NUMERICAL SIMULATION OF THE THERMAL STORAGE TANK OF AN ADSORPTION AIR CONDITIONING SYSTEM POWERED BY SOLAR ENERGY

Cristian Adolfo, cristian_adolfo@hotmail.com
Antonio Pralon Ferreira Leite, antpralon@yahoo.com.br
Laboratório de Energia Solar- Universidade Federal da Paraíba, João Pessoa-PB, Brazil

Belkacem Zeghmati, zeghmati@univ-perp.fr
Mathematical and Physical Laboratory of the Energizing Mechanics Systems- University of Perpignan Via Domitia, France

Abstract. *The environmental conscience is bringing up significant methods and techniques to save energy and substitute the fossile fuel resources by renewable ones. The adsorption air conditioning system at LES-UFPB, has been projected by this point of view, as it's powered by solar energy. One of its components is a heat storage tank, which supply the adsorbers with water at high temperatures and also pre-heat the water flow before it goes to the solar collector field. The main goal of this article is to study numerically the effect of inlets and outlets flow on velocity and temperature fields inside this tank. The numerical computation was performed using the Fluent software. A grid independence was carried out and results of velocity and temperature fileds are presented.*

Keywords*: CFD, heat storage tank, bi-dimensional, temperature field, velocity field.*

1. INTRODUCTION

The heat storage tank is an important component of the adsorption refrigeration system at LES- UFPB, João Pessoa, which is powered by solar energy and also supplied by a gas heater, when needed. Its function is to supply the activated coal and methanol adsorbers of the refrigeration system by storing water at high temperatures so that it can transfer the necessary heat for the regeneration process of these elements. Thus, the tank receives hot water from the solar collector field that is set on the roof of the room where the system is assembled. When the temperature reached at the solar collectors is not sufficiently high, a small gas heater is utilized to complete the warming. The tank is cylindrical, supported by its base, and it has two inlets and two outlets for the water flow.

The main goal of this paper is to analyze the temperature and velocity distribution inside the tank during the operation, since the inlet temperatures are varying throughout the day, as the solar radiation changes. The computational analysis has been done using the commercial software Fluent, and the meshes have been done in Gambit.

2. PROBLEM'S MODELING

The thermal storage tank has been represented in a bi-dimensional scheme, as shown in Fig. 1.

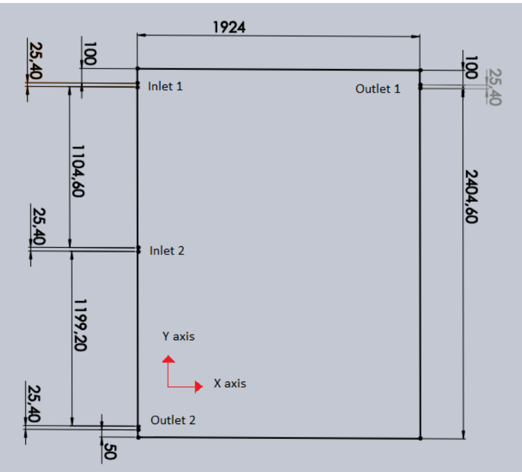


Figure 1. Heat storage tank bi-dimensional scheme, with dimensions given in milimeters (mm).

As stated earlier, Fig. 1 shows two inlet and two outlet ports used for the water flow, which are actually 1 inch pipes, represented in the draft as 25.4 mm sized lines.

2.1 Hypothesis

In this research, it's assumed that the transfers are laminar, bi-dimensional and the flow is unsteady. The physical properties of water, assimilated to a Newtonian fluid, are constant with the exception of its density, which is estimated by the Boussinesq approximation. The viscous dissipation and the radiative transfers by the intern surfaces of the tank´s walls are negligible. Also the heat transfer along the walls is negligible compared to the heat transfers in a perpendicular direction of the wall. Finally, the effects of gravity were ignored in this study.

2.2 Governing equations

The heat transfer and the momentum transfer inside the water storage tank are based on the Navier-Stokes and the energy equations. Fluent uses Eq. (1), the Continuity Equation, Eq. (2), the Momentum Equation for X axis, Eq. (3), the Momentum Equation for Y axis, and Eq. (4), the Energy Equation, to solve this kind of problem.

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

$$\rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y}\right) = -\frac{\partial p}{\partial y} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

$$\rho\, c_p \left(\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y}\right) = k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \tag{4}$$

Where ρ is the fluid density, c_p is the fluid specific heat, u is the fluid velocity in the X axis, v is the fluid velocity in the Y axis, p is the pressure, μ is the fluid viscosity, k is the fluid thermal conductivity and T is the temperature. To start solve this equations, the user has to put initial and boundary conditions, discussed in the following.

2.3 Initial conditions

It's assumed that the tank's initial temperature is slightly lower than the Inlets temperatures, in order to decrease the time the tank would take to get to the steady state. It's been taken the temperature of $T_{Initial}$ = 330 K. The ambient temperature has been taken $T_{ambient}$ = 300 K. Furthermore, the initial velocities are nil. The initial conditions are shown in Eq. (5) and Eq. (6).

$$u(x, y, t = 0) = v(x, y, t = 0) = 0 \tag{5}$$

$$T(x, y, t = 0) = 330\ K \tag{6}$$

2.4 Boundary conditions

2.4.1 Inlet ports modeling

The Inlet 1, also called the Primary Inlet, is at the top of the tank and it is the one that receives water flow from the solar collectors. In Gambit, this shall be set up as "velocity inlet" condition, whose temperature and velocity must be prescribed. Its temperature may reach 100ºC at the hottest moment of the day, about 14h for João Pessoa in a clear day. An experimental result for temperature during a common day should be used as input for the program, so that the air conditioner dynamics can be analyzed. For the start, and all over this article, let's say the Inlet 1 temperature is $T_{Inlet\ 1}$ = 350 K. Its velocity, otherwise, is calculated from the volumetric flow rate Q and the pipe's diameter D by the expression $u_{Inlet\ 1} = 4\ Q / \pi\ D^2$. As known from a previously study, where the optimal volumetric flow rate found was Q = 0.0004 m^3/s, thus, $u_{Inlet\ 1}$ = 0.8 m/s.

The Inlet 2, in turn, also called the Secondary Inlet, is at half-height of the tank. After the water flow has passed by all the adsorbers, it returns to the tank by this entry, so that the flow may be pre-heated just before it goes to the solar collectors again. The "velocity inlet" condition was also set for this port, with the same value for velocity $u_{Inlet\ 2}$ = 0.8 m/s, due to the fact that the whole flow returns to the tank. It's known that its temperature is quite lower than the Primary Inlet's due to the heat losses over the adsorbers, and for the study it has been taken $T_{Inlet\ 2}$ = 340 K.

For these Inlets, the software Fluent uses the boundary conditions like shown in Eq. (7), Eq. (8) and Eq. (9).

$$T(x, y, t) = T_{Inlet\ 1,2} \tag{7}$$

$$u(x,y,t) = u_{Inlet\ 1,2} \tag{8}$$

$$p(x,y,t) = -\frac{1}{2}\,\rho\, u_{Inlet\ 1,2}{}^2 \tag{9}$$

2.4.2 Outlet ports modeling

The Outlet 1, also called Primary Outlet, is a pipe at the top of the tank that takes water flow to the adsorption system. If the temperature measured in this pipe is not enough high for the proper regeneration of the adsorbers, there's a gas heater that supplies the heat that lacks. Else, the Outlet 2, also called Secondary Outlet, is meant to take the water flow, pre-heated, back to the solar collector field, where it can get heated another time.

These two outlets shall be set up in Gambit as "pressure outlet" condition, whose temperature and velocity are not prescribed. In fact, these are two of the results wanted by this work and that must be studied. For these Outlets, Fluent uses the following boundary conditions, shown in Eq. (10), Eq. (11) and Eq. (12).

$$\left.\frac{\partial T}{\partial x}\right|_{x=0;L} = 0 \tag{10}$$

$$\left.\frac{\partial u}{\partial x}\right|_{x=0;L} = \left.\frac{\partial v}{\partial x}\right|_{x=0;L} = 0 \tag{11}$$

$$\left.\frac{\partial p}{\partial x}\right|_{x=0;L} = 0 \tag{12}$$

2.4.3 Wall modeling

The walls have their own condition in Gambit, so-called "wall". There, it can be applied boundary conditions such as non-slip condition, shown in Eq. (13) and the Fourier Boundary Condition, shown in Eq. (14) and Eq. (15), as wanted, where k_{water} is the thermal conductivity of the water, T_{amb} is the ambient temperature, t_{ins} the thickness of the insulating, k_{ins} the thermal conductivity of the insulating and h_{cv} is the heat transfer coefficient by convection between the tank's surface and the external environment.

$$u(x,y,t) = v(x,y,t) = 0 \tag{13}$$

$$-k_{water}\left.\frac{\partial T}{\partial x}\right|_{x=0;L} = \frac{T(x,y,t) - T_{amb}}{\frac{t_{ins}}{k_{ins}} + \frac{1}{h_{cv}}} \tag{14}$$

$$-k_{water}\left.\frac{\partial T}{\partial y}\right|_{x=0;H} = \frac{T(x,y,t) - T_{amb}}{\frac{t_{ins}}{k_{ins}} + \frac{1}{h_{cv}}} \tag{15}$$

Evidently, it's expected that heat losses occur along the tank's surface, probably preventing the tank from reaching the average temperature of its inlets. However, it's also expected the magnitude of these heat losses to be relatively small, due to the fact the tank has an efficient insulating covering it. There is expanded polyethylene, in a total thickness of t_{ins} = 50 mm, coating the walls and the top of the tank. For this study, it's assumed that the base is covered by this material, too. The thermal conductivity for the insulating was considered quite bigger than its real value, k_{ins} = 0.5 W/m K, in order to increase the heat losses and allow a more relevant view about this heat transfer. The ambient temperature was set as T_{amb} = 300 K, and the heat transfer coefficient as h_{cv} = 30 W/ m² K.

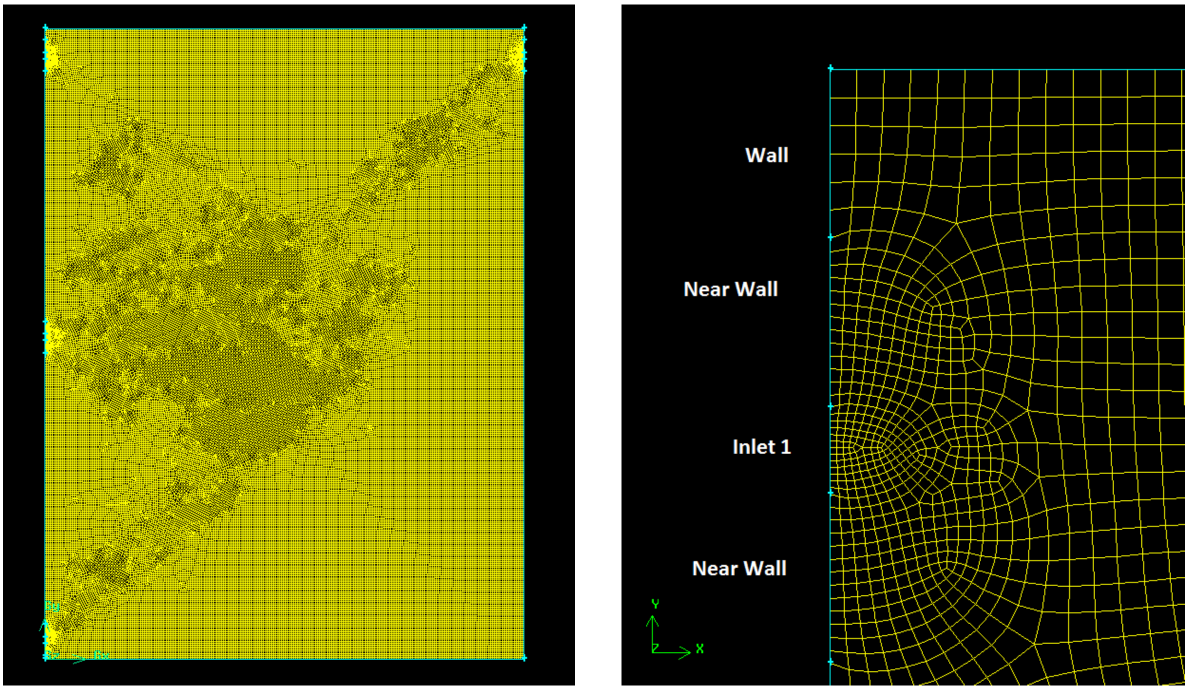
3. NUMERICAL METHODOLOGY

The Eq. (1-3) associated to the initial and boundary conditions [Eq. (4-7)] are solved using the Fluent software.

3.1 Mesh construction

The finite elements mesh is an important part of the Computational Fluid Dynamics (CFD) analysis. The more smoothed the mesh is, the more close to reality the results may be, but the more is the computational time to obtain them. On the other hand, the less refined the mesh is, the less is the computational time to run the simulation, but the less certain are the results. A mesh independence study often solves this conflict, as did bellow.

The mesh has been made in the software Gambit from the draft showed in Fig. 1. It's desirable the mesh to have better refinement near to the tank's inlet and outlet ports, in order to take account of the velocity and temperature gradients, which are more important in these zones. Thinking about it, it's been made some little pieces of wall, right above and right below the tank's inlets and outlets, so that, these could be refined separated from the wall by its own. These pieces were called "near walls", because they're near to the entries. The final result of one of the meshes done is shown in Fig. 2.



(a) (b)

Figure 2. Finite elements mesh created (a) and the detail of the refinement around the inlets of the tank (b).

Figure 2 (b) shows the refinement mentioned previously, and the same has been done at the walls, where smooth elements are substantial to represent the velocity gradient that may appear around them.

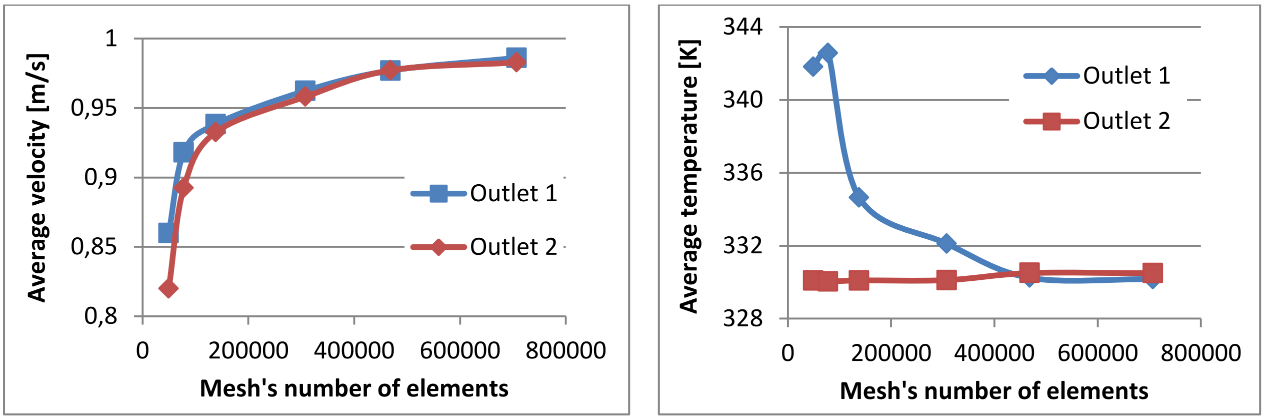
3.2 Grid size sensitivity

A mesh independence study is based on the comparison of some relevant results for different meshes, with a certain number of finite elements each. The smoother the mesh is the closer to reality the results found should be. There's a moment when the solution is over the mesh's refinement and, no matter how smoother the mesh takes, no significant difference can be noticed about the results. That is the refinement level taken for the study. A total of six meshes have been made for the mesh independence study, each one with its elements' smooth level, given in mm, as shown in Tab. 1.

Table 1. Characteristics of the six meshes studied, with constructive elements' sizes given in mm.

	Mesh 1	Mesh 2	Mesh 3	Mesh 4	Mesh 5	Mesh 6
Inlet and Outlet Ports	3,0	2,0	1,5	1,0	0,8	0,5
Near Walls	5,0	4,0	2,5	2,0	1,5	1,0
Walls	10,0	8,0	6,0	4,0	3,3	2,5
Tank's Body	20,0	15,0	12,0	8,0	6,5	5,0
TOTAL OF ELEMENTS	49.609	77.838	138.425	307.871	468.804	706.971

All these meshes have been run in a single time-step simulation, sized as 10 s, while the temperature and velocity fields were monitored, such as the average temperature and velocity in both Outlets. Figure 3 shows a graphic comparing the Outlet's velocity and temperature for each mesh, after 10 s of flow.



(a) (b)

Figure 3. Average velocity (a) and temperature (b) measured at the Outlets according to the mesh refinement.

As expected, the solutions are converging after certain refinement level, which means the number of elements has been properly taken for the case. In fact, the four first meshes have enough variation between their results. However, the results obtained by the meshes 5 and 6 are very similar. The Outlets' temperature in mesh 5, have both less than 0.02% of discrepancy than the same results for mesh 6. For velocity, this difference is of 0.93% for Outlet 1 and 0.57 % for Outlet 2. Thus, the Mesh 5 has been taken for the study, because it can produce results as good as a refiner mesh, but demanding much less computational time. The comparison between the temperature field can also proves the convergence of solution since Mesh 5's refinement level, as shown in Fig. 4.

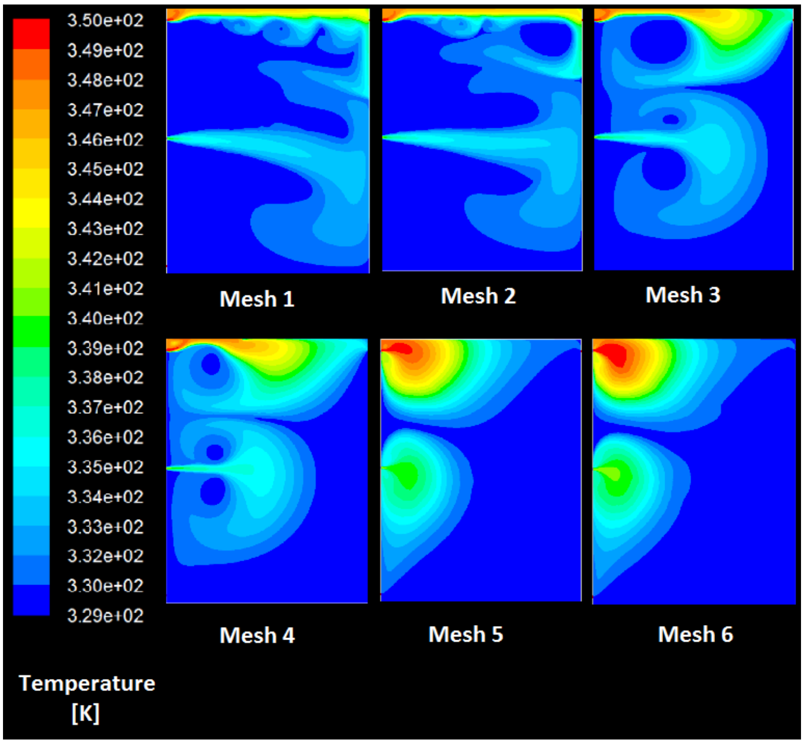
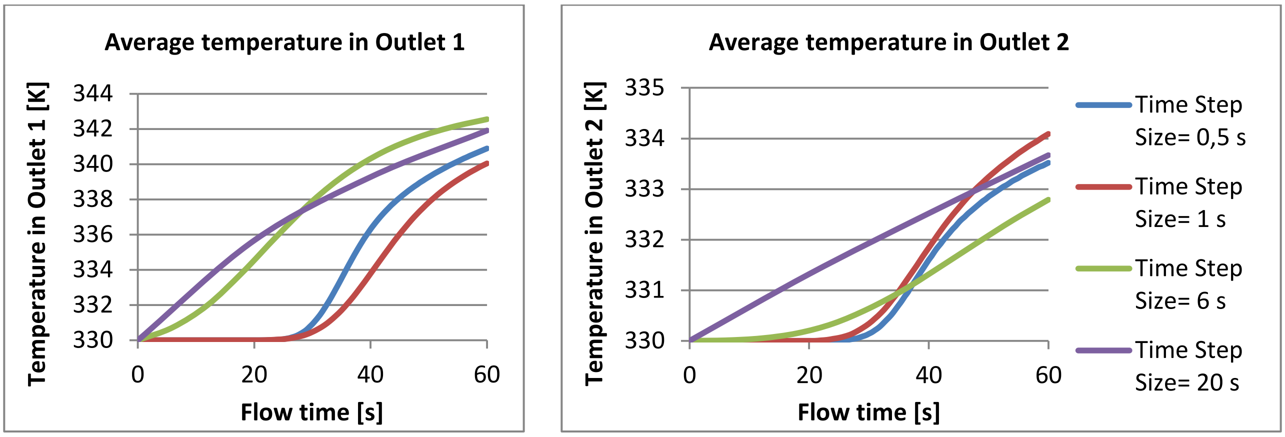


Figure 4. Temperature field obtained with the six different meshes after 10 s of flow.

4. TIME STEP SIZE STUDY

Another factor that should be analyzed when solving an unsteady case is the influence of the time step size chosen. The greater the time step size, the lesser time steps should be run by the simulation in order to reach a reference time, saving computational time. Despite of it, lesser is the accuracy of the solution on transition time periods. As have been made for the mesh independence, it's necessary to know the greater time step size from which its decrease wouldn't change relevantly the results. Taking the Mesh 5, it has been done simulations with time step sizes of 0.5 s, 1 s, 6 s and 20 s until a total flow time of 60 s. The comparison for the Outlets' temperatures is shown in Fig. 5.



(a) (b)

Figure 5. Temperature's evolution of Outlet 1 (a) and of Outlet 2 (b) for each Time Step Size studied.

It's noticed in Fig. 5 that the time step size (TSS) adopted is such relevant to the results for the first minute of flow. There's clear that, with the increase of this temporal property, the earlier the Outlets are affected by the high temperatures coming from the Inlets. This can be explained due to the fact that, with greater time step sizes, the more difficult is to the program to interpret the retention of the flow, caused by the water's viscosity forces. When it's assumed TSS = 6 s, for example, the program realizes that, with one time step, the warm flow from the Inlets could passes through a total length of $L = time \times u_{inlet}$ = 6 [s] × 0.8 [m/s] = 4.8 m distant from the Inlet, ignoring part of the viscous forces in the moments between t = 0 and t = 6 s. Of course that is not real, as the two smallest time step sizes, 0.5 and 1 s, shows. The results of those TSS's, despite of it, are pretty closer between themselves.

However, it can be also noticed that, despite of the TSS chosen, the solutions seem to converge at the end of the first 60 s of flow. Thinking about this, other analysis has been made: the time the flow takes to become steady about its energy. Cases with TSS of 1 s, 10 s and 30 s where compared until 20 minutes of flow, and the steady state is supposed achieved when the relation shown in Eq. (16) is obeyed for the average tank temperature, for the average Outlet 1 temperature and for average Outlet 2 temperature, with T as the temperature.

$$|T(t) - T(t + 30\ s)| < 3 \times 10^{-2}\ K \qquad (16)$$

Thus, the flow can be said as steady once when the increase of the tank temperature after an interval of 30 s is lesser than 0.03 K, i.e., lesser than 0.1% of the temperature value. The Tab. 2 shows the comparison for the time taken by the flow to become steady and the steady temperature reached according to the TSS adopted.

Table 2. Time taken to the flow become steady and T_{tank} reached at that moment.

	Time to steady flow	T_{tank} for steady flow
TS= 1 s	720 s	343,76 K
TS= 10 s	750 s	343,67 K
TS= 30 s	870 s	343,07 K

With the increase of the TSS, is clearly noticed the tendency for greater time for the flow to become steady and the decrease of the final average temperature at the tank. However, the 10 s time step size showed its results very close to the shown by the 1 s one, while the 30 s one didn't.

Therefore, it's relevant to emphasize that, when the aim of the study is the steady flow and the solutions after long flow times, TSS such as 10 s should be adopted. However, when its goal is to analyze the transient interval of the flow, paying attention of the changes of velocity and temperature fields, just like this work, lower TSS, as 1 s should be taken.

5. RESULTS AND DISCUSSIONS

As stated previously, Mesh 5 and 1 s TSS have been chosen for running the heat storage tank simulation in Fluent. This software shows results for velocity and temperature fields, in order to allow the validation of the programming.

5.1 Velocity field

Due to the low velocities and the laminar flow, the velocity field is quickly become into steady form. Figure 6 shows its evolution during time, where can't be noticed such variations, even between distant flow times like 10 s and 7 minutes. It's noticed that the inlet flow spreads radially into the water tank as a jet flow in quiescent water. Consequently, the flow's velocity decreases until the value of the middle of the tank's flow. The velocity field in the vicinity of the outlet ports illustrates cells created by the interaction between the tank's wall and the flow coming from the other side of the tank.

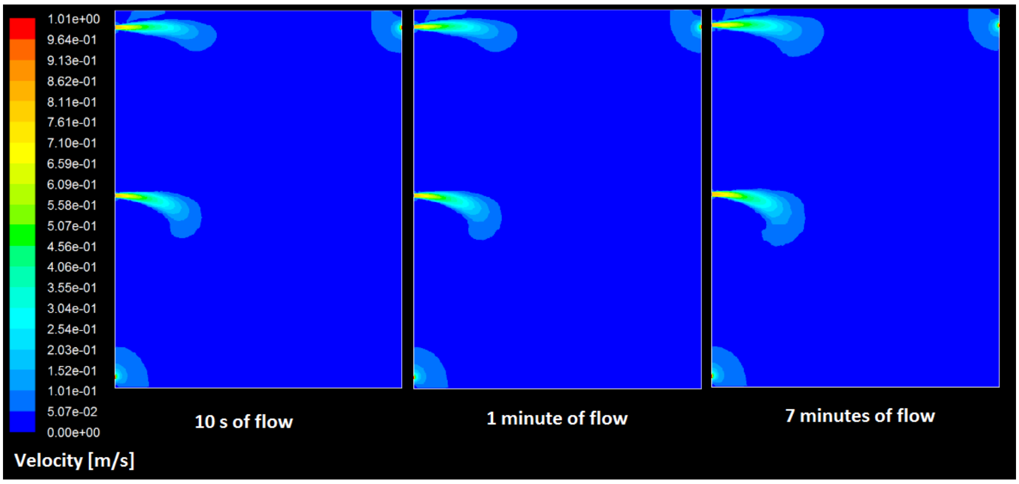
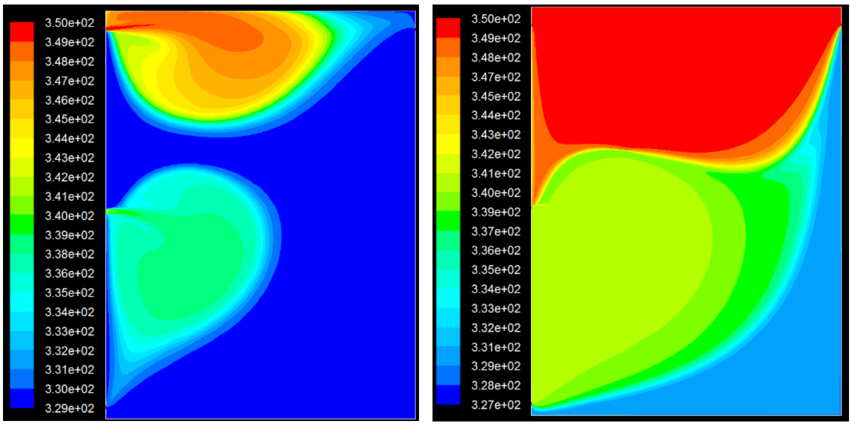


Figure 6. Evolution of the velocity field according to the flow time.

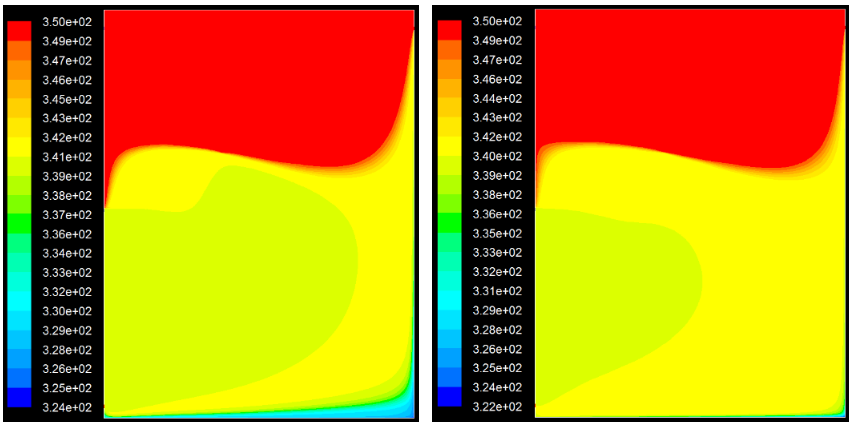
5.2 Temperature field

About the temperature, however, the problem has taken about 12 minutes in order to become steady. The Fig. 7 and Fig. 8 show the temperature field evolution.



(a) (b)

Figure 7. Temperature fields after 30 s of flow (a) and after 3 minutes (b), left-aligned scales in [K].



(a) (b)

Figure 8. Temperature fields after 12 minutes of flow (a) and after 20 minutes (b), left-aligned scales in [K].

Figure 8 shows that the temperature field doesn't change relevantly after 12 minutes of water circulation in the storage tank. In fact, in the time interval between 12 and 20 minutes, the average tank's temperature varies about 0.02%. Despite of it, the Outlets' temperatures have increases a bit bigger than this, but yet unexpressive. There is important to notice that the Outlets temperature increase being greater than the tank's average is possible because of the heat losses around the walls, which brings down the temperature near to them. As flow time passes, the lesser is the minimal temperature of the tank, as shown in the scales of the Fig. 7 and Fig. 8. The monitored average tank's temperature is shown in Fig. 9 for the first 15 minutes of flow.

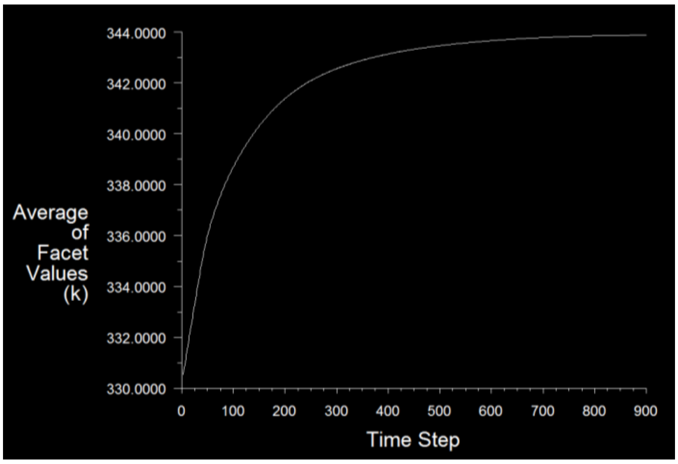


Figure 9. Average tank's temperature variation, given in Kelvin, for each time step as 1 s of flow.

The final average temperature, at steady flow, for the Outlet 1 is 349.54 K and for Outlet 2 is 340.3 K. This is not much interesting once the main goal of the tank's use is not clearly observed. The heat loss between the Inlet 1 and Outlet 1 is very small, as expected. However, the pre-heat gained between the Inlet 2 and the Outlet 2 is quite little, maybe not enough to make up for the losses. There can probably be caused by the weak homogenization of the water tank, due to all the Inlets being at the same side of the tank, and also by the Inlet 2 and Outlet 2 being at the same side, which in fact shorten the flow through them. The side changing of one of these could be a factor to improve the water homogenization.

Nevertheless, it is pretty important to notice that all Inlets' temperatures are fictional, as well as the thermal conductivity adopted for the wall insulating, which has been taken higher to magnify results about heat losses.

Furthermore, when the tank and the air conditioning system are actually working, it can be said the tank is most of time just like the steady results obtained, as soon as the Inlets' temperatures are always changing gradually but very slowly during day-time. Temperature fields as shown in Fig. 8 could probably represent better what really occurs.

6. CONCLUSIONS

The temperature and the velocity fields in a heat storage tank of an adsorption refrigeration system at LES- UFPB have been numerically investigated. Computations have been performed using Fluent software. On the basis of the present analysis it is first concluded that the inlet flow spreads radially into the water tank as a jet flow in quiescent water. The laminar flow and all the flow theory adopted seem perfectly fit to the case studied, showing that the velocity field takes no long time to become steady and that the temperature field can be properly observed for all instants.

Some factors as the fluid confidence at the bottom of the tank and the shortened flow between Inlets and Outlets must be prevented in order to improve the water homogenization, increasing the tank's efficiency. Also, it's concluded that there is no temperature gradient along the radial direction, as well as it hardly influence the Outlets' temperatures.

When experimental results have been made, surely the calculation program defined here will be quite significant to improve the mathematical model and validate its results, in order to have a good tool to predict and to improve the air conditioning system efficiency.

7. REFERENCES

LEITE, A. P. F. et al. A hybrid solar-gas conditioning system based on Adsorption and chilled water storage. World Renewable Energy Congress, Sweden, 2011.

Fluent (ed.). Fluent user's guide, Fluent Incorporated, New Hampshire, 2006.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.