V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal, 14–17 June 2010

TWO DIMESNIONAL MODELLING WITH CFD OF THE BEHAVIOR OF A VENTILATED CERAMIC FAÇADE.

Mesado C., Chiva S., Juliá E., Hernandez L.

Mechanical Engineering and Construction Department, Jaume I University of Castellón
Campus Riu Sec, s/n, Castellón, Spain
e-mail: schiva@emc.uji.es

Key words: Ventilated Façades, CFD model, MATLAB model, Mediterranean Climate.

Abstract. *Several studies found in the literature are carried out with CFD simulations about the behavior of the thermodynamic phenomena of the double skin façades systems, but the number decreases dramatically if the study is focus on ceramic façades. In this contribution a mathematical model has been developed [1], so the behavior of a ventilated façade with a single channel and steady state can be described; obtaining temperatures distribution of the system and the air flow through the channel. The model input arguments needed are sun radiation, environmental temperature, room temperature and channel wide, as well some materials properties.*

1 INTRODUCTION

A ventilated façade seems a normal façade but it has an extra channel between the concrete wall and the (double skin) façade. Because the façade has, at least, two openings, the air can flow through the channel; this is the main point in ventilated façades. This kind of façades are very useful in sunshine countries, the sun radiation heats the façade, but instead of being accumulated in the system, the air flow increases the heat capacity of air, which is pull out through the upper opening due to the buoyancy effect. In winter season the system is less effective, but the air in the channel can still act as an extra layer of insulation material.

In the recent years the studies of double skin façades have increased, it is because a better thermal comfort is obtained and therefore, these buildings save energy and are more sustainable for the environment. Some of the studies are mathematic models [3,4] or expressions deductions [5,6], but the use of some CFD tools is increasing. Maybe to obtain some parameter [5], that otherwise it would be very difficult to measure or calculate, or just to study which model suited better in this kind of simulations [7]. Only a few studies have been found about different geometries, like [8] in which the channel has been divided in two sub-channels by means of a venetian blind. Also a few 3D simulation studies [8] have been found, due to the huge increased calculation time.

Besides almost all the studies found are with a glass façade [6,4], which for very sunny climates it has been shown that overheats the building.

This study shows that the heat transfer effect is not only by convection (buoyancy) but also a big fraction of the total heat is transferred by radiation between the inner façade and the concrete wall. This is a great consideration when one tries to improve the design of such systems. Nowadays the design of these façades is done in a very inefficient way due to the fact that a small knowledge of the system has been reached. These studies are difficult to carry out because the system is strongly dependent on the local climate [4,10], which can change a lot from one to another country, or even inside one country and/or different seasons.

But the studies done are not enough, more studies about different geometries or materials in use (for example ceramic materials) should be done. Moreover all this studies are very complex to architects; who are, at the end, the people who have to design the façade. Due to this fact, in this project a very simple model with MATLAB has been developed, so the main variables of the phenomena can be obtained; it can be very useful in order to aid architects to design, in a very easy way, a much more efficient façades in their projects.

Summing up as a benefit of this development, one can find an easy way to design façades for architects, so it can improve the amount of energy saved as well as a higher thermal comfort in buildings along all the year. Also, what climates are better to install the façades, what is the appropriate material for the skin façade… among others. As it can be seen, regarding to all of these benefits, the new buildings, which ventilated façades, can be more environmentally friendly.

2 CFD MODEL

2.1 Description

The goal of this point is just to model the system using a CFD commercial code. ANSYS-CFX V12 is chosen. In order to check this model, an experimental facility data are considered. The experimental facility is located in the Technical Institute of Ceramics (Castellón), it consists in a ventilated (ceramic) façade with a simple channel. Nine thermocouples are installed in different points inside the channel, the temperature in the rear wall, environmental temperature , sun radiation , and the air velocity using anemometers inside the channel are measured during several typical days in Mediterranean climates, in summer and winter season. In the figure 1, a caption of CFD model with the different domains can be seen.

Once the model with ANSYS-CFX is constructed, with the same geometry and materials, i.e. ceramic façade and simple channel, the data extracted from the experimental model (environmental temperature, temperature in the rear wall and sun radiation) can be used as boundary conditions. Then the simulation can be run and the results can be compared with the data obtained with the thermocouples and anemometers. Different 24 hours simulations have been performed, both 2D and 3D; is important to note that the results for 2D and 3D are very similar, but the last ones are very time-consuming. In figure 2 the different boundary conditions can be seen implemented in CFD model.

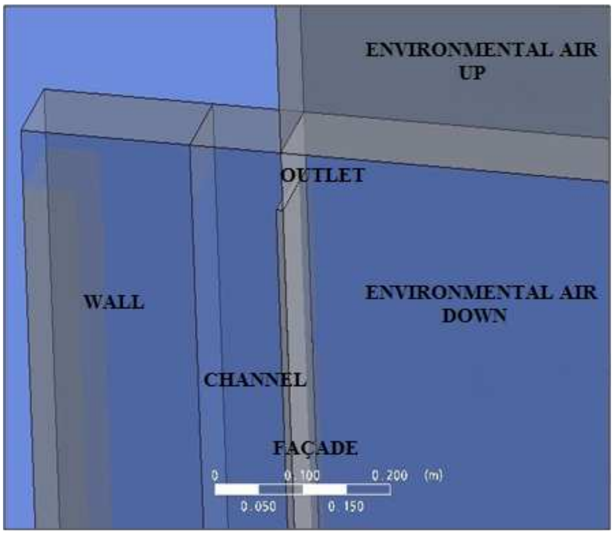


Figure 1: different domains in CFD model.

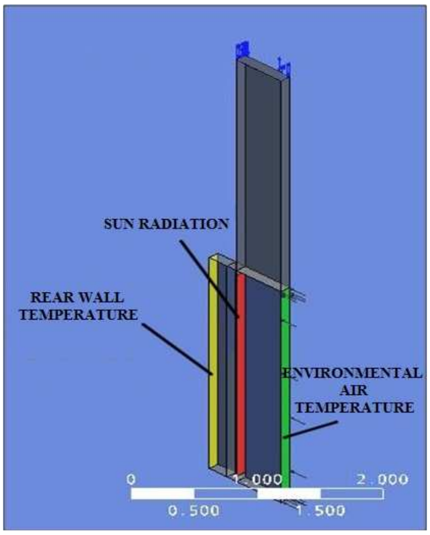


Figure 2: boundary conditions in CFD model.

Domain	Material	Wide (m)	Conductivity (W/mK)
Façade	Tile	0.008	1.6
Channel	Air	0.1	0.02564
Wall	Bricks & concrete	0.140	0.5

Table 1: domain properties in the system.

Different options are chosen to model heat transfer, turbulence and radiation. To model heat transfer, the thermal energy model is chosen, i.e. it neglects the kinetic energy, but it is not affect the calculations since the air velocity is always low. Shear stress transport with Gamma theta transition model is selected to model turbulence, this model is necessary if one wants to get accuracy in the boundary layers, which is very important in this case. This model takes on account the turbulence even at low Reynolds numbers and its transition from laminar to turbulence, in special near the wall where the temperature and velocity gradient is higher Regarding to radiation, discreet transfer model is incorporated.

2.2 CFD model calibration

In this point some comparisons between experimental data and the CFD model are showed. In figure 3, the real temperature in the middle of channel (dots) is compared against the same temperature point in simulations (line); it can be seen that the simulation obtains a slightly decreased temperatures around mid day, this effect is produced because the thermocouples in the real model are not protected against radiation, so the radiation emitted between walls slightly increases the thermocouples measurement.

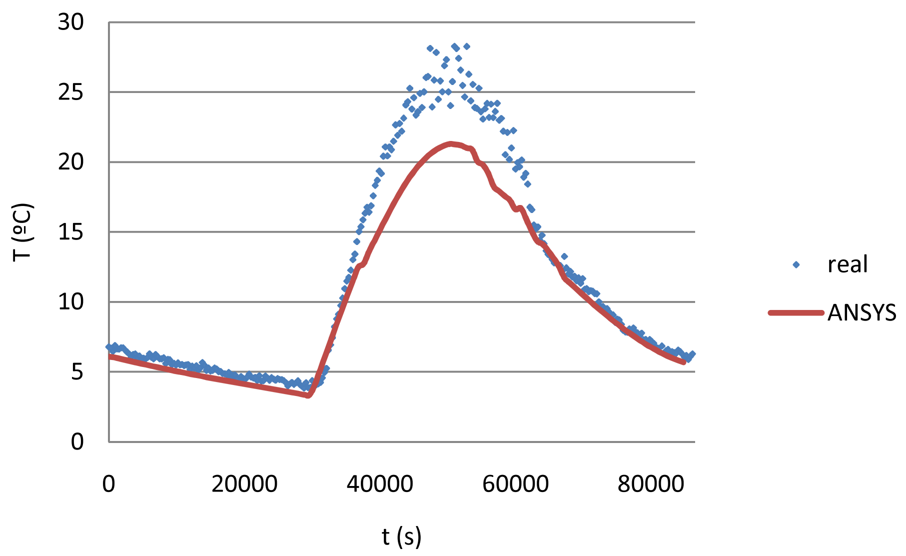


Figure 3: comparison between real (dots) and simulated (line) temperature inside the channel.

In figure 4 the same comparison is done with the air velocity, considering that the air velocity are very low and that it is a variable relatively difficult to measure, the comparison is quite good.

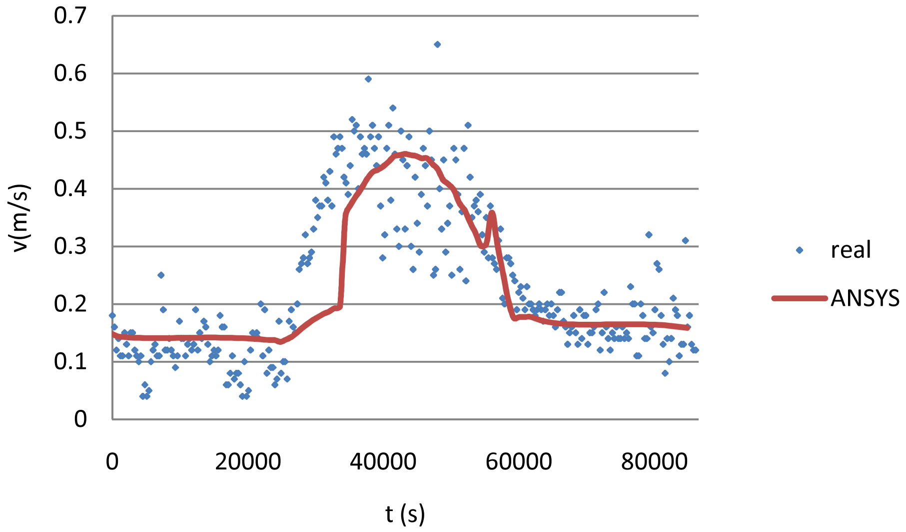


Figure 4: comparison between real (dots) and simulated (line) velocity air inside the channel.

It can seem that the radiation between the interior walls could be neglected, but in fact, it is proportional to the fourth degree of temperature and as it has been explained in figure 3, it affects the phenomena inside the channel. Therefore one wants to know how much this radiation is important in the global heat transfer, in figure 5 the radiative heat (blue) emitted by the façade interior and the convective heat (red) extracted from the façade by the air flow are compared.

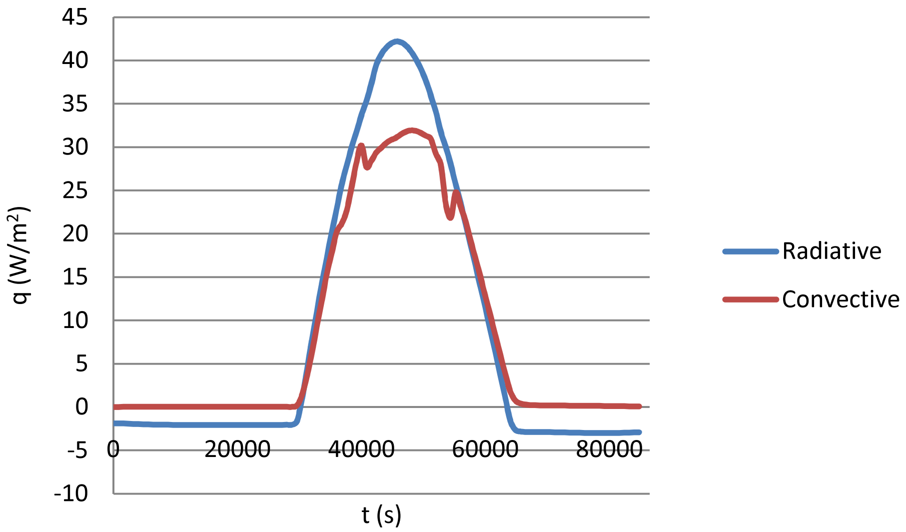


Figure 5: comparison between radiative (blue) and convective (red) inside the channel.

From figure 5 can be deduced that the radiative heat is more important than the convective heat when the sun is at zenith, therefore to improve the design a very simple idea has been implemented: a blind halfway through the channel. In one hand if the blind is closed, the concrete wall is protected against the radiative heat, so the temperature inside the building is reduced; in the other hand if it is open, the radiative transfer is promoted. This phenomenon is observed in figure 6, where the volumetric temperature in the concrete wall is plotted with the blind open (red) and closed (blue).

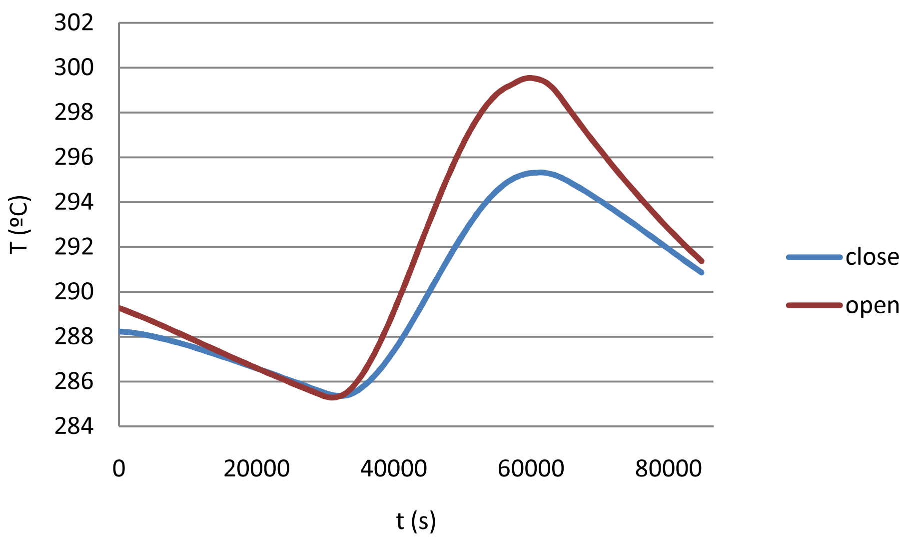


Figure 6: volumetric temperature in the rear wall, with open (red) and closed (blue) blind.

It is clear that the blind accomplish its objective, the volumetric temperature in the concrete wall is higher with the blind open, and again this phenomenon is stronger when the sun is at zenith.

The comparisons are not only with the real data, but also with the papers already published. In [3], Jörn von Grabe shows a possible air velocity distribution through the channel; in [7], M. Coussirat et al. show the outlet air temperature; in [10], D. Faggembauu et al. show the air temperature distribution inside the channel and the mass flow; also in [8], Nassim Safer et al show the air velocity distribution in a channel with a venetian blind. All of these distributions have been checked with the CFD model, and very similar shape distributions have been found.

To conclude this point some conclusions can be made: first, the blind is an effective way to control the radiative heat transfer, which is even more important that convective heat transfer during the day; and second, the model created with CFD is suitable to simulate a ventilated façade with a simple channel. But it should be used carefully in the bottom part, a swirl is observed in the simulations near the inlet air, see figure 7.

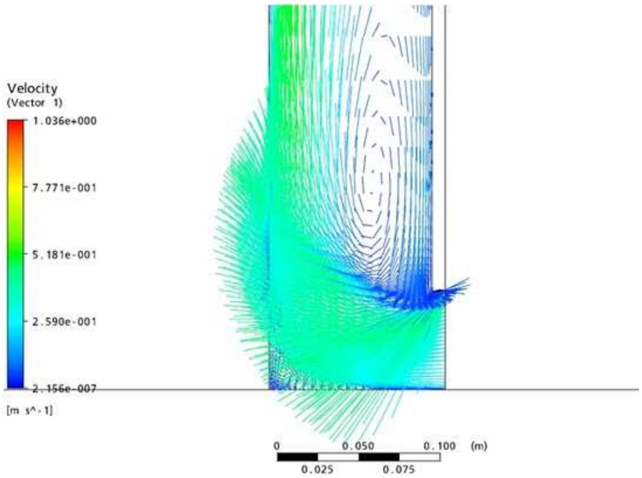


Figure 7: swirl observed at the bottom part, near the inlet air.

3 MATLAB MODEL

In this point a mathematical model of the ventilated façade, developed using MATLAB, is presented. This model has as input arguments: sun radiation, environmental temperature, room temperature, channel dimensions and materials properties; and as output: some surface and volume temperatures, the mass flow, the global heat transfer coefficient and a graphic representation of the temperature distribution through all the channel. The CFD model explained in point two is taken as a basis to develop the MATLAB model and obtain data that were difficult or impossible to be obtained by experimentation.

In figure 8 a sketch of the system with the different heat transfers and also the temperatures variables used latter are represented. The radiative heat transfer is represented by q_{rad}, the convective heat transfer, q_{conv}, and conductive heat transfer, q_{cond}. The temperatures are over some arbitrary surfaces:

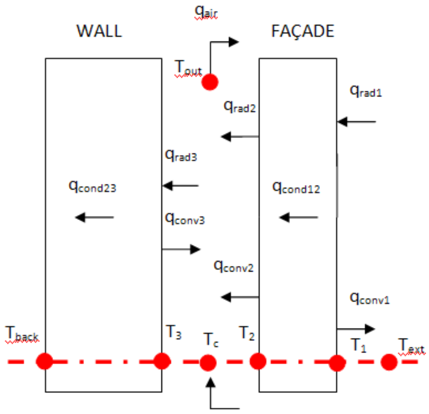


Figure 8: sketch of the system with main temperatures and heats.

3.1 Constitutive Equations

Regarding figure 8, and taking the basis of transport equations in heat transfer, some energy balances are presented. First a balance on the outer façade surface, equation (1); a balance on the inner façade surface, equation (4); then a balance in the channel (5) where q_{air} is the heat that the flowing air takes from the façade and wall by convection; and finally a balance on the concrete wall surface (7).

$$q_{rad1} + q_{conv1} = q_{cond12} \quad (1)$$

$$G + \frac{(T_{ext} - T_1)}{\frac{1}{h_{ext}}} = \frac{(T_1 - T_2)}{\frac{L_t}{k_t}} \quad (2)$$

$$q_{cond12} = q_{rad2} + q_{conv2} \quad (3)$$

$$\frac{(T_1 - T_2)}{\frac{L_t}{k_t}} = \frac{(T_2 - T_c)}{\frac{1}{h_2}} + \frac{(T_2^4 - T_3^4)}{\frac{1}{\varepsilon \cdot \sigma \cdot F}} \quad (4)$$

$$q_{rad2} + q_{conv2} = q_{rad3} + q_{conv3} + q_{air} \quad (5)$$

$$\frac{(T_2^4 - T_3^4)}{\frac{1}{\varepsilon \cdot \sigma \cdot F}} + \frac{(T_2 - T_c)}{\frac{1}{h_2}} = \frac{(T_2^4 - T_3^4)}{\frac{1}{\varepsilon \cdot \sigma \cdot F}} + \frac{(T_c - T_3)}{\frac{1}{h_3}} + q_{air} \quad (6)$$

$$q_{rad3} + q_{conv3} = q_{cond23} \quad (7)$$

$$\frac{(T_2^4 - T_3^4)}{\frac{1}{\varepsilon \cdot \sigma \cdot F}} + \frac{(T_c - T_3)}{\frac{1}{h_3}} = \frac{(T_3 - T_{back})}{\frac{L_w}{k_w}} \quad (8)$$

h_{ext} is the heat transfer coefficient between the environment and the ceramic façade. The value for h_{ext} is found in [17] equal to 14 W/m^2K. Furthermore h_2 and h_3, represent the same coefficient but between a vertical plane and the surrounding flow (inside the channel), these coefficients were found in [14] and [15] to be equal to the next equations;

$$h = \frac{k}{Z}\left(0.825 + \frac{0.387Ra^{1/6}}{\left(1+\left(\frac{0.492}{Pr}\right)^{9/16}\right)^{8/27}}\right)^2 \quad (9)$$

$$Ra = \frac{Pr \cdot g \cdot \beta \cdot (T_p - T_c) \cdot Z^3}{\nu^2} \quad (10)$$

Where Ra is the Rayleigh parameter, this dimensionless number can be seen as the ratio of buoyancy forces and the product of thermal and momentum diffusivities. Note that in the discussion above the different properties of air are taken as constants, this is due to the fact that inside the range of temperature used, their change is negligible.

We have six equations: four balances equations (2), (4), (6) and (8); and two equations for both heat transfer coefficients, which are calculated with the same equation (9), this equation depends on the Rayleigh number, but this in turn depends on the temperatures on the fluid and the wall, so it does not add new variables. But, the variables are seven, namely: T_1, T_2, T_3, T_c, h_2, h_3 and q_{air}. So a new equation should be added, this equation is extracted from the CFD model explained in point two, some simulations are solved with different input parameters, i.e. sun radiation, environmental temperature and channel wide. All the combinations in table 2 are simulated, leading to a total number of 72 simulations.

L (m)
0.075
0.1
0.125

G (W/m2)
200
400
600
800

Tamb (K)
287
290
293
296
299
302

Table 2: simulations in CFD model, summarize of different cases.

From these simulations data were extracted, both to validate the MATLAB model (which is shown in point four) and to create a new equation. This equation is the minimum temperature of the air in the channel, T_c, for a certain height in function of the dimensionless height, z/Z, sun radiation, G, environmental temperature, T_{ext} and channel wide, L.

$$T_c = \left(\frac{G}{100} + \frac{-70 \cdot L + 16}{9}\right)\frac{z}{Z} + \left(T_{ext} - \frac{102 \cdot L + 11}{3}\right) \quad (11)$$

In the next graph, some discrete data obtained in CFD model about the minimum air temperature in the channel are shown. It can be seen that if the temperature is represented against the dimensionless height, some lines are obtained with the same slope but different intersection depend on the environmental temperature.

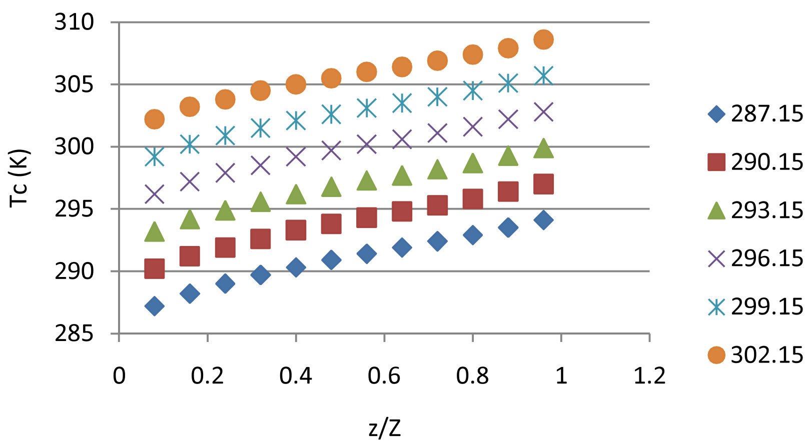


Figure 9: minimum air temperature against dimensionless height, different series represent the environmental temperature, ranging from 14 ºC to 29 ºC.

With equation (11) the number of equations and variables are the same, so a set of seven equations, some of them nonlinear, can be solved in MATLAB to obtain the variables: T_1, T_2, T_3, T_c, h_2, h_3 and q_{air}.

The next step is to calculate a volumetric temperature of air inside the channel; this variable is interesting in order to get an idea about the system performance. First a temperature distribution over the entire channel is needed, this can be provided by equation (12). This equation express the dimensionless temperature in function of dimensionless deep across the channel, at a certain height, see figure 10.

$$\frac{T_y - T_c}{T_p - T_c} = \left(-\left(\frac{y}{L}\right)^{\frac{1}{7}} + 1\right) \qquad (12)$$

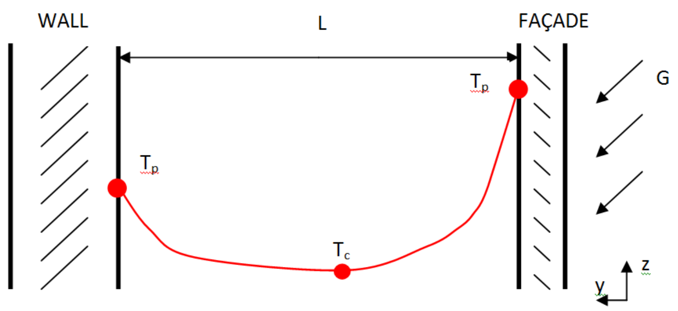


Figure 10: temperature distribution between two infinite vertical planes.

Note that there are two planes (walls), and the equation (12) gives the temperature distribution over one plane. So to find the temperature distribution, with equation (12), the channel is divided into two regions, one for each plane (or wall); and the separation of these regions is denoted by the minimum temperature at this height, Tc, which could not be in the middle of the channel.

Once the temperature over all the façade high in the different surfaces and inside the channel are obtained, some new variables can be calculated, such as volumetric channel temperature, equation (14), superficial temperatures (15), and outlet air temperature

(16). This new temperatures could be calculated by means of an average temperature over all the points in a volume or area (area or line in 2D), which mathematically is expressed as:

$$T_{vol} = \frac{\int_0^Z \int_0^L T_y \, dy \, dz}{Z \cdot L} \quad (14)$$

$$T_{sup} = \frac{\int_0^Z T_p \, dz}{Z} \quad (15)$$

$$T_{out} = \frac{\int_0^L T_y(z = 1) \, dy}{L} \quad (16)$$

Moreover the mass flow through the channel can be calculated with a simple energy balance.

$$m = \frac{q_{air}}{C_p(T_{out} - T_{ext})} \quad (17)$$

The last parameter is the global heat transfer coefficient, U, which is the inverse of the sum of all the individual resistances in the system; with this parameter some heat transfer calculus can be done easily. It involves two resistances across a solid, façade and wall; and three resistances across a fluid, one because the environmental air and two inside the channel (remember that the channel is divided into two regions, each one with a different individual heat transfer, h_2 and h_3). The form of these resistances is known, the system is just a semi-infinite vertical planes through with the heat is transferred in serial way.

$$R_{ext} = \frac{1}{h_{ext}} \quad (18)$$

$$R_{façade} = \frac{L_t}{k_t} \quad (19)$$

$$R_{wall} = \frac{L_w}{k_w} \quad (20)$$

For the fluid resistance, an average individual heat transfer is defined as:

$$\bar{h} = \frac{\int_0^Z h \, dz}{Z} \quad (21)$$

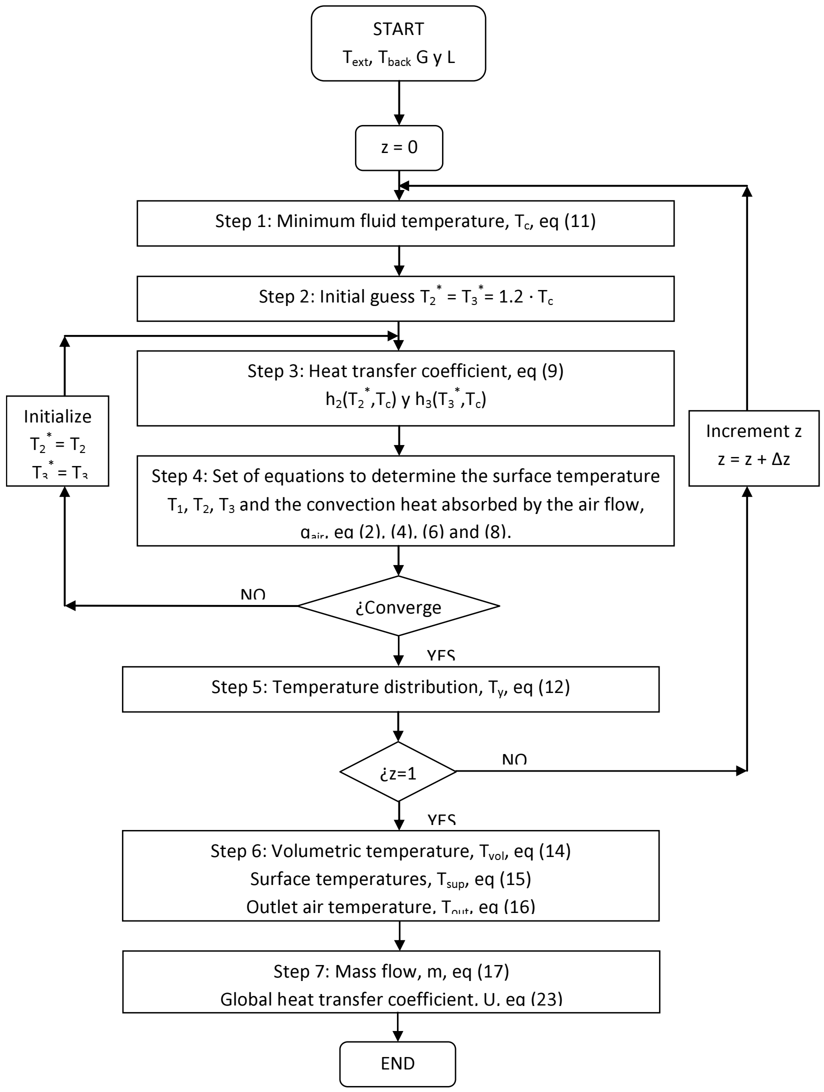
$$R_{channel} = \frac{1}{\bar{h}_2} + \frac{1}{\bar{h}_3} \quad (22)$$

After all the calculations are done, this parameter is easy obtained with equation (23).

$$U = \frac{1}{\frac{1}{h_{ext}} + \frac{L_t}{k_t} + \frac{1}{\bar{h}_2} + \frac{1}{\bar{h}_3} + \frac{L_w}{k_w}} \quad (23)$$

3.2 Model flow chart

Next, a flow chart of the complete model is sketched; it starts with the some parameters provided by the used, it has two loops: the main loop, in which the height is increased progressively, and the secondary loop to converge a set of equations. After that some variables are calculated using the temperature distributions.



4 MODEL VALIDATION

In point three some equations are explained; with them some important parameters in a ventilated ceramic façade can be calculated easily. In this point some validations are carried out, the goal is to compare the data extracted from MATLAB model with CFD model, which is already explained in point two. Moreover, in this point, the dots in all the charts are the data extracted from CFD model, and the continuous lines are MATLAB model results; for the bar charts the relative error is shown above each set of bars.

4.1 Punctual temperatures: surfaces, minimum and distribution

First, the surface temperature is compared for the main surfaces, outer and inner sides of façade and the inner side of the wall. In figure 11 can be seen that the agreement is quite good, all the surfaces are cooled due to the air flow, this cooling is effective while the air flow is not heat saturated, once the air is saturated, the cooling is slowed down. Some disturbances appear near the air inflow, them are due to the swirl commented at the end of point two. These data are obtained solving simultaneously the equations (2), (4), (6) and (8).

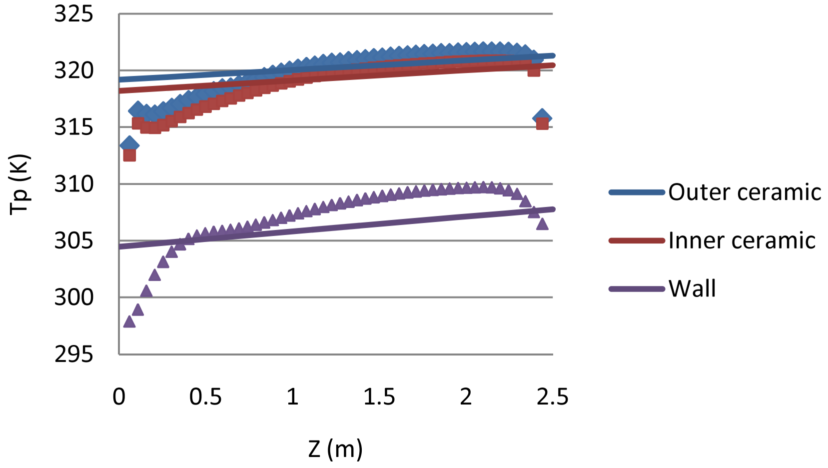


Figure 11: punctual surface temperature against the height, with parameters G = 600 W/m^2 and T_{ext} = 17 ºC. Three total surfaces: outer façade side (blue), inner façade side (red) and concrete wall channel side (purple); colored version online.

In figure 12 the minimum air temperature in the channel (equation 11) is represented against the dimensionless height for different environmental temperatures, it is observed that this temperature increases while the air flows through the channel, as commented before the air absorbs heat from the walls, so it is heated up.

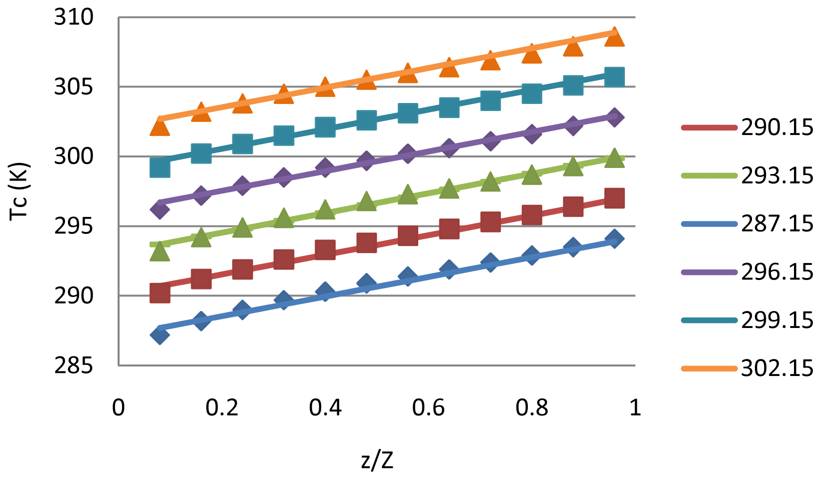


Figure 12: minimum air temperature at a certain height against dimensionless height, different series represent different environmental temperature, while sun radiation is constant to G = 600 W/m^2, in this case.

In figure 13 the temperature distribution over the entire channel with equation 12 is plotted against dimensionless deep, 0 is the façade side and 1 is the wall side; five series represent the profiles at different heights, over a total height of 2.5 m. Once more can be deduced that the air temperature is increased while it flows through the channel, and a minimum temperature around the middle deep of the channel is found, as represented by equation 11. Next to the walls the temperature gradient is much larger than in the middle, and of course the temperature in the façade side is higher than in the wall side. The results are in agreement with the CFD results and the experimental ones.

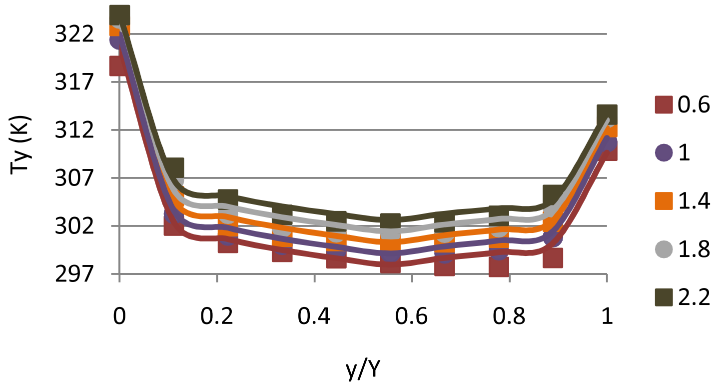


Figure 13: temperature distribution inside the channel against dimensionless deep, 0 is the façade side and 1 is the wall side. Different series represent different heights, the parameters are G = 600 W/m^2 and T_{ext} = 23 ºC.

4.2 Heats flux: convective and radiative

Solving simultaneously the equations (2), (4), (6) and (8), the heat absorbed by the air can be resolved. Figure 14 shows the heat absorbed by the air against the environmental temperature for the 800 W/m^2 case. This heat is reduced with an increase of environmental temperature. The relative error is a little bit higher than in the other figures, but this is due to the fact that the mass flow through the channel is small, so a little deviation produces a relative greater error

After all the variables in the set of equations are worked out, the different heats represented in figure 8 can be calculate, here just the convective and radiative heat from and to the wall are represented in figure 15 and 16 respectively. The convective heat is reduced with environmental air because saturation of air effect commented before; moreover the radiative heat is proportional to the environmental temperature because the difference between inner façade and wall temperature (figure 19 and 20) also increase with environmental temperature, this effect is reinforced with the fourth degree of temperature.

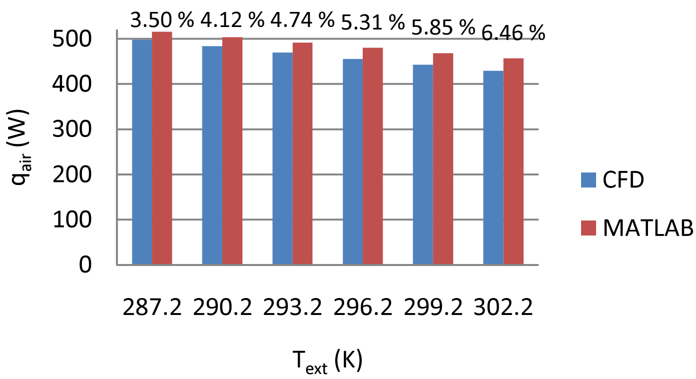


Figure 14: heat absorbed by the air flow against environmental temperature, in this case the sun radiation is constant to G = 800 W/m^2. Blue bars are CFD data and red bars are Matlab data.

.

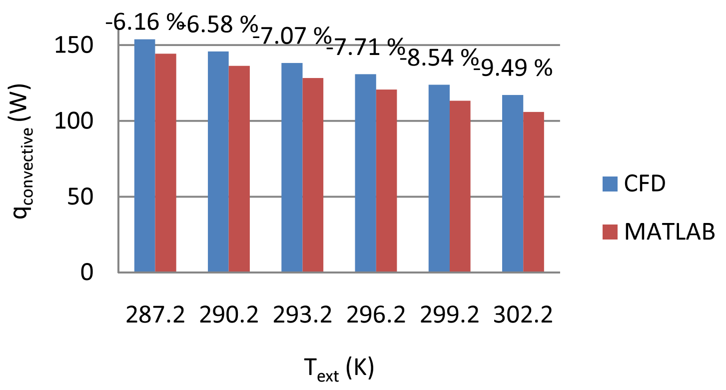


Figure 15: convective heat yielded from concrete wall against environmental temperature, sun radiation is constant to G = 800 W/m^2. Blue bars are CFD data and red bars are Matlab data.

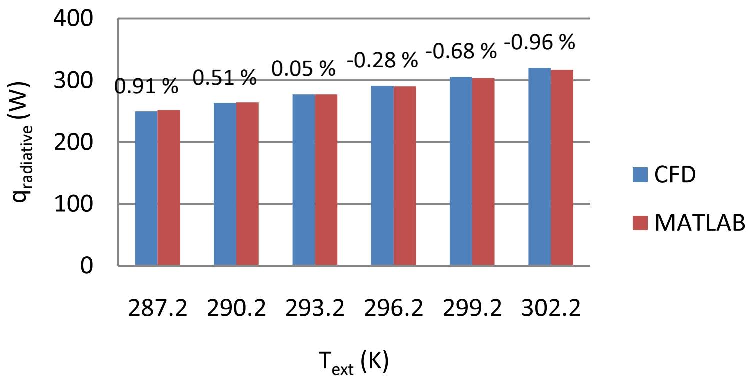


Figure 16: radiative heat to concrete wall against environmental temperature, sun radiation is constant to G = 800 W/m^2. Blue bars are CFD data and red bars are Matlab data.

4.3 Volumetric temperature in the channel

The volumetric temperature validation, with equation (14), is pretty good in all cases. Here the data for 400 W/m^2 and different environmental temperature are plotted. Of course the value increases with environmental temperature.

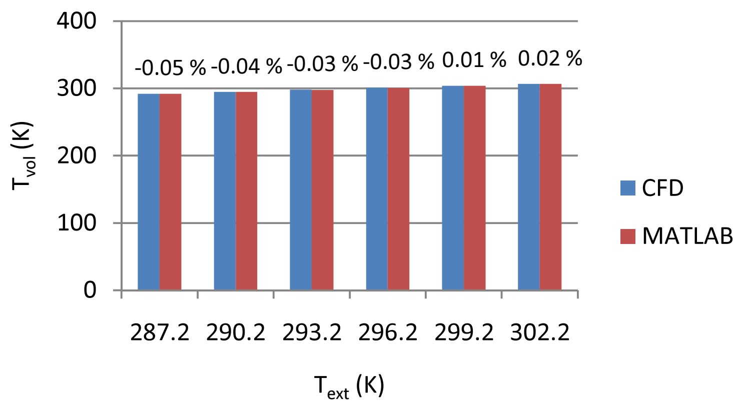


Figure 17: volumetric channel temperature against environmental temperature, sun radiation is constant to G = 400 W/m^2. Blue bars are CFD data and red bars are Matlab data.

4.4 Surface temperature: inner and outer façade and inner wall surface

Also the data with equation (15) and (16), surface temperature and outlet temperature respectively, are quite good respect to the CFD data. Next the result for the 800 W/m^2 case are shown for the three main surfaces (figure 18, 19 and 20). All of them increase with environmental temperature.

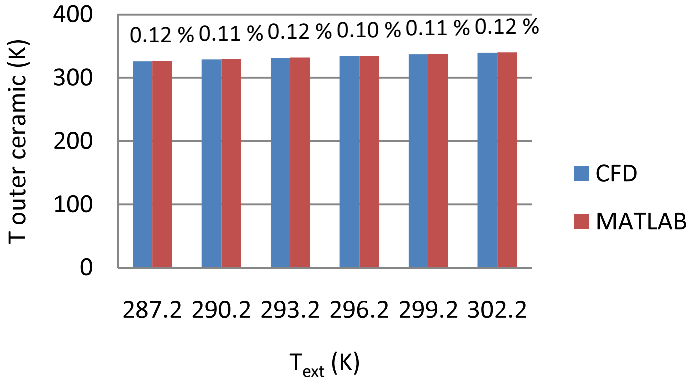


Figure 18: outer ceramic surface temperature against environmental temperature, sun radiation is constant to G = 800 W/m^2. Blue bars are CFD data and red bars are Matlab data.

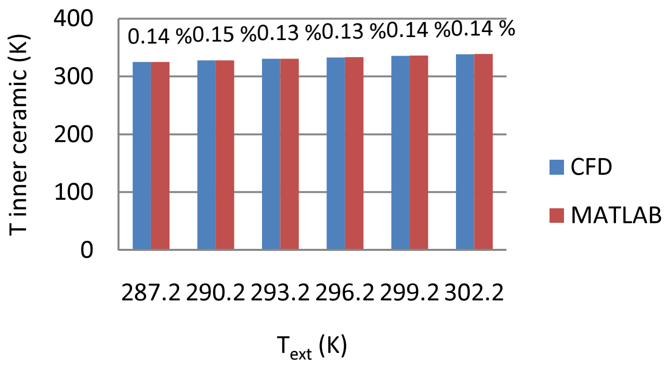


Figure 19: inner ceramic surface temperature against environmental temperature, sun radiation is constant to G = 800 W/m^2. Blue bars are CFD data and red bars are Matlab data.

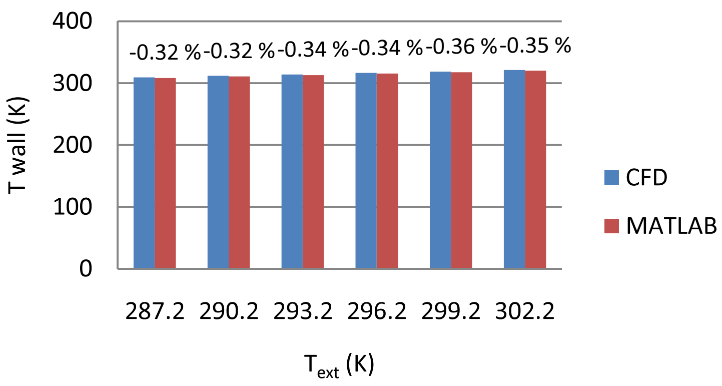


Figure 20: concrete wall surface temperature against environmental temperature, sun radiation is constant to G = 800 W/m^2. Blue bars are CFD data and red bars are Matlab data.

4.5 Outlet air temperature

The outlet air temperature for the 400 W/m^2 case is shown at figure 21.

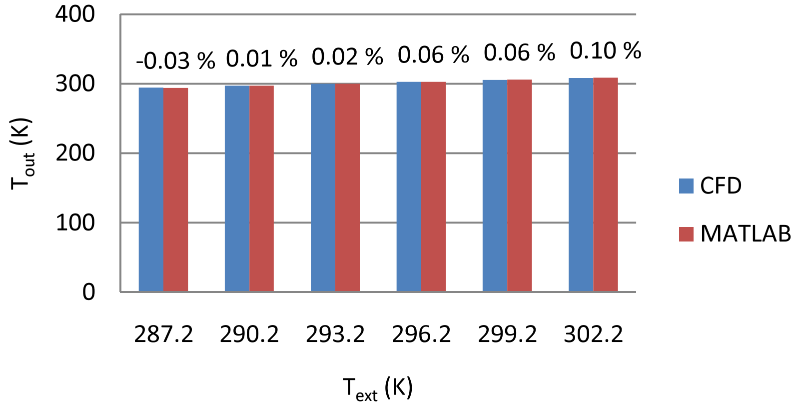


Figure 21: outlet air temperature against environmental temperature, sun radiation is constant to G = 400 W/m^2. Blue bars are CFD data and red bars are Matlab data.

4.6 Mass flow

Mass flow can be calculated with equation (17), which is plotted in figure 22, for the 600 W/m^2 case. As in figure 14 (heat absorbed by the air), in figure 22 the relative error seems significant, but again due to the small mass flow, a little deviation produces a relative big error. The mass flow reduces with the environmental temperature because the difference between outlet (figure 21) and inlet temperature is also decreased with environmental temperature; moreover the mass flow increases with sun radiation.

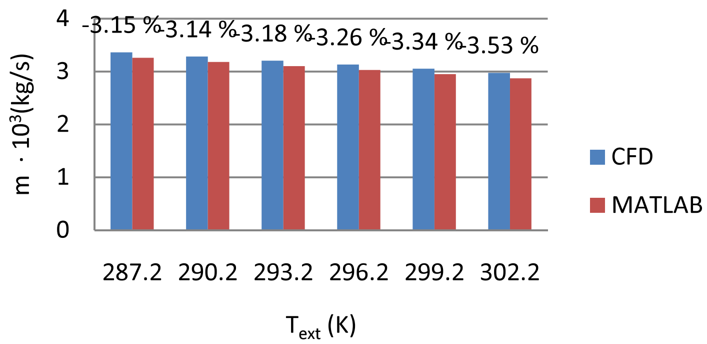


Figure 22: mass flow against environmental temperature, sun radiation is constant to G = 600 W/m^2. Blue bars are CFD data and red bars are Matlab data.

4.7 Global heat transfer coefficient

Finally the global heat transfer coefficient, equation (23), is plotted against the environmental temperature in figure 23, for different sun radiations, ranging from 200 W/m^2 to 800 W/m^2.

It is important that the bigger resistance is provided by the channel, it is expected because in fluids the heat transfer is more difficult than in solids and furthermore the flow velocity is very small so the air behaves as a laminar flow, which also difficult the heat transmission. Also the concrete wall resistance is big due its wide, and the façade resistance negligible due its high conductivity and small wide.

These data cannot be compared with CFD model, since it is no possible to obtain the global transfer coefficient in a straight way in CFD.

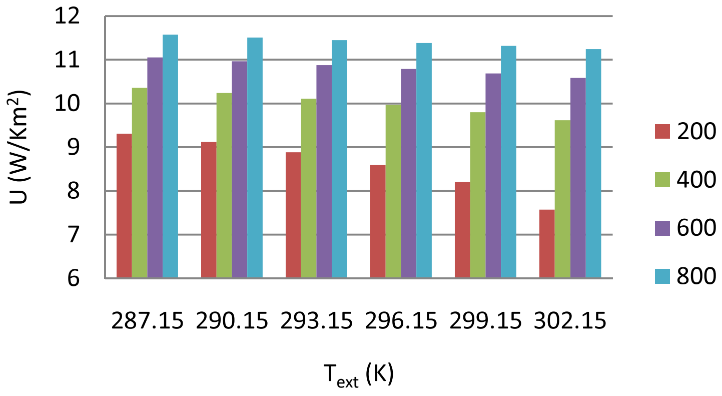


Figure 23: global transfer coefficient against environmental temperature, different series represent sun radiation. All bars are Matlab data since it is not possible to obtain in CFD model.

In figure 23 it is observed that the global coefficient increases significantly with sun radiation, which seems logical, but slightly decreases with environmental temperature. This effect is explained next. In one hand, when the sun radiation is high, we can approach that the global coefficient is not environment-temperature dependent; on the other hand, when the sun radiation decreases, the global coefficient turns to be strong dependent on the environmental temperature (inversely proportional). To explain what happens, it is needed to look at the individual resistances: when the environmental temperature is reduced the resistance associated with the concrete wall and the external-channel (façade side) increase; but the resistance associated with the internal-channel (wall side) decreases; note that these are the most significant resistances (more than 90%). The increase in the internal-channel resistance is enough to gain the decrease in the external-channel plus wall resistance, thus the global resistance is proportional to the environmental temperature, so the global coefficient is inversely proportional to environmental temperature.

5 CONCLUSIONS

A model in MATLAB has been implemented to calculate some important parameter in a ventilated system (surface, volumetric and distribution temperatures, the global heat transfer coefficient and mass flow). This model has been validated by means of a model created in CFD, which is also validated by some experimental data extracted from a real ventilated façade.

Some conclusions could be made. A blind installed in the middle plane through the channel is a good idea to control efficiently the radiation transfer in this system. Furthermore the system (with simple channel) has a better heat transmission when the sun radiation is higher and a low environmental temperature.

So it is concluded that the system, with a simple channel, is good at least in winter Mediterranean climates when one wants heat transfer. In such climates the average environmental temperature is a little bit lower than 14 ºC (the minimum temperature studied in this work) and there are a lot of days with intense sun radiation. But in summer, when one does not want heat transfer and sun radiation is mainly high, should be a good idea to install the blind commented in the last paragraph.

The model could be improved if a transitory state is implemented or even update to 3D; besides convective or radiative sub-models could be changed in order to increase the accuracy; also more variables could be added to the model, and finally it would be interesting if the range of environmental temperature and/or sun radiation could be extended.

6 NOMENCLATURE

β: air compressibility, 3.421·10-3 m^3/N.
*ε: tail façade emissivity, dimensionless.
ν: air cinematic viscosity, 15.09·10-6 m^2/s.
σ: Stefan-Boltzmann constant, 5.67·10-8, W/m^2K^4.
Cp: specific heat of air, 1004 J/kgK
F: vision factor between the tile and the concrete wall, 0.9 dimensionless.
g: gravity acceleration, 9.81 m/s^2.
*G: sun radiation, W/m^2.
h_2: heat transfer coefficient between the ceramic façade and the chamber, W/m^2K.
h_3: heat transfer coefficient between the concrete wall and the chamber, W/m^2K.
h_{ext}; heat transfer coefficient between the environment and the façade, W/m2K.
k: air thermal conductivity, 25.64·10-3 W/mK.
*k_w: concrete wall thermal conductivity, W/mK.
*k_t: tile façade thermal conductivity, W/mK.
*L: total wide of channel, m.
*L_w: concrete wall wide, m.
*L_t: façade wide, m.
m: mass flow through the channel, kg/s.
Pr: Prand number for air, 0.713 dimensionless.
q_{air}: heat absorbed by air flow in the channel, W/m^2.
q_{conv}: heat transfer by convection, W/m^2.
q_{rad}: heat transfer by radiation, W/m^2.
q_{cond}: heat transfer by conduction, W/m^2.
Ra: Rayleigh number, dimensionless.
T_1: façade surface temperature (outer), K.
T_2: façade surface temperature (inner) , K.
T_3: concrete wall surfaces, K.
*T_{back}: temperature in the inside wall surface, K.
T_c: minimum temperature in the channel at a concrete height, K.

*T_{ext}: environmental temperature, K.
T_{out}: outflow air temperature, K.
T_p: wall temperature at a concrete height, K.
T_{sup}: surface temperature, K.
T_{vol}: volumetric channel temperature, K.
T_y: fluid temperature inside the channel at a certain height, K.
y: deep coordinate, m.
z: height coordinate, m.
*Z: total height of channel, m.

REFERENCES

[1] Mesado C. "*Modelización del comportamiento de una fachada ventilada*", Master Thesis, Jaume I University, (2009).

[2] Elisabeth Gratia, André De Herde, Natural ventilation in a double-skin façade, Energy & Buildings (2003).

[3] Jörn von Grabe, A prediction tool for the temperature filed of double facades, Energy & Building (2002).

[4] D. Faggembauu, M. Costa, M. Soria, A. Oliva, Numerial analysis of the termal behaviour of ventilated glazed facades in Meditterranean climates. Part I: development and validation of a numerical model, Solar Energy (2003).

[5] A. Pappas, Z. Zhai, Numerial investigation on thermal performance and correlation of double skin façade with buoyancy-driven airflow, Energy & Buildings (2007).

[6] Guohui Gan, Thermal transmittance of multiple glazing: computational fluid dynamics prediction, Applied Thermal Engineering (2001).

[7] M. Coussirat, A. Guardo, E. Jou, E. Egusquiza, E. Cuerv, P. Alavedra, Performance and influence of numerical sub-models on the CFD simulation of free and forced convection in doublé-glazed ventilated façades, Energy & Buildings (2008).

[8] Nassim Safer, Monika Woloszyn, Jean Jacques Roux, Three-dimensional simulation with a CFD tool of the airflow phenomena in single floor double-skin façade equipped with a venetian blind, Solar Energy (2004).

[9] Guohui Gan, Simulation of buoyancy-induced flow in open cavities for natural ventilation, Energy & Buildings (2005).

[10] D. Faggembauu, M. Costa, M. Soria, A. Oliva, Numerial analysis of the termal behaviour of ventilated glazed facades in Meditterranean climates. Part II: applications and analysis of results, Solar Energy (2003).

[11] Heinrich Manz, Numerical simulation of heat transfer by natural convection in cavities of facade elements, Energy & Buildings (2002).

[12] Wang Liping, Wong Nyuk Hien, The impact of ventilation strategies and facade on indoor termal environment for natural ventilated residential buildings in Singapore, Energy & Buildings (2006).

[13] H. Altan, I. Ward, J. Mohelnikova, F. Vajkay, An internal assessment of the thermal confort and daylighting conditions of a naturally ventilated building with an active glazed façade in a temperate climate, Energy & Buildings (2008).

[14] Churchill, S. W., & H. H. S. Chu, Correlations Equations for Laminar and Turbulent Free Convection from a Vertical Plate, Int. J. Heat Mass Transfer, Vol. 18, 1975, p. 1323.

[15] Alan J. Chapman, Transmisión de Calor, Editorial Bellisco, 3º edition.

[16] Louis C. Burmeister: Convective heat transfer, Department of Mechanical Engineering, Univeristy of Kansas, 2º edition.

[17] Spanish technical code for building, Volume DB-HE: Save energy.