ϵ_{itmin} : accuracy at current iteration
Φ: variable
Φ^0: variable value at previous instant
Φ^*: guess value of a variable

REFERENCES

ASHRAE, 1989, *Handbook of Fundamentals*. Atlanta.

Axley, J., 2007, Multizone airflow modelling in buildingshistory and theory. *HVAC & R Research*,13, pp. 907- 928.

Blay, D., Mergui, S., Nicolae, C. Confined turbulent mixed convection in the presence of a horizontal buoyant wall jet. *Fundamentals of Mixed Convection*, HTD, 213, pp. 65-72, 1992.

Chen, Q., 2009, Ventilation performance of buildings: a method overview and recent applications. *Building and Environment*, 44, pp. 848-858.

Chiva, J., Ventosa, J., Lehmkuhl, O., Pérez-Segarra, C.D., 2011, Modelization of low-Mach Navier-Stokes equations on unstructured meses, *7th International Conference on Computational Heat and Mass Transfer*, Istanbul, Turkey, p.403-411.

Damle, R., Lehmkuhl, O., Perez-Segarra, C.D., 2011, Mixed convection in a ventilated 3D cavity: comparison of different LES models with experimental data, *7th International Conference on Computational Heat and Mass Transfer*.

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.

Feustel, H.E., 1999, COMIS-an international multizone air-flow and contaminant transport model. *Energy and Buildings*, 30, pp. 3-1.

Hughes, T.J.R., Mazzei, L., Jansen, K.E., 2000, Large Eddy Simulation and the variational multiscale method. *Computing and Visualization in Science*, 3, No. 1-2, pp. 47-59.

Judkoff, R., Neymark, J., 1995, Building energy simulation test (BESTEST) and diagnostic method, NREL/TP-472-6231. Golden, Colorado 80401-3393: National Renewable Energy Laboratory.

Lehmkuhl, O., Borrell, R., P´erez-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.

Mergui, S., 1993, Caractérisation expérimentale des écoulements d'air de convection naturelle et mixte dans une cavité fermée. Thèse de l'Université de Poitiers, France.

Musser, A., 2001, An analysis of combined CFD and multizone IAQ model assembly issues, *ASHRAE Transactions*, 107, pp. 371-382.

Negrao, C.O.R., 1998, Integration of computational fluid dynamics with building thermal and mass flow simulation, *Energy and Buildings*, 27 (2), pp. 155-165.

Nicoud, F., Ducros,F., 1999, Subgrid-Scale Stress modelling based on the square of the velocity gradient tensor. *Flow, Turbulence and Combustion*, 62, pp. 183-200.

Stewart, J., Ren, Z., 2006, COwZ:-a subzonal indoor airflow, temperature and contaminant dispersion model. *Building and Environment*, 41, pp. 1631-1648.

Voeltzel, A., Carrié, F.R., 2001, Guarracino, G. Thermal and ventilation modelling of large highlyglazed spaces. *Energy and Buildings*, 33, pp.121-132.

Walton, G.N., 1977, Contam 96 users manual. nistir 6055, USA. National Institute of Standard and Technology.

Wang, L., Chen, Q., 2007, Theoretical and numerical studies of coupling multizone and CFD models for building air distribution simulations, *Indoor Air*, 17, pp. 348-361.

Zhai, Z., Chen, Q., 2003, Solution characters of iterative coupling between energy simulation and CFD programs, *Energy and Buildings*, 35, pp. 493-505.

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.

ACKNOWLEDGEMENT

This work has been partially financed by the Spanish government ("MINECO, Secretaría de Estado de Investigación, Desarollo e Innovación), ref. ENE2009-09496.