

THREE-DIMENSIONAL NUMERICAL SIMULATION OF HORIZONTAL CYLINDRICAL RESERVOIRS COUPLED WITH SOLAR COLLECTORS

Darci Luiz Savicki, darcilus@mecanica.ufrgs.br
Horácio A. Vielmo, vielmoh@mecanica.ufrgs.br
Department of Mechanical Engineering – UFRGS. Rua Sarmento Leite, 425. 90050-170 - Porto Alegre, RS, Brazil

Arno Krenzinger, arno@mecanica.ufrgs.br
Mechanical Engineering Graduate Program – UFRGS. Rua Sarmento Leite, 425. 90050-170 - Porto Alegre, RS, Brazil

Abstract. *A numerical analysis of the thermal and hydrodynamic behavior inside cylindrical horizontal reservoirs, coupled with solar collectors, is presented. The mathematical model is solved using the Finite Volume Method in structured mesh by a proper three-dimensional computational code. By numerical simulations, it was possible to study the transient operation of the thermal reservoirs, that involves the convective movement of the fluid and the stratification of the temperature. Calculations with different inlet configurations were carried out to illustrate the varying behavior of the thermal conditions in a solar store. Through the simulation of the reservoir with inlet and outlet fluid it was verified that the use of the flat baffle placed opposite to the inlet jet, and mainly the adequate choice of the height of this jet, makes possible to get a better thermal stratification.*

***Keywords**: Natural Convection, thermal reservoirs, numerical simulations, finite volume method*

1. INTRODUCTION

In the presence of a gravitational field, the fluid specific mass variation due to a non-uniform temperature field determines the appearance of buoyancy forces. The resultant movement of the fluid, well-known as natural convection, is very important in many technical applications, such as heating or cooling devices, solar collectors, thermal reservoirs, processes of solidification and crystal growth, among others. The optimization of the thermal energy storage devices is the focus in many systems characterized by a delay between the time production and its consumption. One of the most illustrative cases is the solar water heating system, where a good performance of the thermal reservoir means a considerable increase in the whole global efficiency of the installation. Basically, a solar water heating system is composed by the solar collectors, a thermal reservoir and a cold water box, as it shown in Fig. 1.

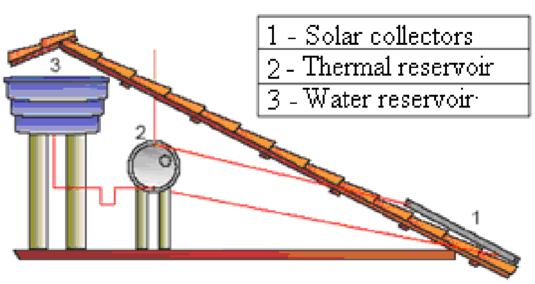

Figure 1. Illustration of a solar water heating system.

A desirable characteristic of such systems is good temperature stratification in the reservoir, which increases its efficiency. The temperature stratification in the reservoir depends on the velocity and the mass flow through the reservoir, as well as the difference between the inlet and outlet temperatures of the collectors. The solar collector efficiency decreases with reduction of the difference between these temperatures values, in way that a good efficiency requires the preservation of the thermal stratification. The temperature stratification can be preserved if the velocity values are low, and thus only laminar flows are allowed. The main problem of the mass flow is due to the position and the speed of the jet. Several works advises to a deviation of the inlet jet to reduce its velocity, forcing the dispersion of the flow next to the reservoir walls.

There are many theoretical and/or experimental works which approach the natural convection problem inside a rectangular or cylindrical cavity. However, in the numerical simulations, usually is considered some type of symmetry that allows a 2D numerical simulation only. Eames and Norton, 1998, presented a numerical simulation and an experimental study for the cases where the jet temperature is variable. It was shown that for the thermal stratification preservation, the velocity inside the reservoir must be low. A better thermal stratification can also be obtained applying different inlet heights.

Alizadeh, 1999, analyzes (numerically and experimentally) the thermal stratification in horizontal reservoirs simulating only the load processes (cold water inlet) and discharge (hot water outlet), as well as the influence of different types of diffusers (straight, conical) in the thermal stratification.

Oliveski, Krenzinger and Vielmo, 2000, developed a numerical-experimental study of the temperature and velocity fields for transient conditions in hot water reservoirs with a good agreement to the numerical solutions with the proper experimental results.

Shah and Furbo, 2003, simulated the influence of entrance jet on the temperature stratification in vertical reservoirs, using Fluent 5.5. For the high mass flow rates, around of 10 liters per minute, a better thermal stratification was obtained with the use of a flat plate opposite to the inlet jet.

Consul et al., 2004, performed a three-dimensional numerical simulation of cylindrical horizontal thermal reservoirs. With a multiblocks algorithm, simulations for load and discharge process were made (hot water consumption), and investigated the influence of the outflow on the thermal stratification. They also shown the potentiality of the parallel computation allied to the multiblocks technique. By dividing the domain into six blocks, and using six PCs, they reached a time factor approximately 5 times in comparison to the one PC case.

Just a few numerical studies works approach the natural convection problem for geometries which demand three-dimensional solutions. One of the reasons is due to the increase of the linear algebraic equations system and consequently the computational time increase. This work presents a three-dimensional numerical study of the temperature and velocity fields in a cylindrical thermal horizontal reservoir subject to heat loss for the environment due to a third kind boundary condition. This is a typical situation in solar systems of water heating.

2. MATHEMATICAL MODEL

Figure 2 presents the geometry and the dimensions of the reservoir analyzed in the present work.

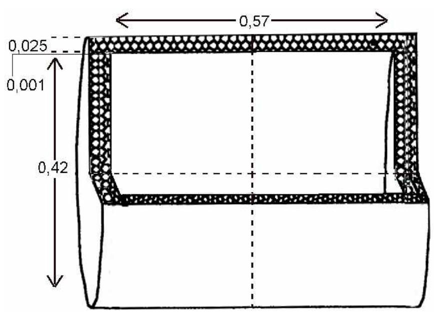

Figure 2. Dimensions for the simulated cylindrical horizontal reservoir (in meters).

The inner metallic wall (copper) and the insulation (polyurethane) thickness values are the same in both radial and axial directions, which are e_{pm}=0,001m and e_{iso}=0,025m, respectively. In the numerical simulations, it was considered, as initial condition, a prescribed temperature for whole the reservoir. The environment temperature and the heat transfer coefficient, h=8W/m^{2}K, (combined convection and radiation) are also imposed as boundary conditions.

2.1. Mathematical equations

In cylindrical coordinates, applying the Boussinesq approach, the problem can be described as follow:
Continuity Equation

$$\frac{1}{r}\frac{\partial\left(rV_r\right)}{\partial r}+\frac{1}{r}\frac{\partial V_\theta}{\partial\theta}+\frac{\partial V_z}{\partial z}=0 \tag{1}$$

Momentum Equations
Radial direction:

$$\rho_\infty\left(\frac{\partial V_r}{\partial t}+V_r\frac{\partial V_r}{\partial r}+\frac{V_\theta}{r}\frac{\partial V_r}{\partial\theta}-\frac{V_\theta^2}{r}+V_z\frac{\partial V_r}{\partial z}\right)= \\ -\frac{\partial p_H}{\partial r}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial\left(rV_r\right)}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 V_r}{\partial\theta^2}-\frac{2}{r^2}\frac{\partial V_\theta}{\partial\theta}+\frac{\partial^2 V_r}{\partial z^2}\right]+\rho_\infty\beta\left(T-T_\infty\right)g\cos\theta \tag{2}$$

Angular direction:

$$\rho_\infty\left(\frac{\partial V_\theta}{\partial t}+V_r\frac{\partial V_\theta}{\partial r}+\frac{V_\theta}{r}\frac{\partial V_\theta}{\partial \theta}-\frac{V_rV_\theta}{r}+V_z\frac{\partial V_\theta}{\partial z}\right)=$$
$$-\frac{\partial p_H}{\partial \theta}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial (rV_\theta)}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 V_\theta}{\partial \theta^2}+\frac{2}{r^2}\frac{\partial V_r}{\partial \theta}+\frac{\partial^2 V_\theta}{\partial z^2}\right]-\rho_\infty\beta\left(T-T_\infty\right)g\,sen\theta \tag{3}$$

Axial direction:

$$\rho_\infty\left(\frac{\partial V_z}{\partial t}+V_r\frac{\partial V_z}{\partial r}+\frac{V_\theta}{r}\frac{\partial V_z}{\partial \theta}+V_z\frac{\partial V_z}{\partial z}\right)=$$
$$-\frac{\partial p_H}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial V_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 V_z}{\partial \theta^2}+\frac{\partial^2 V_z}{\partial z^2}\right] \tag{4}$$

Initial conditions:

For all domain: $V_r=0;\ V_\theta=0;\ V_z=0$

Boundary conditions:

For the insulated region and the metallic wall: $V_r=0;\ V_\theta=0;\ V_z=0$

In the line r=0: $V_\theta=0;\ V_r=0;\ \frac{\partial V_z}{\partial r}=0$

In the symmetry plane (r, θ), where z=0: $\frac{\partial V_\theta}{\partial z}=0, \frac{\partial V_r}{\partial z}=0, V_z=0$

Energy Equation:

$$\rho_\infty c_p\left(\frac{\partial T}{\partial t}+V_r\frac{\partial T}{\partial r}+\frac{V_\theta}{r}\frac{\partial T}{\partial \theta}+V_z\frac{\partial T}{\partial z}\right)=k\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 T}{\partial \theta^2}+\frac{\partial^2 T}{\partial z^2}\right]+S^T \tag{5}$$

Initial conditions:

$T=T_{ini}$ in the fluid region (inside of the reservoir).

Linear variation of T between T_{ini} and T_{ext} in the thermal insulated region

Boundary conditions:

$\frac{\partial T}{\partial r}=0$ in the line r=0

$\frac{\partial T}{\partial z}=0$ in the symmetry plane, where z=0

$\frac{\partial T}{\partial \theta}=0$ in the symmetry planes rz, where $\theta=0$ and $\theta=\pi$

Prescribed conditions h_{ext} and T_{ext} at the boundaries $z=Z_{ext}$ and $r=R_{ext}$

For the reservoir simulation, considering that fluid is entering and outgoing from the reservoir, the calculus domain corresponds to a half reservoir, because the symmetry plane which cross its vertical diameter, as show in Fig. 3.

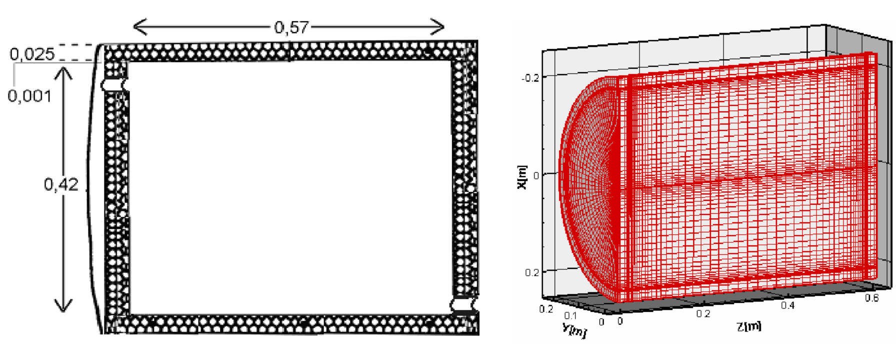

Figure 3. Calculus domain and computational mesh.

The defined boundary conditions for the inlet and outlet jets are presented as follow:

Inlet Jet Boundary

$$T = T_{in}$$
$$V_r = 0;\ V_\theta = 0;\ V_z = V_{in}$$

Outlet jet boundary

$$\frac{\partial T}{\partial z} = 0$$
$$V_r = 0;\ V_\theta = 0;\ V_z = V_{out}$$

where T_{in} is the inlet temperature (prescribed), and V_{in} is the velocity component in z direction, which is calculated from the water mass flow prescription (Q) at the reservoir inlet and the inlet cross section area inlet (A_{in}), given by:

$$V_{in} = \frac{Q}{A_{in}} \tag{6}$$

The outgoing jet velocity is calculated to satisfy the continuity equation, that is,

$$V_{out} = \frac{V_{in} A_{in}}{A_{out}} \tag{7}$$

As alternative for the outlet boundary condition, could be still used either the technique of the factor correction for the mass flow or the use of local parabolic condition, that would allow a better definition of the velocity field. However, considering that the outlet jet diameter is very lesser than the diameter of the reservoir, it was observed that with the direct prescription of the velocity in the outlet is reached a realistic behavior in the interest zone (bulk of the reservoir), with a lower time of CPU.

Cases were studied for the inlet jet at 2/3 of the reservoir diameter, and it was investigated the gain obtained with the use of the opposite flat baffle to inlet jet for the preservation of the thermal stratification. Moreover, were considered cases where the inlet jet was situated in the top of the reservoir and studied the gain gotten with the optimization of the position of this jet.

2.2. Method of solution

The proposed 3D algorithm is an extension of the general purpose Patankar (1980) sub-routines, which considers the control volume method for discretizing equations. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) is used for the velocity correction. For the linear algebraic equations system solution was used the TDMA. As interpolation function, a Power Law scheme was applied.

When discretizing the calculation domain for the fluid region in the radial and axial directions, non-uniform refinement is applied, thereby concentrating more volumes by the wall, in order to obtain information on the thermal and hydrodynamic limit layers. For numerical reasons, refinement is also applied by the central axis for the radial direction. One further volume is used in the radial and axial directions for the metallic wall, and two volumes for the thermal insulation layer. In the angular direction, discretization with uniform spacing is employed. Taking into account the volume referring to the metallic wall, and the two volumes referring to the thermal insulation layer, the grid is 45 x 60 x 60 in directions (r, θ, z).

The code was tested extensively for several simulations, producing solutions in agreement to theoretical and experimental results, available in literature. For the adopted considerations in the present work any result was found until the present moment. The equations are iteratively solved for each time step of 1 second, in the implicit form, until satisfy the convergence criterion $\frac{T^{n+1} - T^n}{T^n} \leq 10^{-6}$ for the thermal field and SMAX$\leq 10^{-6}$ (maximum mass residual) for the hydrodynamic field.

3. RESULTS

3.1 Inlet jet with temperature 8ºC above the outlet jet: the influence of the baffle plate

A situation of interest in solar heating systems is the process where the outlet jet is heated by the solar collectors, returning to the reservoir. A heating of 8ºC was considered. The flow rate is 1 liter per minute, causing an inlet velocity $V_{in} = 0,0154\,m/s$.

The field of initial temperature has a stratified profile as showed in Fig. 4. This profile was chosen for better representing a real situation in systems of solar heating.

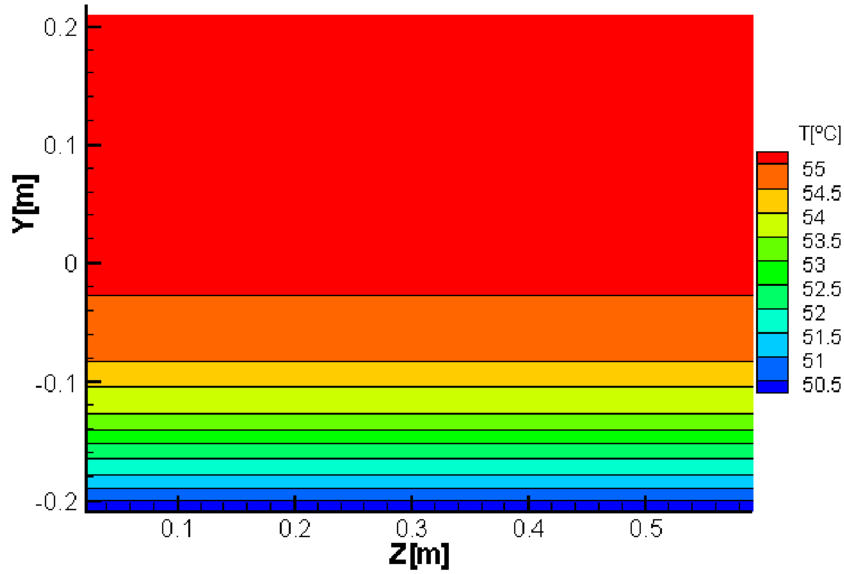

Figure 4. Stratified field temperature used as initial condition in numerical simulations.

Initially, for the case without flat baffle, where there is nothing obstructing the inlet flow, Fig. 5 show the temperature profiles for successive increments of 2 liters (2 minutes) of hot water to reservoir (and consequently equal extraction), until reaching a volume of 26 liters, what corresponds the circulation of 1/3 V through the reservoir, where V is its total volume.

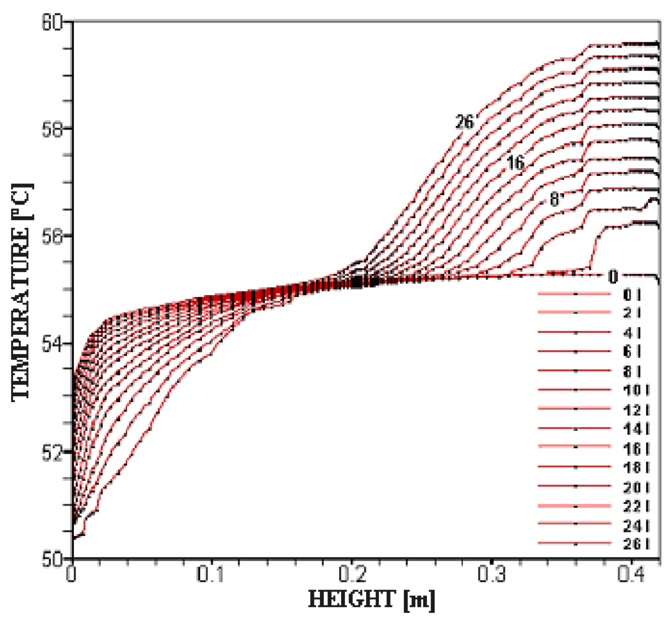

Figure 5. Successive profiles of temperature at the center line of the reservoir, for the case without flat baffle, with an inlet jet 8ºC above the outlet temperature, at 1 liter per minute.

In the Fig. 5 are indicated only the curves that corresponds to circulation of 0, 8, 16 and 26 liters. It is observed that in the top of the reservoir, the continuous hot water circulation implies in the formation of one layer with higher temperature. The opposite situation can be seen in the bottom.

For the profile that corresponds to the circulation of 26 liters, as showed in Fig. 5, Fig. 6 shows the associated temperature field in the plane of symmetry. In the same figure the components of the velocity field are shown.

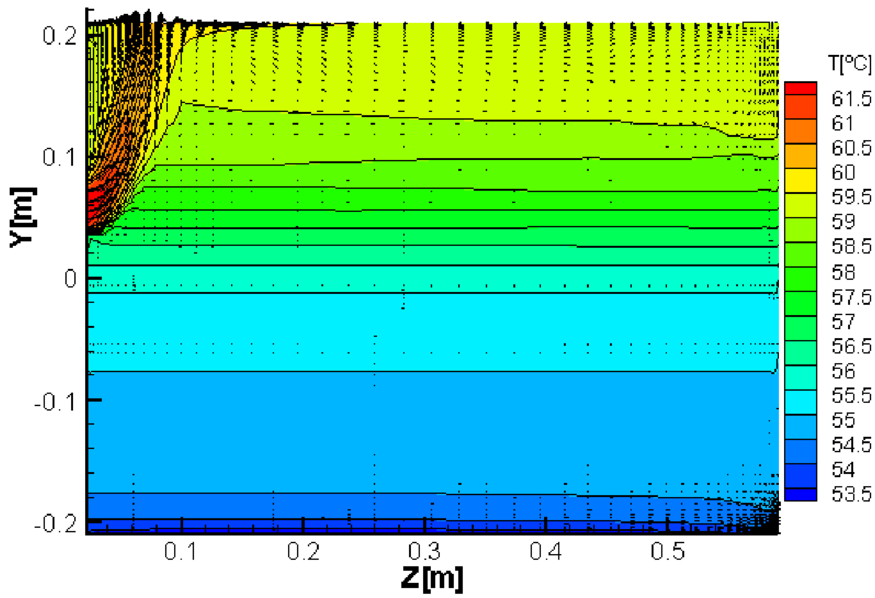

Figure 6. View of the temperature field, and velocity vectors, for the case without flat baffle, after the circulation of 1/3 of the reservoir volume.

It is verified that the inlet jet, due to higher temperature, moves upward by buoyancy force, causing some mixture. On the other hand, one perceives that the outlet jet has minor influence on the thermal stratification.

Figure 7 shows the case with flat baffle, keeping the same volumetric flow and temperature.

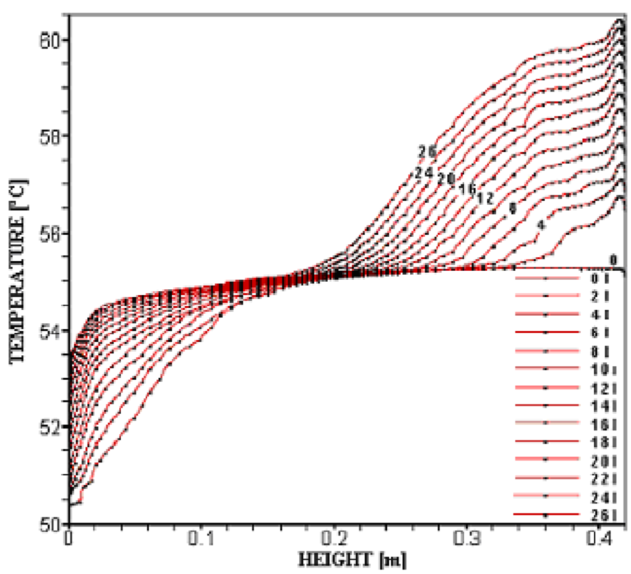

Figure 7. Successive temperature profiles at the center line of the reservoir, for the case with flat baffle, with an inlet jet 8ºC above the outlet jet temperature, at 1 liter per minute.

It is observed in Fig. 7 that whereas in the bottom the behavior is almost the same for both the cases (with and without flat baffle), in the top the behavior is significantly different. It is observed that in the top of the reservoir the temperature profiles show one “crest” didn’t visualized in the case without flat baffle. This crest represents a lower mixture between the hot water loaded and the water inside the reservoir. The top water temperature is almost the same of the inlet jet temperate.

Figure 8 shows for this case an overview of the temperature field and velocity vectors in the symmetry plane,, after the circulation of 26 liters of hot water.

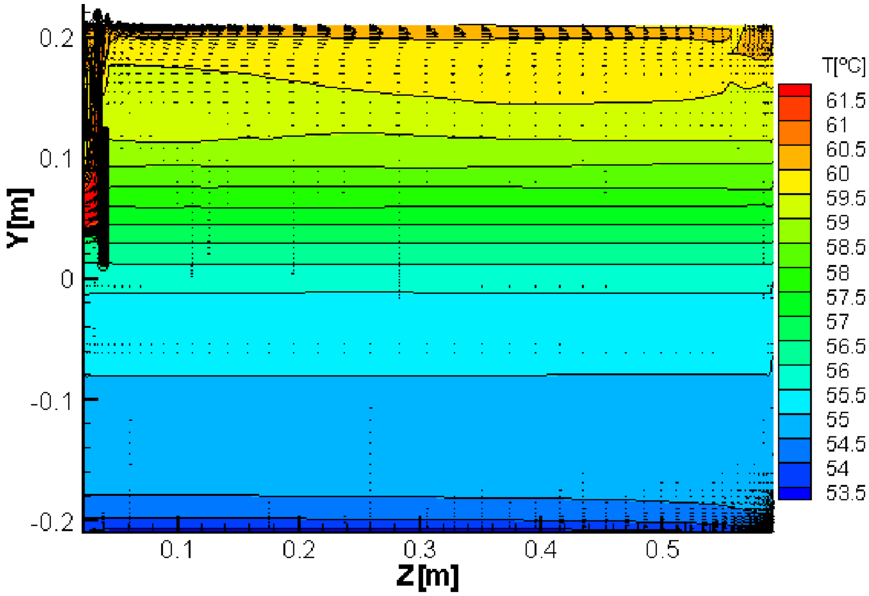

Figure 8. Overview of the temperature field and velocity vectors for the case with flat baffle, after the circulation of 1/3 of the reservoir volume.

It is possible to see differences between the cases with and without the flat baffle. For example, it is observed that the use of the flat baffle reduces the mixture of the inlet jet with the water of the reservoir.

To facilitate the comparison of the two cases, with and without flat baffle, Fig 9 shows a comparison of some temperature profiles.

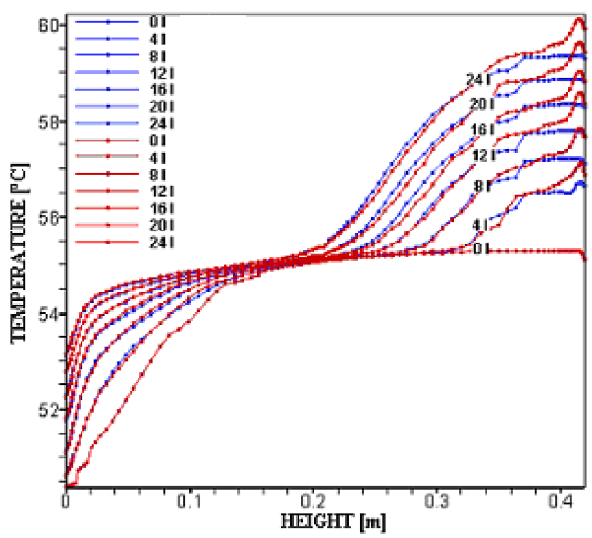

Figure 9. Comparison of the temperature profiles at the center line of the reservoir for the cases with and without flat baffle. (——— without flat baffle; ——— with flat baffle)

From the comparison between the cases shown in Fig. 9, it can be concluded that for the circulation of 1/3 of the reservoir volume the use of the flat baffle allows a temperature profile with bigger thermal stratification. Moreover, can be observed that, for the studied conditions, the use of the flat baffle enables the formation of temperature profiles with a higher temperatures crest in the top of the tank..

Due to initial temperature profile which was used in these simulations, presenting stratification at the bottom, in the phase of discharge of this stratified layer, due to imposed boundary condition, the temperature of the inlet jet is successively higher than in the previous instants.

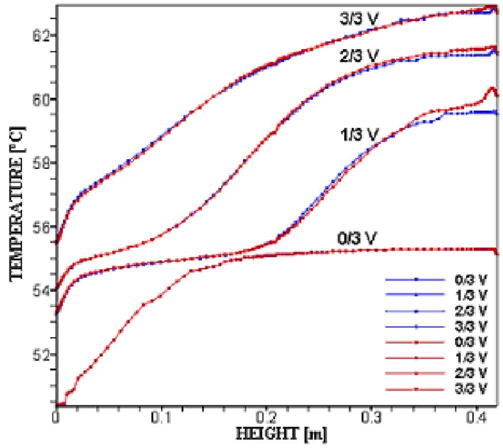

Figure 10. Comparison of the temperature profiles at the center line of the reservoir for the cases with and without flat baffle, after a complete circulation of one tank volume
(——— without flat baffle; ——— with flat baffle)

For the temperature profile used as initial condition (0/3V), after discharged the initial 1/3 V, the inlet jet temperature will decrease its variation, because from this moment, the outlet jet temperature presents low variation. Thus, the occurrence of these crests in the top of the reservoir disappear, as can be seen in the Fig. 10, curves “2/3 V” and “3/3 V”. Due to the characteristic of the initial temperature profile, in spite of the use of flat baffle, the stratification showed in curves "2/3 V" and "3/3 V" for both cases is practically the same. That is, the use of the flat baffle only promotes a better stratification when the temperature of the water in the bottom is stratified.

3.2 Inlet jet with temperature 8 °C above of the outlet temperature: the influence of the inlet jet position

The possibility of increasing the thermal stratification by changing the inlet jet position is now investigated. New simulations were performed using the inlet jet in the top of the reservoir for situations in which the temperature of the inlet jet is always higher than that existing in the interior of the reservoir. The initial temperature field is the same showed in Fig. 4. Figure 11 shows an overview of the temperature field and velocity vectors in the symmetry plane, after the circulation of 26 liters of hot water.

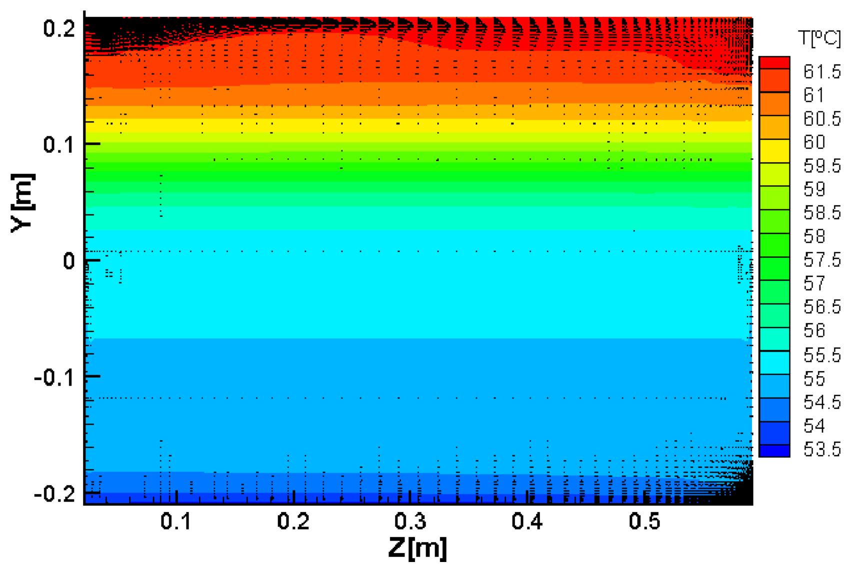

Figure 11. Overview of the temperature field and velocity vectors for the case with inlet jet in the top of the reservoir, after the circulation of 1/3 of the reservoir volume.

In the Fig. 12 a comparison of temperature profiles for the cases where the inlet jet is located at 2/3 of the diameter versus inlet jet located in the top of the reservoir.

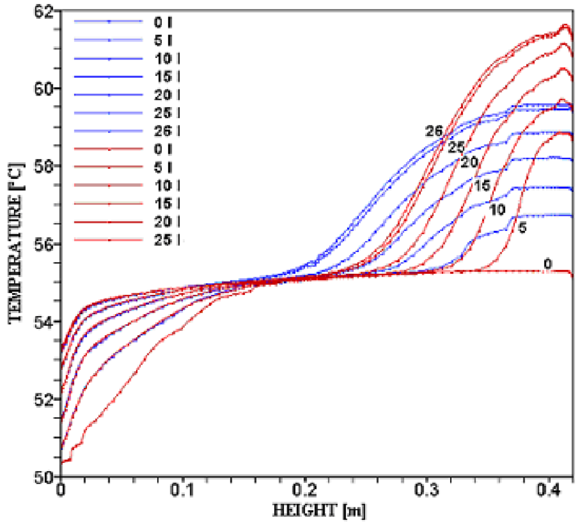

Figure 12. Comparison of the temperature profiles at the center line of the reservoir for the cases where the inlet jet takes place in 2/3 of the diameter versus inlet jet placed in the top - circulation of 1/3 of the total volume of the reservoir. (——— 2/3 of the diameter; ——— top of the reservoir)

From the comparison of the temperature profiles it can be observed that the inlet jet in the top of the reservoir determines a better thermal stratification. It is observed that for the case of the inlet jet in the top also occurs the formation of a high temperature crest., Moreover, it is observed that in the bottom does not have significant difference between the temperature profiles.

By the other hand, analyzing the curves "2/3 V" and "3/3 V" in Fig. 13, it is verified that the best stratification constructed in the initial phase was degraded along the time. At the end of a cycle (circulation of 1 V), the temperature profiles for two cases are similar.

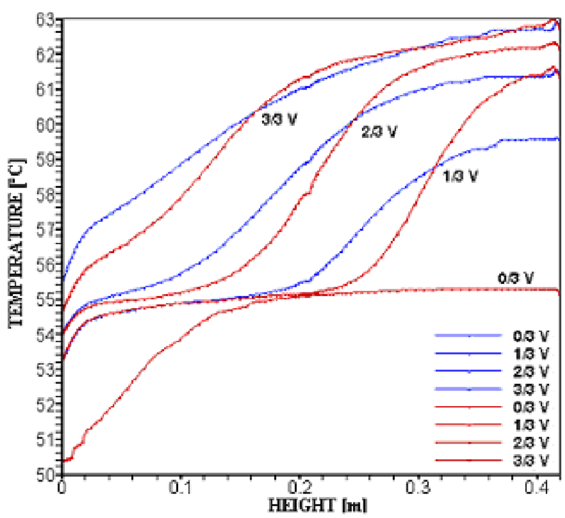

Figure 13. Comparison of the temperature profiles at the center line of the reservoir for the cases where the inlet jet takes place in 2/3 of the diameter versus inlet jet placed in the top – analysis for one cycle. (——— 2/3 of the diameter; ——— top of the reservoir)

4. CONCLUSIONS

In this work a study of the dynamics of the transient operation of cylindrical horizontal thermal reservoirs was performed, through a three-dimensional numerical simulation. It was considered one situation where the thermal reservoir is linked to solar collectors, and investigated the influence of the position of the inlet jet, as well as the influence of the use of a flat baffle in front of the inlet jet, on the thermal stratification. The results showed that the use of the flat baffle opposite to the inlet jet, as well as the position of this jet on the top of the reservoir, enable to obtain a better thermal stratification. On the other hand, when the inlet jet temperature remains constant for long time, the temperature of this jet becomes very close to the temperature of the existing water in the reservoir in same height, and thus the advantage almost disappears. Also it was demonstrated that the location and design of the inlet port are very important. Beyond the problem here considered, the computational program developed and validated in this work can be used for the study of an wide range of problems requiring a three-dimensional numerical simulation.

5. ACKNOWLEDGEMENTS

This work has been developed under the sponsorship of the Conselho Nacional de Desenvolvimento Científico e Tecnológico – CNPq, Brazilian government entity for scientific and technology development.

6. REFERENCES

Alizadeh, S., 1999, “An Experimental and Numerical Study of Thermal Stratification in a Horizontal Cylindrical Solar Storage Tank”, Solar Energy, Vol. 66, pp. 409-421.

Eames, P. C., Norton, B., 1998. “The Effect of Tank Geometry on Thermally Stratified Sensible Heat Storage Subject to Low Reynolds Number Flow”, Int. J. Heat Mass Transfer. Vol. 41, No. 14, pp. 2131-2142.

Oliveski R. C., Vielmo H. A., Krenzinger, A., 2003. “Cooling of Cylindrical Vertical Tanks Submitted to Natural Internal Convection”, International Journal of Heat and Mass Transfer, Vol. 46, pp 2015-2026.

Oliveski R. C., Vielmo H. A., Krenzinger, A. 2003. Comparison Between Models for the Simulation of Hot Water Storage Tanks, Solar Energy, Vol. 75, Issue 2, pp. 121-134

Patankar, S.V., 1980. Numerical Heat Transfer and Fluid Flow, McGraw Hill, New York.

Shah, L. J., Furbo, S., 2003. “Entrance Effects in Solar Storage Tanks”, Solar Energy, Vol. 75, pp 337-348.

Consul, R. Rodrigues, I., Perez-Segarra, C. D., Soria, M., 2004. “Virtual Prototyping of Storage Tanks by Means of Three-Dimensional CFD and Heat Transfer Numerical Simulation”, Solar Energy, Vol. 77, pp. 179-191.

5. RESPONSIBILITY NOTICE

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