23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

NUMERICAL STUDY OF THE INFLUENCE OF GEOMETRIC PARAMETERS ON THE AVALIABLE POWER IN A SOLAR CHIMNEY

Rodrigo Spotorno Vieira
Christopher Garcia
Jeferson Avila Souza
School of Engineering, Universidade Federal do Rio Grande, Av. Itália, km 8, Carreiros, 96203-900 – Rio Grande, RS, Brazil.
rodrigospotorno@yahoo.com.br, chrissefir@gmail.com, jasouza@furg.br

Luiz Alberto de Oliveira Rocha
Department of Mechanical Engineering, Federal University of Rio Grande do Sul, Rua Sarmento Leite, 425, 90050-170 - Porto Alegre, RS, Brazil
luizrocha@mecanica.ufrgs.br

Liércio André Isoldi
Elizaldo Domingues dos Santos
School of Engineering, Universidade Federal do Rio Grande, Av. Itália, km 8, Carreiros, 96203-900 – Rio Grande, RS, Brazil.
liercioisoldi@furg.br, elizaldosantos@furg.br

Abstract. *In the presented work, it is made a numerical study about the main physical principle of a solar chimney (SCPP – Solar Chimney Power Plant) and the influence of some geometric parameters on the available power in the SCPP. The height of chimney and the collector radius are changed according to Constructal Design. The generator area is maintained constant and the geometry is submitted to two degrees of freedom: R/H (ratio between the collector radius and collector inlet height) and* R_1/H_2 *(ratio between the chimney outlet radius and SCPP height). The main objectives are to test the applicability of the proposed numerical model in future studies of SCPP geometric optimization and to test the influence of the ratios R/H and* R_1/H_2 *on the available power of the device. For that it is considered an incompressible, turbulent, steady flow with mixed convective heat transfer in a two-dimensional and axisymmetric domain, similar to the one found in a solar chimney. The conservation equations of mass, momentum and energy are numerically solved using the finite volume method, more specifically with the FLUENT® software. The classical turbulence modeling (RANS) was used for the turbulence approach with standard model* $k - \varepsilon$*. For every studied cases, Constructal Design allowed the improvement of available power. The best performance was almost 80 % higher than the worst evaluated geometry. It was also seen that the ratio* R_1/H_2 *and R/H has a large sensibility in the available power.*

Keywords: *Constructal Design, Solar chimney, Geometric optimization, Numerical study*

1. INTRODUCTION

The rises of oil prices and energy demand combined with recent environmental restrictions have quickly raised the global demand for renewable energy sources. The solar energy is one of the most promising solutions, especially considering its technology, improvements and fast development. According to Li *et al*. (2014), solar energy is free from carbon emission, inexhaustible, sustainable, free from costs (in its basic form) and practically unlimited. Beyond that, it has an enormous potential, which could supply the world energy demand. Therefore, recently, with the decrease of fossil energetic resources and the increased effects of global warming, many countries have provided financial incentives for the development of solar power plants, including not returnable subsidy, long-term loans with reduced taxes and others. One of the alternatives proposed in the literature to supply those demands is the solar chimney power plant (SCPP), which is a device that turns solar energy into electricity.

The SCPP is composed by three essential parts: the collector, the chimney and the turbines. The chimney is a long cylindrical structure usually positioned in the center of a greenhouse collector, which is made of transparent glass or plastic film (Patel *et al.*, 2014). According to Jones (2011), the physical principle of operation of the solar chimney consists in the heating of an air layer, which is hold inside the collector. The heating is carried out by the incidence of solar energy in this film, where some of the radiant energy is going to be transmitted through the collector to the air between the film and the ground. The transmitted solar energy hits the surface of the ground and the latter absorb part of it, while the other part is reflected back to the collector. With modified wavelength, upon the incidence in the ground, the radiant energy suffers multiple reflections between the ground and the collector film, resulting in a raise in absorbed energy by the ground. Once the ground is warm, the air inside the collector also warms up and tend to move by natural convection to the chimney region, expanding though a turbine, which is located between the collector and the chimney (which has its outlet in an atmospheric pressure region). It is worth notice that, when the warm air moves up to the chimney, a depression region is generated in the collector periphery which causes the external air mass suction that exchanges heat with the ground and the collector, intensifying the thermal exchange in the collector region and increases the movement quantity of the air that expands through the turbine. This kinetic energy of the air turns the turbine blades, which move the energy generators. That way, one of the main objectives in solar chimney projects is to increase the movement quantity in the turbine region.

The energy harnessing technology of the solar chimney is well seen as one of the methods that can utilize solar energy in large scale. The solar chimney has no adverse effect on the environment, do not need water for refrigeration and has a low maintenance cost, unlike what occurs in other solar energy devices that require high financial cost technology (which makes it unfeasible in large scale). Those advantages makes the SCPP a promising technology for electricity generation in regions with abundant solar energy such as Brazil, for example (Li *et al.*, 2014).

Some studies were performed about this subject, such as the experimental investigation carried out by Zhou *et al.* (2007) where the environmental temperature, distribution of the temperature, ground temperature and air velocity in a solar chimney system were analyzed.

According to Dhahri and Omri (2013), the SCPP research can be separated in four groups: (1) the concept and validation of working principle, (2) simulation of device performance and structural optimization, (3) estimative of power capacity and (4) technical and economic evaluation. The concept and working principle of the SCPP are validated mainly through experimental devices and analytical calculations, one of the most successful among the presented is the Manzanares plant, in Spain (Haaf *et al.*, 1983). It is also found in literature the development of mathematical models to predict the system performance (as Pretorius and Kröger, 2006). Relative to numerical studies, some has been dedicated to understanding the behavior of velocity field, pressure and temperature in the SCPP and regarding the turbine performance (as Xu *et al.*, 2011); while others are developed aiming to maximize its useful available power in the turbine region by means a parametric variation, as in Patel *et al.* (2014) which developed a Computation Fluid Dynamics (CFD) three-dimensional model in the software ANSYS-CFX to perform twelve simulations to attain a tendency of the effect of the collector height, collector outlet diameter and the chimney diameter on the available power in the turbine.

In the presented study will be evaluated the use of a simplified numerical model, similar to the one used to Patel *et al.* (2014), in a two-dimensional axisymmetric domain in a incompressible, turbulent and permanent flow with heat transfer through mixed convection. The main advantage of using a simplified model is the possibility to minimize computational effort, allowing the achievement of a greater number of simulations. At all were attained 28 simulations allowing the evaluation of the collector height and chimney outlet diameter influence on the useful available power on the SCPP turbine. To do so, continuity, momentum and energy equations were numerically solved by the FLUENT software, which is based on the Finite Volume Method (FVM) (Patankar, 1980; Verteeg and Malalasekera, 2007). For the numerical approach of the turbulence is used the Reynolds Averaged Navier-Stokes (RANS) classical modeling which consists in the application of a time average operator in the conservation equations (Wilcox, 2002). To solve the closing problem of the turbulence is used the model of two differential equations $k - \varepsilon$ (Launder and Spalding, 1972).

2. THEORY

The time averaged conservation equations of mass, momentum and energy for the solution of turbulent, incompressible flows with heat transfer by mixed convection are given by (Bejan, 2013):

$$\frac{\partial \bar{v}_j}{\partial x_j} = 0 \qquad (j = 1, 2 \text{ and } 3) \text{ in } t \times \Omega \tag{1}$$

$$\rho_0 \left[\frac{\partial}{\partial t}\left(\overline{v_i}\right) + \frac{\partial}{\partial x_j}\left(\overline{v_i v_j}\right) \right] + \frac{\partial \bar{P}}{\partial x_j}\delta_{ij} - \frac{\partial}{\partial x_j}\left\{ \left(\mu + \mu_t\right)\left(\frac{\partial \overline{v_i}}{\partial x_j} + \frac{\partial \overline{v_j}}{\partial x_i} \right) \right\} + \rho_0 g_i \beta \left(\bar{T} - T_0 \right) = 0 \quad (i, j = 1, 2 \text{ and } 3) \text{ in } t \times \Omega \tag{2}$$

$$\frac{\partial \bar{T}}{\partial t} + \frac{\partial}{\partial x_j}\left(\overline{v_j T}\right) - \frac{\partial}{\partial x_j}\left\{ \left(\alpha + \alpha_t\right) \frac{\partial \bar{T}}{\partial x_j} \right\} = 0 \qquad (j = 1, 2 \text{ and } 3) \text{ in } t \times \Omega \tag{3}$$

where $(\bar{\ })$ represents the time average operator, ρ_0 is the fluid density at the reference temperature (kg/m³); β is the thermal expansion coefficient (K^{-1}); μ is the dynamic viscosity (kg/ms); λ is the bulk viscosity (kg/ms); υ is the kinematic viscosity (m²/s); α is the thermal diffusivity (m²/s); v_i is the velocity in the direction i, j = 1, 2 and 3 (m/s); x_i corresponds to the spatial coordinate, i, j = 1, 2 and 3 (m); P is the pressure (N/m); T is the temperature (°C or K); T_0 is a reference temperature (°C or K); g_i is the gravity acceleration in the direction i = 1, 2 and 3 (m/s²); δ_{ij} is the Kronecker delta; Ω is the spatial domain (m); and t is the time domain (s).

In Eq. (2) and Eq. (3) the turbulent viscosity (μ_t) and the turbulent diffusivity (α_t) are given by:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

$$\alpha_t = \frac{\upsilon_t}{\mathrm{Pr}_t} \tag{5}$$

To estimate the μ_t is necessary to obtain values of the turbulent kinetic energy (k) and its dissipation rate (ε) along the domain. For this, two additional equations are solved (Wilcox, 2002; Launder and Spalding, 1972):

$$\frac{\partial k}{\partial t} + \bar{v}_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial}{\partial x_j}\left[\left(\upsilon + \frac{\upsilon_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] - \varepsilon \tag{6}$$

$$\frac{\partial \varepsilon}{\partial t} + \bar{v}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\upsilon + \frac{\upsilon_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{\varepsilon 1}\frac{\varepsilon}{k}\tau_{ij}\frac{\partial \bar{v}_i}{\partial x_j} - C_{\varepsilon 2}\frac{\varepsilon^2}{k} \tag{7}$$

The constants employed to solve the additional transport equations, Eqs. (4) - (7), are shown in Tab. 1.

Table 1. Constants used in the k – ε model, Eqs. (4) - (7).

Constants	Values
C_μ	0.09
$C_{\varepsilon 1}$	1.44
$C_{\varepsilon 2}$	1.92
σ_k	1.00
σ_ε	1.30
Pr_t	1.00

The conservation equations that model the problem, Eqs. (1) - (3), as well as the differential equations of the turbulence k – ε model are solved by the Finite Volume Method (FVM), specifically using the software FLUENT (FLUENT, 2007). The solver is pressure based and all the simulations use the 2nd Order Upwind advection scheme and the SIMPLE method for the pressure-velocity coupling. More details about the FVM can be found in Patankar (1980) and Versteeg and Malalasekera (2007).

The numerical simulations were attained in a computer equipped with 6 processors AMD FX6300 with 3.5 GHz clock and an installed 4 GB RAM memory. The processing time for the simulations was approximately 1.5×10^2 s. The simulations were considered converged when, between two consecutive iterations, the residuals for mass, momentum, and energy equations were smaller than 10^{-6}, 10^{-6}, and 10^{-8}, respectively, as well as the residual for the turbulence model equations was smaller than 10^{-6}. In all of the simulations were carried out 7000 iterations. Besides, sub-relaxation factors of 0.7 were imposed in the conservation equations. It is important to mention that a independence mesh test was performed to define an adequate mesh to be used in this kind of problem. The study above quoted is going to be presented after the problem definition (next section).

3. PROBLEM DESCRIPTION

The problem consists in the application of the constructal design for the influence evaluation of some geometric parameters variation over the behavior of a solar chimney. To do so, considering Fig. 1, the degrees of freedom R/H and R_1/H_2 can vary; while the ratio H/H_1 = 10, the inlet radius (R_2 = 0.125 m), the outlet radius (R_3 = 0.5 m), and the height (H_3 = 0.6 m) of the generator chamber are constants. Hence, the SCPP height, the collector radius, the inlet and outlet radius of the collector section are variable according to the values showed in the Tab. 2. The physical conditions of the problem are the same considered by Patel *et al.* (2014), except for the computational domain that was considered here as axisymmetric. The work fluid is air, being considered as ideal gas. The acceleration of the gravitational field acts in the negative way of the y direction, as shown in Fig. 1. The reference pressure is atmospheric (p_{ref} = 1.0 atm). As boundary conditions, a prescribed pressure (p = 1.0 atm) and a prescribed temperature (T = 303 K) are imposed at the collector inlet (surface S_1) and at the collector outlet (surface S_2). This temperature can be considered since in a height H = 10.0 m the air temperature does not differ considerable in respect to the air temperature on the ground region. In addition, to the ground (surface S_3) a boundary condition of no slip and impermeability with a prescribed temperature of 323 K is adopted. This temperature represents the soil warming effect by the energy absorbed in the collector. The remaining surfaces in the computational domain have no slip and impermeability condition and are thermally isolated (adiabatic).

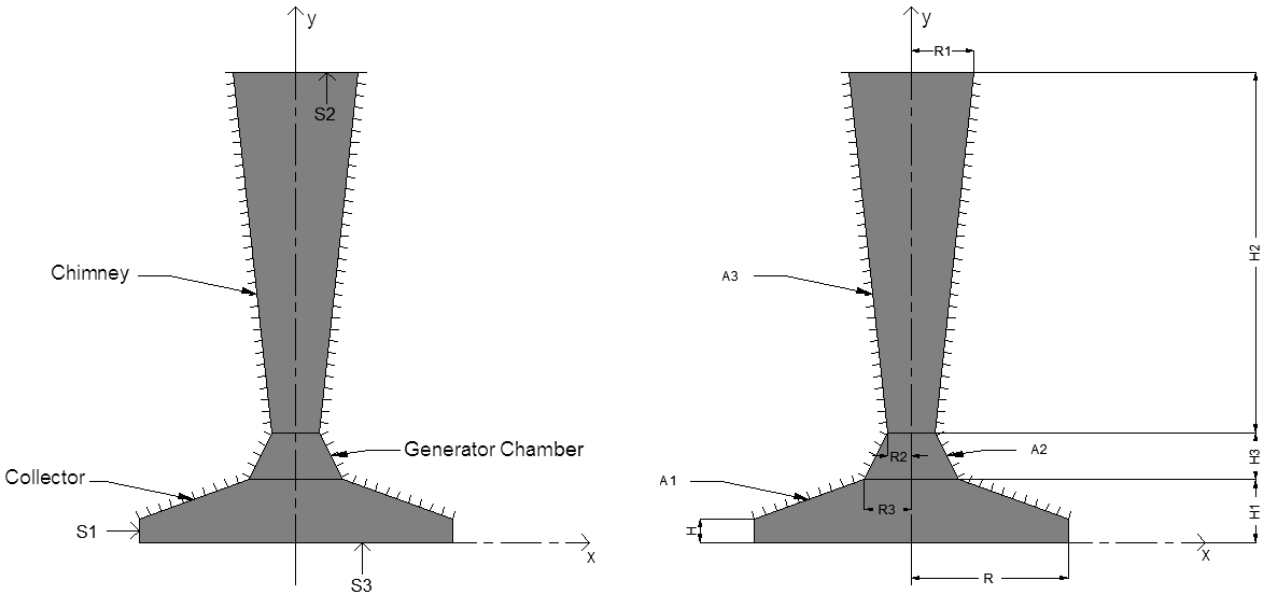


Figure 1. SCPP geometry: (a) components, and (b) dimensions.

Table 2. Geometric variation of degrees of freedom

R/H	H (m)	H_1 (m)	R (m)
10	0,1294	1,2942	1,2942
20	0,0953	0,9526	1,9052
40	0,0693	0,6930	2,7720
60	0,0573	0,5730	3,4381
80	0,0500	0,5000	4,0000
100	0,0450	0,4495	4,4952
150	0,0370	0,3700	5,5496
200	0,0322	0,3219	6,4387
250	0,0289	0,2889	7,2221
350	0,0245	0,2452	8,5819
500	0,0206	0,2059	10,2964
800	0,0163	0,1635	13,0772

It is worth to mention that the parameters presented in Tab. 2 were defined considering the imposed restrictions for the collector and chimney areas (A_1 and A_3, see Fig. 3) and the degrees of freedom R_1/H_2, R/H, and H_1/H. In the first geometric evaluation process the DOF R/H is varied while R_1/H_2 and H_1/H are kept constants, being the maximum available power obtained called as once maximized and the correspondent optimal geometry called as once optimized R/H ratio. After that, the DOF R_1/H_2 assumes different values and for each of these values the DOF R/H is also varied, being the maximum available power reached in this process called as twice maximized and the correspondents geometries, $(R_1/H_2)_o$ and $(R/H)_o$, are called as once and twice optimized geometries. Therefore, the Tab. 2 presents an evaluation range for the DOF R/H for the different values of DOF R_1/H_2 studied in this work.

The main goal of the presented work is to maximize the solar chimney available power, evaluating the effect of the degrees of freedom *R/H* and R_1/H_2 in the device performance. The SCPP available power is given by:

$$P_{avaiable} = \frac{\rho A \bar{v}^3}{2} \tag{8}$$

where ρ is the air density (kg/m^3), *A* is the cross sectional area of the SCPP in the turbine region (m^2) and $\bar{v}$ is the spatial average velocity at the analyzed section (m/s).

4. RESULTS AND DISCUSSIONS

A mesh independence study was carried out, considering discretizations for the computational domain with 18000, 19600, 20400, 23600, and 25500 rectangular computational cells. It was adopted in the wall and turbine regions a further refinement of the mesh using a stretched mesh. The velocity profile at the domain symmetry region attained with these meshes are presented in Fig. 2. The results indicate that the velocity obtained from the meshes with 23600 and 25500 cells present a relative difference less than 1.0 %. On that basis, it was considered the independent mesh with 23600 cells, being this mesh used for all numerical simulations performed in the present work.

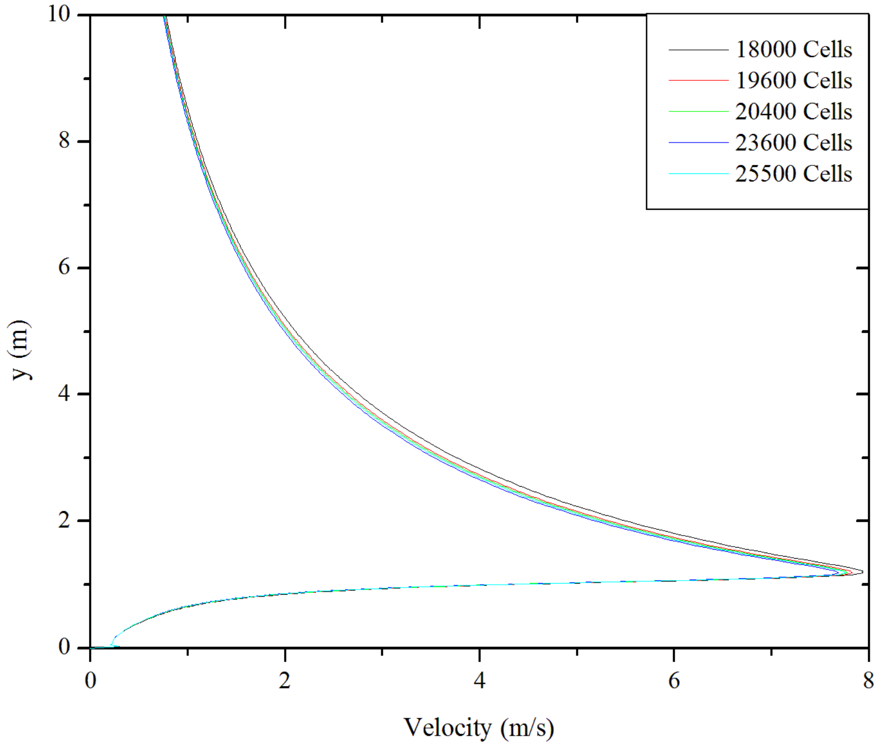


Figure 2. Mesh independence study.

After the mesh independence study, a verification of the proposed numerical model was realized comparing its results for the velocity field magnitude in the axis region of the SCPP with those obtained by Patel *et al*. (2014), as can be seen in Fig. 3. It can be observed in Fig. 3 that the results present an average deviation of about 2.0 %. The maximum velocity found in the present simulations is $v_{\max}$ = 7.68 m/s while in Patel *et al.* (2014) was reached a top velocity of $v_{\max}$ = 7.82 m/s. Besides, in the Patel *et al.* (2014) results there is a fluctuation in the velocity profile in the chimney region comprised in 6 m $\leq y \leq$ 10 m that is not obtained in the simulations of the present work. The behavior found in the reference is not foreseen once there is no force in the chimney central region that can reduce the profile only in that region. Despite the differences shown between the present results and those of Patel *et al*. (2014), one can note that the results are in agreement, mostly in the turbine region where the adequate estimative is more important for the SCPP available power prediction.

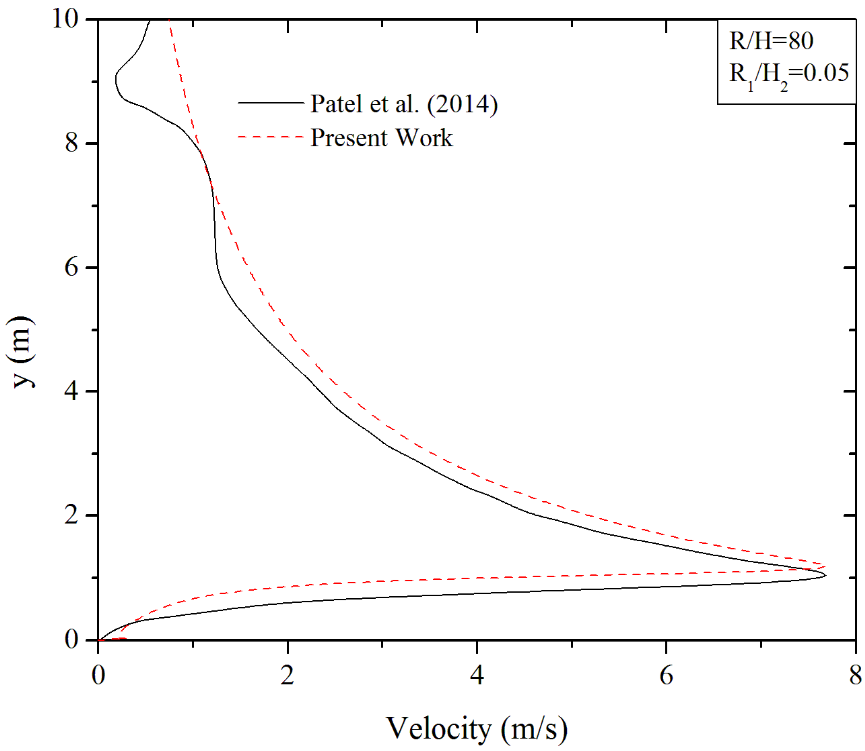


Figure 3. Verification of the proposed computational model.

Figure 4 shows the effect of the *R/H* variation for two values of R_1/H_2. Generally, the results indicate that the reason *R/H* presented a great influence on the available power (*P*). The increase of the ratio *R/H* conducts to the generation of higher powers, due to the greater radius in these cases if compared to the lower *R/H* values. Since the flow rate is the same for all studied cases, the greatest collector radius is the best option because it can promote a largest capture of the radiation energy which hence is transferred to the inside air of the SCPP. So, for example, the available power for $R_1/H_2 = 0.05$ with $R/H = 800$ is 28.73 W and with $R/H = 10$ is 6.03 W, being achieved an improvement around 79%. This behavior was also observed for the cases with $R_1/H_2 = 0.10$. In addition, it is possible to note in Fig. 4 that from *R/H* around 350 there is a stabilization trend for the available power around a constant value in both values of R_1/H_2. It is also showed in Fig. 4 that the augmentation of the R_1/H_2 value leads to a reduction of the available power. If compared the cases with $R/H = 800$, the difference between the available power when $R_1/H_2 = 0.05$ is over than 50% which when $R_1/H_2 = 0.10$. Therefore, these results indicate that the degree of freedom R_1/H_2 has influence on the available power (*P*).

Figure 5 presents the velocity magnitude distribution of three studied cases, for the ratio $R_1/H_2 = 0.05$. It can be noticed a lower velocity magnitude when $R/H = 20$, justifying the lower value for the available power in Fig. 4. However, when $R/H = 350$ there is an augmentation in the velocity magnitude which causes the improvement on available power (see Fig. 4). Besides, there is no considerable increase in the velocity magnitude from $R/H = 350$ to $R/H = 800$, explaining why the available power has the almost constant behavior already mentioned (see Fig. 4).

Finally, the Fig. 6 shows the temperature distribution of the same cases presented in Fig. 5, allowing to evaluate the effect of different collector radius on the SCPP thermal behavior. As the value of ratio *R/H* increases one can note a more uniform distribution of maximum temperature in the computational domain. It can be explained because for higher values of *R/H* the collector has a larger dimension *R* (see Fig. 1b) allowing a better radiant energy absorption. Hence, this affect directly the available power, since the air has more energy to speed up and move the generator's blades.

5. CONCLUSION

In this work a numerical study was conducted to investigate the applicability of constructal design in a SCPP, as well as, to evaluate the influence of some SCPP geometric parameters influence in its available power. In addition, other objective of this work was to verify the proposed numerical model for the simulation of SCPP operating principle, aiming to employ these model in future researches. In all numerically simulated cases it was considered compressible, turbulent, and steady flow, with heat transfer through mixed convection in an axisymmetric two-dimensional domain. The mass, momentum, and energy conservation equations are solved numerically using the finite volume method, by means the FLUENT software. To address the turbulence was used the $k-\varepsilon$ model.

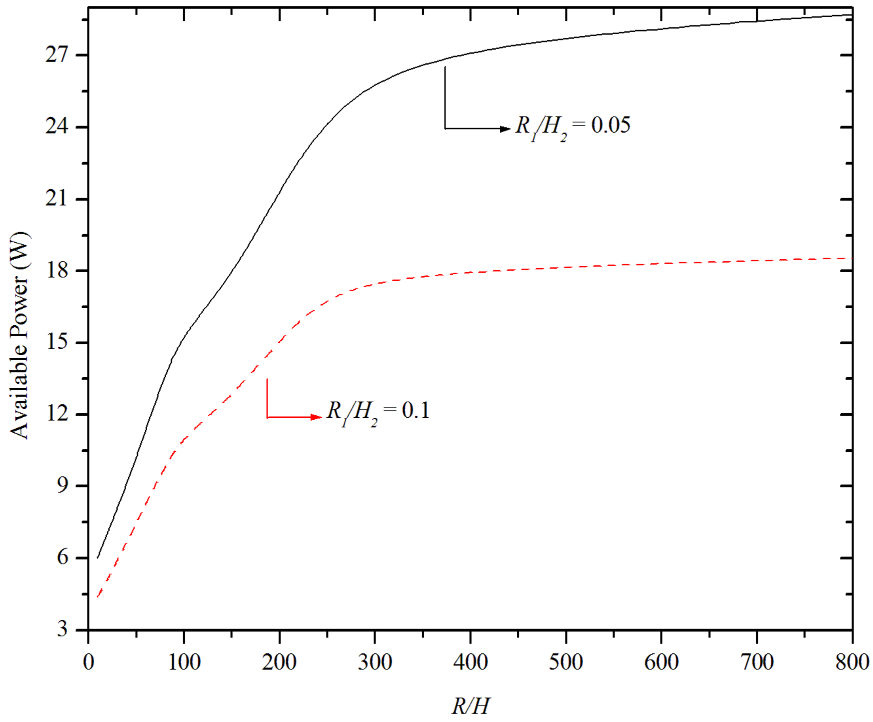


Figure 4. Effect of variation *R/H* in the available power for differents R_1/H_2.

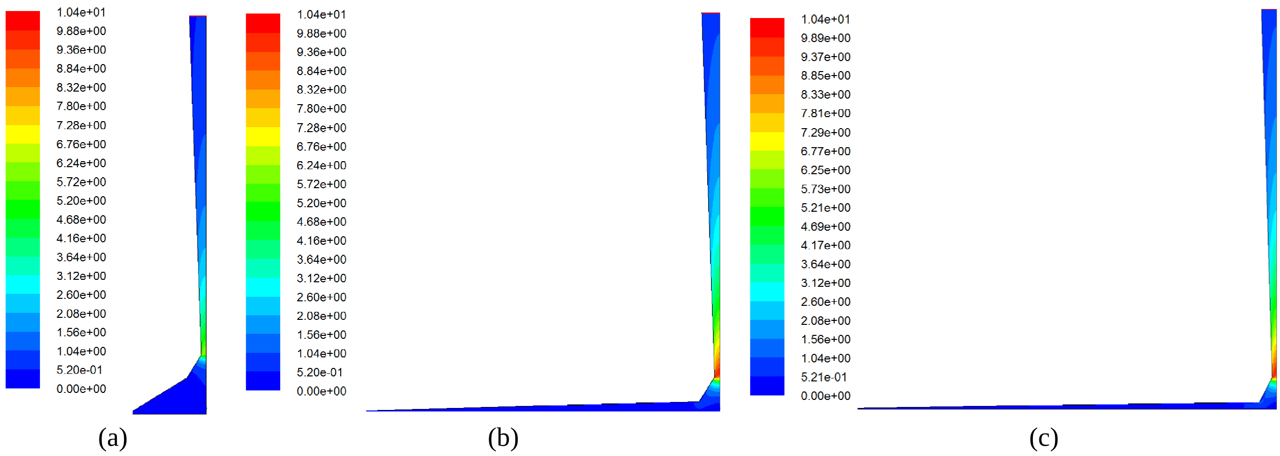


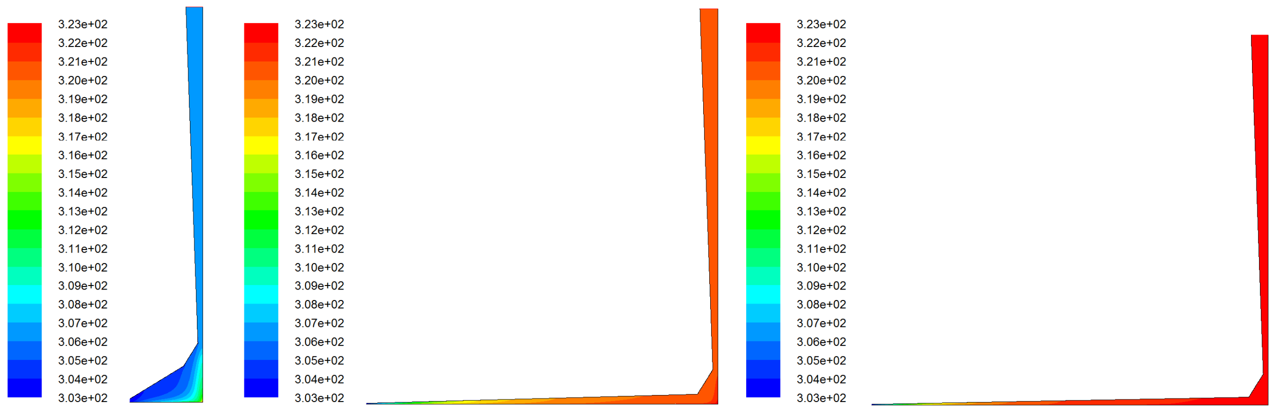
Figure 5. Velocity magnitude (axisymmetric): (a) *R/H* = 20, (b) *R/H* = 350, and (c) *R/H* = 800.

The results obtained with the proposed numerical model were in agreement with those presented by Patel *et al.* (2014), verifying the computational modeling.

The study about the influence of the degree of freedom *R/H* on the SCPP available power indicates that the increase of ratio *R/H* can significantly improve the available power. The results show that it is possible to improve the available power in almost 80 % applying the constructal design method by the variation of ratio *R/H*. Enhancements of up to 55 % were reached when the ratio R_1/H_2 was changed from 0.10 to 0.05.

Finally, analyzing the velocity and temperature fields one can note that the use of an adequate geometric configuration of SCPP promotes better fluid dynamic and heat transfer behaviors, explaining its performance improvements.

Therefore, the numerical simulation allied to the constructal design method can be applied to study the geometry influence of an SCPP device, allowing to find best shapes which conduct to a superior performances. In future works other values for ratio R_1/H_2 will be investigated.



(a) (b) (c)

Figure 6. Temperature field (axisymmetric): (a) *R/H* = 20, (b) *R/H* = 350, and (c) *R/H* = 800.

6. ACKNOWLEDGEMENTS

The authors J. A. Souza, E. D. dos Santos, L. A. Isoldi, and L. A. O. Rocha thank CNPq for financial support.

7. REFERENCES

Bejan, A., 2013, Convection heat transfer, Wiley, Hoboken, New Jersey, Fourth edition.

Dhahri, A. and Omri, A., 2013, “A Review of solar Chimney Power Generation Technology”, International Journal of Engineering and Advanced Technology, Vol. 2, No. 3, pp. 1 – 17.

FLUENT., 2007, “Documentation Manual – FLUENT 6.3.16”, Ansys, Inc.

Haaf, W., Friedrich, K., Mayr, G. and Schlaich, J., 1983, “Solar Chimneys Part I: Principle and Construction of the Pilot Plant in Manzanares”, International Journal of Solar Energy, Vol. 2,, pp. 3–20.

Jones, I., 2011, Engineering strategies for greenhouse gas mitigation, Cambridge University Press, Cambridge; New York, available at: http://dx.doi.org/10.1017/CBO9780511977909 (accessed 8 September 2014).

Launder, B.E. and Spalding, D.B., 1972, Lectures in mathematical models of turbulence, Academic Press, London, England.

Li, W., Wei, P. and Zhou, X., 2014, “A cost-benefit analysis of power generation from commercial reinforced concrete solar chimney power plant”, Energy Conversion and Management, Vol. 79, pp. 104–113.

Patankar, S.V., 1980, Numerical heat transfer and fluid flow, McGraw Hill, New York, USA.

Patel, S.K., Prasad, D. and Ahmed, M.R., 2014, “Computational studies on the effect of geometric parameters on the performance of a solar chimney power plant”, Energy Conversion and Management, Vol. 77, pp. 424–431.

Pretorius, J.P. and Kröger, D.G., 2006, “Critical evaluation of solar chimney power plant performance”, Solar Energy, Vol. 80, No. 5, pp. 535–544.

Versteeg, Henk Kaarle and Malalasekera, Weeratunge., 2007, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Pearson.

Wilcox, D. C., 2002, Turbulence modeling for CFD, DCW Industries, La Canada, USA, 2nded.

Xu, G., Ming, T., Pan, Y., Meng, F. and Zhou, C., 2011, “Numerical analysis on the performance of solar chimney power plant system”, Energy Conversion and Management, Vol. 52, No. 2, pp. 876–883.

Zhou, X., Yang, J., Xiao, B. and Hou, G., 2007, “Experimental study of temperature field in a solar chimney power setup”, Applied Thermal Engineering, Vol. 27, No. 11-12, pp. 2044–2050.

8. RESPONSIBILITY NOTICE

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