

# INTERIOR ROOMS VENTILATION SYSTEM SIMULATION

Raul Stelian Balota<sup>1</sup>, Marius Lolea<sup>2</sup>

<sup>1</sup>University of Oradea, raulbalota@gmail.com

<sup>2</sup>University of Oradea, mmlolea@uoradea.ro

**ABSTRACT:** In this paper we will present the results of a simulation of an interior ventilation system which distributes fresh air in 2 office rooms. The aim is to study the distribution of the temperature inside the 2 rooms in the first seconds after the system starts. We analyzed the velocity and the temperature of the air in different scenarios. Also we are interested in finding the correct velocity which should be used by the system, in order not to determine a sudden temperature change in the room. The study is time dependent, and was simulated using Comsol 5.0 (demo version).

**KEY WORDS:** building, ventilation, simulation, room, air, temperature, velocity, heat flux

## 1. INTRODUCTION

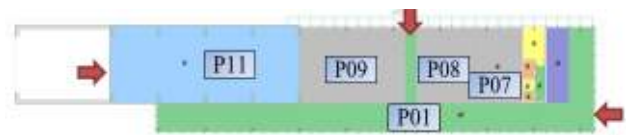
The ventilation system of the building has a very important role, because it must supply the interior ambient with fresh air at a certain speed, having a certain temperature and humidity. This task is not easy to fulfill, especially when we are thinking also at the energetic consumption which is required to be lower every day. In order to meet these many requirements, the ventilation system must be smart which means, to be automated and controlled. The functional principles and parameters of the ventilations system can be established after a detailed analysis of the whole system. Further in our paper, we will analyze the interior air parameters as a result of HVAC distribution in one zone of the building. The building's space destination is as an office building. In this study we will model 2 adjacent rooms, which are used as offices by the employees. The placement of the ventilation inlets and outlets is the same as in the existing building.

## 2. MODEL DESCRIPTION

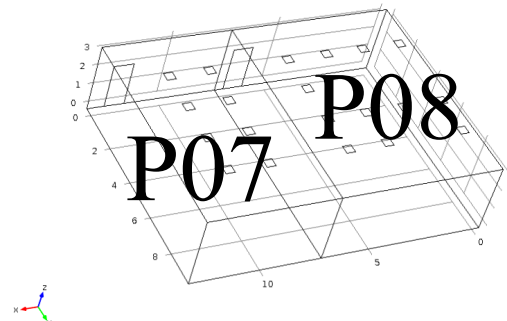
Our paper studies the distribution of the interior air mass, introduced by the HVAC system in the room, and through the ceiling mounted inlets. This study does not take into account the internal heat gains or losses from or to neighboring zones or heat transfer between the studied rooms and exterior zones, it only handles the heat transfer and the motion in the fluids and the heat fluxes between the 2 studied rooms. We used Comsol (demo) CFD (Computed Fluid Dynamic) module. The CFD Module is the platform for simulating devices and systems that involve sophisticated fluid flow models. As is the case with all modules in the COMSOL Product Suite, the CFD Module provides ready-made physics interfaces that are configured to receive model

inputs via the graphical user interface (GUI), and to use these inputs to formulate model equations. The particular physics interfaces that the CFD Module is equipped with enable you to model most aspects of fluid flow, including descriptions of compressible, non-isothermal, non-Newtonian, two-phase, and porous media flows – all in the laminar and turbulent flow regimes.

The geometrical model was designed in AutoCAD 2013 (demo) to be used with the finite element method software Comsol 5.0. This software will enable us to accurately simulate the distribution of the interior air parameters at the start of the HVAC system. The study does not take into account the external factors like radiation, temperature, humidity, pressure. In **Figure1**, is graphically represented the ground floor of the studied building. Our paper will analyze room P08 and P07.



**Figure1.** Ground floor conceptual plan view



**Figure2.** Studied zone – 3D Wireframe view

In the figure above **Figure2**, a 3D wireframe view of the studied zone is presented. As we can observe in **Table1**, room P07 has an area of 56.7 m<sup>2</sup> and room P08 has an area of 72.5 m<sup>2</sup>. Room P07 has 4 occupants and room P08 has 8 occupants. The net

height of the two rooms is 3.00m. Every room has an open door to an interior circulation area, which in our model is considered as an open boundary. The temperature in the circulation area was set to 20°C. The Open Boundary node adds a boundary condition for modeling heat flux across an open boundary; the heat can flow out of the domain or into the domain with a specified exterior temperature. The dimensions of the doors are 900x2150 mm.

### 3. INPUT DATA AND SIMULATION MODELLING

The materials used in this model are imported from the Comsol materials library. See the table below for the materials.

Room P07	Material
North Wall	Concrete
East Wall	5mm Glass
South Wall	5mm Glass
West Wall - Exterior	Curtain Wall
Ceiling	600x600 gypsum board
Room P08	Material
North Wall	5mm Glass
East Wall	5mm Glass
South Wall	5mm Glass
West Wall - Exterior	Curtain Wall
Ceiling	600x600 gypsum board

In Figure3, we represented the initial position of the ceiling inlets, for the 2 rooms. Room P07 has 2 HVAC air inlets (inlets area is 0.72m<sup>2</sup>) and room P08 has 5 HVAC air inlets (inlets area is 1.8m<sup>2</sup>). The inflow velocity of the introduced air for every inlet is the same and constant, 0.2 m/s. The temperature of the introduced air is 24°C and is constant for this study. The characteristics of the intermediate glass wall are:

$$k_s = 0.94 \frac{W}{m \cdot K} \quad \text{Thermal conductivity}$$

$$\rho_{glass} = 2500 \frac{kg}{m^3} \quad \text{Density}$$

$$C_p = 840 \frac{J}{kg \cdot K} \quad \text{Heat Capacity}$$

The study uses 2 modules (**Error! Reference source not found.**) to combine the effect of the fluid inside the 2 rooms. The laminar flow module and the heat transfer in fluids module are combined in a Non-Isothermal Flow study. The laminar flow module which is used to compute the velocity and pressure fields for the flow of a single-phase fluid in the laminar flow regime, so there is no turbulence taken into account. A flow remains laminar as long as the Reynolds number is below a certain critical value. At higher Reynolds numbers, disturbances

have a tendency to grow and cause transition to turbulence.

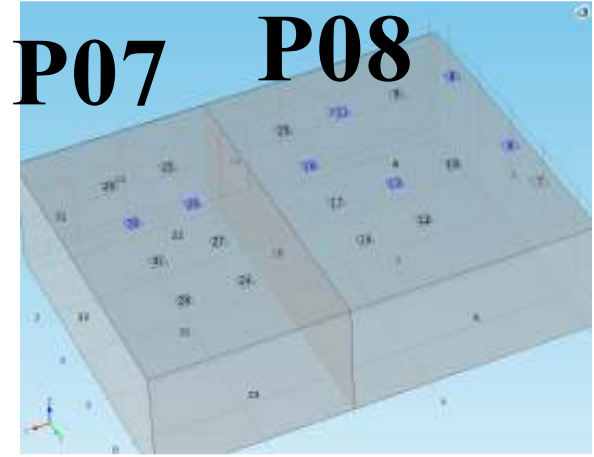


Figure3. Ventilation system inlets, for introduced air

This critical Reynolds number depends on the model, but a classical example is pipe flow where the critical Reynolds number is known to be approximately 2000.

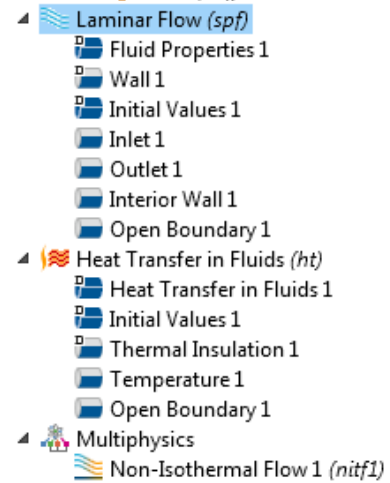


Figure 4. Comsol module interface

The physics interface supports incompressible flows and compressible flows at low Mach numbers (typically less than 0.3). It also supports non-Newtonian fluids.

The equations solved by the Laminar Flow interface are the Navier-Stokes equations for conservation of momentum and the continuity equation for conservation of mass.

The Laminar Flow interface can be used for stationary and time-dependent analyses. Note that for higher Reynolds numbers, a flow becomes inherently time dependent and three-dimensional, and time-dependent studies have to be used.

The Heat Transfer in Fluids is used to model heat transfer by conduction, convection, and radiation. A Heat Transfer in Fluids model is active by default on

all domains. All functionality for including other domain types, such as a solid domain, is also available.

The common wall of the 2 rooms is modelled as and Interior Wall boundary type which is describing the fluid-flow conditions at an interior wall. The Interior Wall condition can only be applied on interior boundaries. It is similar to the Wall boundary condition available on exterior boundaries except that it applies on both sides of an internal boundary.

It allows discontinuities (velocity, pressure, or turbulence) across the boundary. We use the Interior Wall boundary condition to avoid meshing thin structures by applying no-slip conditions on interior curves and surfaces instead. You can also prescribe slip conditions and conditions for a moving wall. The temperature equation defined in fluid domains corresponds to the convection-diffusion equation that may contain additional contributions like heat sources.

	ID	Room name	Destination	Area (m2)	Volume (m3)	Max. occupants [pers]	Density [pers/m2]
Ground Floor	P01	Corridor	Circulation	237.2	782.6	20	0.08
	P02	Cafeteria	Leisure	27.7	91.5	8	0.29
	P03	Restroom Hall	Circulation	7.0	23.1	2	0.29
	P04	Restroom	Restrooms	2.8	9.2	1	0.36
	P05	Storage	Storage	2.8	9.2	1	0.36
	P06	Restroom	Restrooms	14.3	47.2	3	0.21
	P07	Office	Multiple Office	56.7	187.1	4	0.07
	P08	Office	Multiple Office	72.5	239.1	8	0.11
	P09	Classroom	Education	135.5	447.1	15	0.11
	P10	Data Centre	Electronic Equipment	2,97	9,81		
	P11	Atelier	Workshop	226.7	1802.4	7	0.03

**Table1.** Ground floor zone table

In order to take into account the heat transfer between the 2 rooms through the common wall, we used the thin layer feature. This feature is used to define thermal and thickness properties of a thermally resistive material layer located on boundaries. With the addition of the Heat Transfer Module, you can also model heat transfer in thin highly conductive layers.

We use the Non-Isothermal Flow multiphysics coupling to simulate fluid flows where the fluid properties depend on temperature. Models can also include heat transfer in solids as well as surface-to-surface radiation and radiation in participating media, with the Heat Transfer Module. The physics interface supports low Mach numbers (typically less than 0.3).

The Non-Isothermal Flow, Laminar Flow multiphysics interface solves for conservation of energy, mass and momentum in fluids and for conservation of energy in solids. It synchronizes the features from the Heat Transfer and Fluid Flow interface when a turbulent flow regime is defined. It also complements the Screen and Interior Fan feature from the flow interface to account for thermal effects. The physics interface can be used for stationary and time-dependent analysis.

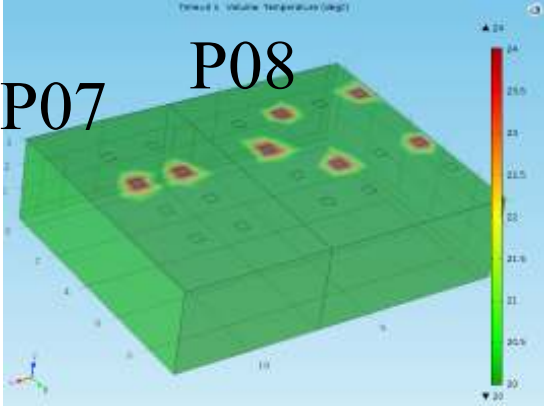
The initial temperature inside the 2 studied offices is of 20°C, the air pressure is 0 Pa and the air velocity is 0 m/s.

## 4. RESULTS AND DISSCUTIONS

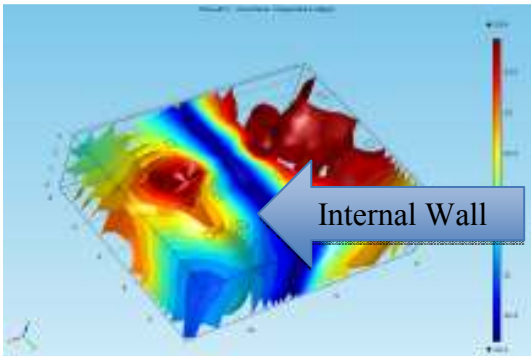
### 4.1 Temperature

Our simulation is time dependent, starting at 0 seconds and ending at 60 seconds. The used step is 5s. In **Figure 4** we observe the graphical temperature illustration, through isothermal lines at the start of the simulation. The HVAC air inlets area has the highest temperature of 24°C. In the figure above, it is visible that the cold air is pushed to the exterior face. This means that, if the thermostat or the temperature sensor is mounted on the interior wall, the readings might not be valid for the entire room, keeping in mind that both rooms have an increased area. According to the current placement of the ventilation inlets, the temperature in the room will not be distributed equally. In the figure above, is represented the isothermal contours of the air temperature in the 2 rooms. It can be seen that warmer air is concentrated around the inlet areas, and is flowing to the edges. The temperatures, in both rooms have values between 20.9 °C and 24 °C. In **Figure6** we observe that the average temperature increases the fastest in room P08, reaching the value

of 23.6°C, while the average temperature in P07 reaches a little below 22.9°C. So in 60s we have a difference of 0.7 °C. The minimum temperature if after 60s in P07 20.9°C, while in P08 22°C. The difference here is of 1.1 °C. Due to the fact that the air is not distributed equally some volumes of air in the room will remain colder. This fact can be observe by the difference between the minimum and the average temperature in the same room.

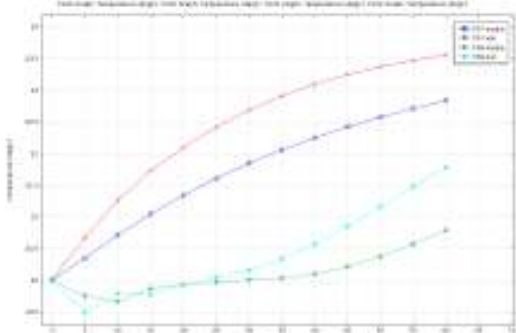


**Figure 4.** Isothermal contours at the start of the simulation.



**Figure5.** Isothermal contours at 60s after the start

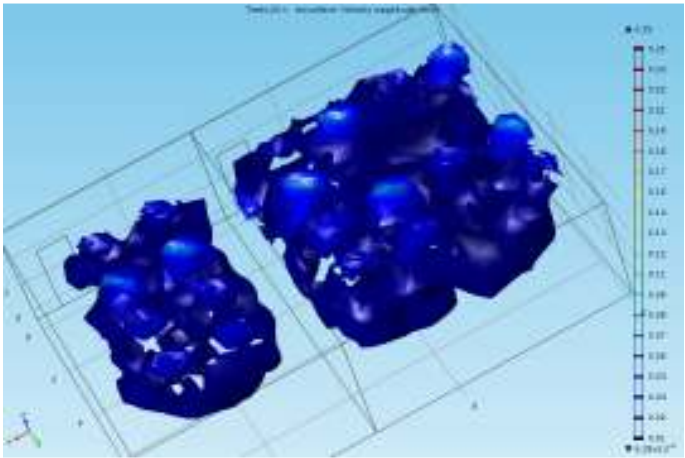
At a speed of 0.2 m/s, according to our simulation in none of the rooms was the air velocity limit surpassed. At 60s the air flow trajectory gets more concentrated in the middle third of the room, please see **Figure7**, which is exactly the area where the office inhabitants ‘head is mostly positioned while standing. This means that, this is an area where the inside air parameters should be optimized for comfort, and the position of the inlets and outlets should be modified.



**Figure6.** Average and minimum room temperature reported to time

#### 4.2 Velocity

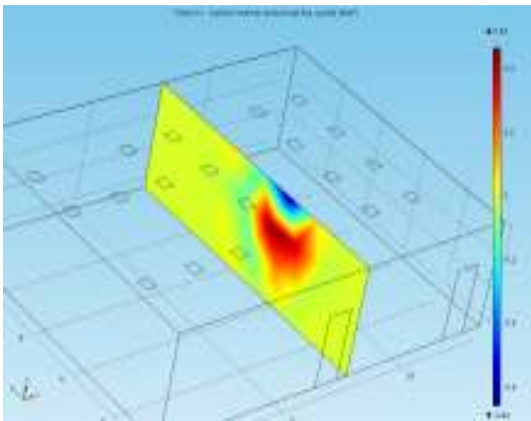
From the air velocity point of view, we observe in **Figure7** that in room P08, a strong air flow in the centre of the room is created. This may cause to the inhabitants discomfort if the interior air velocity passes the value of 0.5 m/s. It can be the case when the doors are open and the ventilation system starts.



**Figure7.** Velocity isosurfaces – area with the same air velocity

#### 4.3 Heat Flux

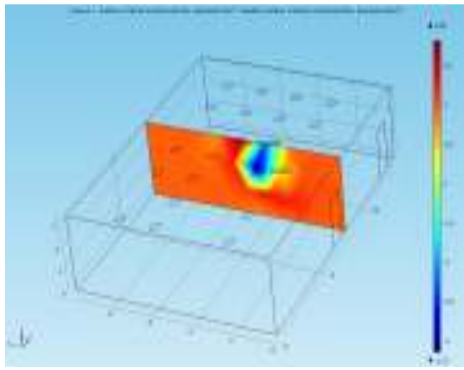
The heat exchange is one of the most important element studied in our simulation. In our paper we studied the conductive heat transfer between the 2 rooms through the intermediate glass wall. This scenario is based on the situation when the air start entering in the both rooms at the same time. Boundary Heat Fluxes - All the domain heat fluxes (vector quantity) are also available as boundary heat fluxes. The boundary heat fluxes are then equal to the mean value of the heat fluxes on adjacent domains. In addition, normal boundary heat fluxes (scalar quantity) are available on boundaries.



**Figure8.** Normal total heat flux values for the glass wall on the face towards P07 at start

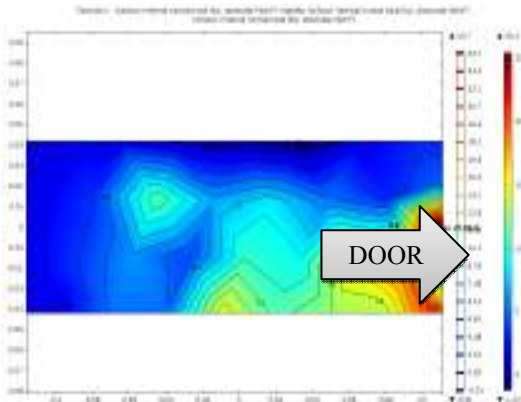
It can be observed in **Figure8**, that as expected the first area which is exposed to heat flux is the upper part of the glass wall, when the warm air starts entering the room.



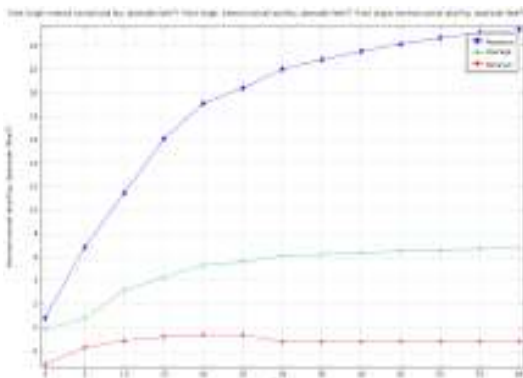


**Figure9.**Normal total heat flux values for the glass wall on the face towards P08 at start

This area (with red in **Figure8**) is below the inlet and in the action range of the incoming air. This part of the wall is the first to be “warmed up”. The value of the absolute maximum heat flux at this point is  $0.65 \text{ W/m}^2$ .



**Figure10.**Internal normal total flux, downside for the glass wall between the two rooms at 60s P08 wall



**Figure11.**Normal total heat flux for the interior wall on the face toward P08

## 5. CONCLUSIONS

In our paper we started a study which can continue with multiple scenarios. This type of analysis is very important in the building design phase. The results of the analyses provided us information about the air movement inside the studied building and sensible areas which are not reached by the fresh air supplied by the HVAC system. This areas can cause the appearance of problems regarding the inhabitants

comfort. The building we studied in our paper is an office building with exterior curtain walls. This means a wide glazed area which is exposed to extreme weather conditions. If on the inside layer of the curtain wall the temperature and ventilations are not optimized, the condensation phenomena could appear. The air distribution inside a large space building, is most accurate modelled and simulated using finite element analysis.

In our model, the number and the position of the air inlets in room P07, are not satisfying the comfort and functional needs of the room. In the temperature analysis we observed that the exterior volume is not enough covered. While the temperature of the air increases in some zones after the HVAC system starts, the distribution of the air remains focused on the same area. The lack of ventilation in that area will create condition for a very unwanted phenomena inside a building, condensation on the glazing element. The interior glass wall is only 5mm thick. Because of this and the disadvantage given by the thermal conductivity of the material, the heat exchange between the 2 rooms is intense. In order to tackle this problem we suggest to use other type of materials if possible or to use special glass.

The heat flux through the intermediate glass wall reaches to a maximum of  $25 \text{ W/m}^2$ , value which is not to be forgotten when taking into account internal heat gains from one room to another.

## 6. REFERENCES

1. COMSOL. 2014. CFD Module. Comsol.com. [Online] 23.12.2014. [Cited: 23.12.2014.] <http://www.comsol.com/cfd-module>.
2. Ade Asmi, Jouvan Chandra Pratama Putra, Ismail Abdul Rahman. 2013. *Fluid Mechanics and Heat Transfer*. Applied Mechanics and Materials . 4th, 2013, Vols. 465 - 466.
3. COMSOL BLOG. <http://www.comsol.com/blogs/modeling-displacement-ventilation-system/>. COMSOL BLOG. [Online] <http://www.comsol.com/blogs/modeling-displacement-ventilation-system/>.
4. COMSOL *Multiphysics validation as simulation software for heat transfer calculation in buildings: Building simulation software validation*. Kateřina Sulovská, Martin Zálešák, Vladimír Gerlich. 2013. Zlín : Tomas Bata University in Zlín, 2013.
5. Yang Li Paul Fazio, PhD Jiwu Rao, PhD. 2010. *Numerical Modeling of Hygrothermal Response in a Full-Scale Experimental Room and through the Envelope*. s.l. : ASHRAE, 201