CONSTRUCTAL DESIGN OF AVAILABLE POWER OF A SOLAR CHIMNEY AT DIFFERENT IMPOSED GROUND TEMPERATURE

Rodrigo Spotorno Vieira

Christopher Garcia

rodrigospotorno@yahoo.com.br

chrissefir@gmail.com

School of Engineering, Universidade Federal do Rio Grande.

Av. Itália, km 8, Carreiros, 96203-900 – Rio Grande, RS, Brazil.

Mateus das Neves Gomes

mateus.gomes@ifpr.edu.br

Instituto Federal do Paraná, Campus Paranaguá.

Rua Antônio Carlos Rodrigues, 453, Bairro Porto Seguro, Paranaguá, PR, Brasil.

Luiz Alberto de Oliveira Rocha

luizrocha@mecanica.ufrgs.br

Department of Mechanical Engineering, Federal University of Rio Grande do Sul.

Rua Sarmento Leite, 425, 90050-170 - Porto Alegre, RS, Brazil.

Liércio André Isoldi

Elizaldo Domingues dos Santos

liercioisoldi@furg.br

elizaldosantos@furg.br

School of Engineering, Universidade Federal do Rio Grande.

Av. Itália, km 8, Carreiros, 96203-900 – Rio Grande, RS, Brazil.

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 numerical study is performed for different imposed ground temperatures, mimicking the effect of different radiative energy absorbed in device. Constructal Design is used to evaluate the studied geometry. The problem* *is submitted to three constraints: collector, turbine and chimney areas. Moreover, three degrees of freedom are taken into account: R_1/H_2 = 0.03 (ratio between the radius and height of chimney), H_1/H = 10.0 (ratio between the height of collector base and collector inlet height) and R/H (ratio between the collector radius and collector inlet height) which is varied in the present work. The main objectives are to test the applicability of the studied numerical model in future studies of SCPP geometric optimization and to test the effect of the ground temperature in the collector 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 FLUENT™ software. The time-averaged turbulence modeling (RANS) was used for the turbulence approach with standard model k - ε. For every studied cases, Constructal Design allowed the improvement of available power. Even for the lowest ground temperature studied (T_S = 303 K) the ratio between the optimal shape and the worst shape in the available power was almost two times. Moreover, it was noticed a large sensibility in the available power with ground temperature.*

Keywords: *Constructal Design, solar chimney, numerical study, available power.*

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 an increase of the absorbed energy by the ground. Once the ground is warm, the air inside the collector also warms up and tends to move by natural convection to the chimney region, expanding though a turbine, which is located between the collector and the chimney. Later, the air displaces toward chimney outlet, which has an atmospheric pressure. 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, improving the heat exchange of the air with the ground and the collector, as well as, intensifying the air momentum in collector and turbine regions. 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 design is the increase of air momentum 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 make the SCPP a promising technology for electricity generation in regions with abundant solar energy such as Brazil (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 works is concerned with the building of Manzanares plant, in Spain (Haaf *et al.*, 1983). It is also found in literature the development of mathematical models to predict the system performance (Pretorius and Kröger, 2006). Into the numerical studies framework, some has been dedicated to understanding the behavior of velocity, pressure and temperature fields in the SCPP, as well as, regarding the turbine performance (Xu *et al*., 2011). Other studies 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), where it is considered an incompressible, mixed convective, turbulent flow at the steady state. The domain is considered two-dimensional and axisymmetric. The main advantage of using a simplified model is the possibility to minimize computational effort, allowing the increase in the number of simulations. At all were attained 28 simulations allowing the evaluation of the effect of the ratio between the collector radius and collector inlet height (R/H) over the available power of SCPP turbine for four different ground temperatures (T_S). 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 it is used the Reynolds Averaged Navier-Stokes (RANS) classical modeling (Wilcox, 2002). To solve the closing problem of turbulence it is used the two differential equations model $k - \varepsilon$ (Launder and Spalding, 1972).

2 MATHEMATICAL AND NUMERICAL MODELING

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):

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

$$\rho_0\left[\frac{\partial}{\partial t}\left(\overline{v_i}\right)+\frac{\partial}{\partial x_j}\left(\overline{v_i v_j}\right)\right]+\frac{\partial \overline{P}}{\partial x_j}\delta_{ij}-\frac{\partial}{\partial x_j}\left\{\left(\mu+\mu_t\right)\left(\frac{\partial \overline{v_i}}{\partial x_j}+\frac{\partial \overline{v_j}}{\partial x_i}\right)\right\}+\rho_0 g_i \beta\left(\overline{T}-T_0\right)=0$$

$(i, j = 1, 2 \text{ and } 3)$ in $t \times \Omega$ (2)

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

where $\overline{(\,)}$ 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):

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

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

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).

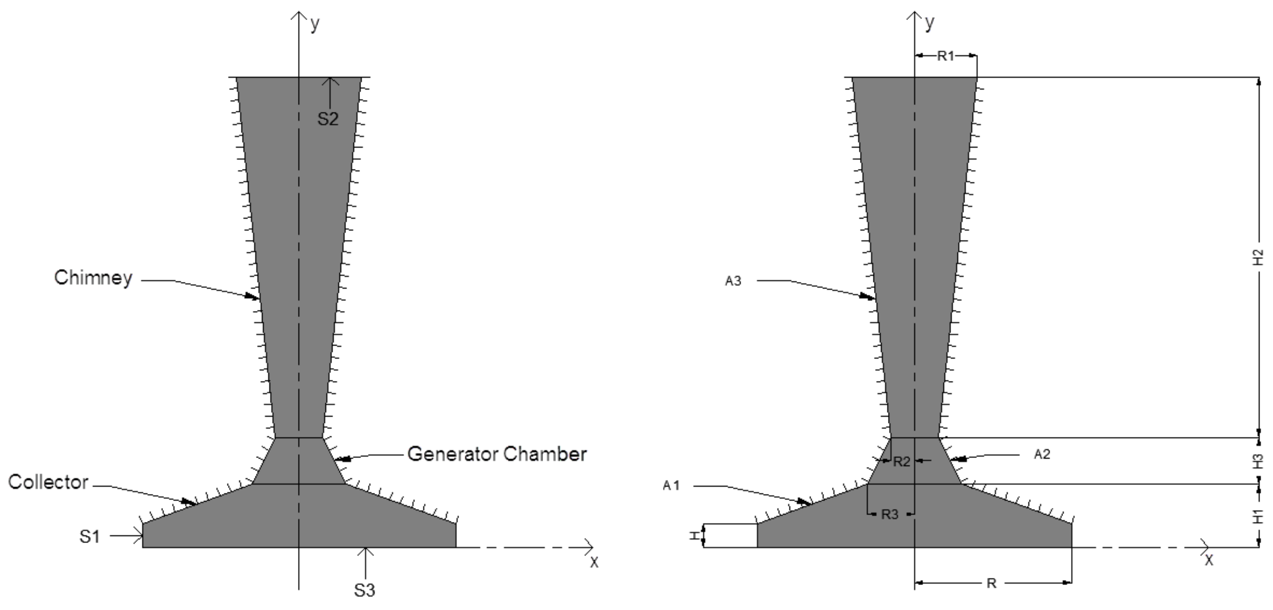
Constants	Values
C_μ	**0.09**
$C_{\varepsilon 1}$	**1.44**
$C_{\varepsilon 2}$	**1.92**
σ_k	**1.00**
σ_ε	**1.30**
Pr_t	**1.00**

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 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 an 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 and the ground temperature variation over the behavior of a solar chimney. To do so, considering Fig. 1, the degree of freedom *R/H* can vary; while the ratios R_1/H_2=0.03, H_1/H = 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. 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 *S2*). This temperature can be considered since for a height of H = 10.0 m the air temperature does not differ considerable from the air temperature on the ground region. In addition, in the ground (surface *S3*) a boundary condition of no slip and impermeability is imposed. Concerning the thermal boundary condition, four different prescribed temperatures are imposed in surface S_3: T_S = 293K, 303K, 313K and 323K. 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).



(a) **(b)**

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

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

$$P_{available} = \frac{\rho A v^{-3}}{2} \tag{8}$$

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).

According to Constructal Design, the present problem is subjected to three constraints. The collector area:

$$A_1 = (R_3 H_1) + \frac{(H_1 + H)}{2} \cdot (R - R_3) \tag{9}$$

The turbine region area,

$$A_2 = \frac{(R_2 + R_3)}{2} \cdot H_3 \tag{10}$$

and the chimney area,

$$A_3 = \frac{(R_1 + R_2)}{2} \cdot H_2 \tag{11}$$

Considering the constant areas and the degrees of freedom, 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.

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

4 RESULTS AND DISCUSSIONS

A mesh independence study is carried out, considering discretizations for the computational domain with 18,000, 19,600, 20,400, 23,600 and 25,500 rectangular computational cells. It was adopted in the wall and turbine regions a further refinement of the mesh using a stretched mesh. Velocity profiles 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 23,600 and 25,500 cells present a relative difference less than 1.0 %. On that basis, it was considered the independent mesh with 23,600 cells, being this mesh used for all numerical simulations performed in the present work.

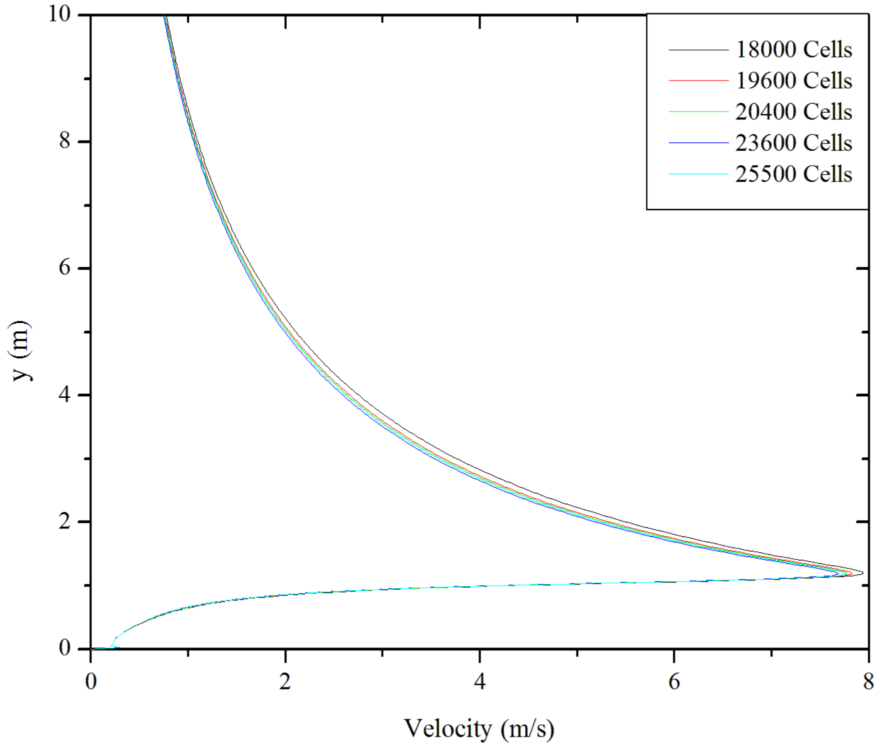


Figure 2. Mesh independence study.

After the mesh independence study, a verification of the proposed numerical model is realized comparing the predicted 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 presented 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 slight differences shown between the present results and those of Patel *et al*. (2014), one can note that the results are in agreement, mainly in the turbine region where the adequate estimative is more important for the SCPP available power prediction.

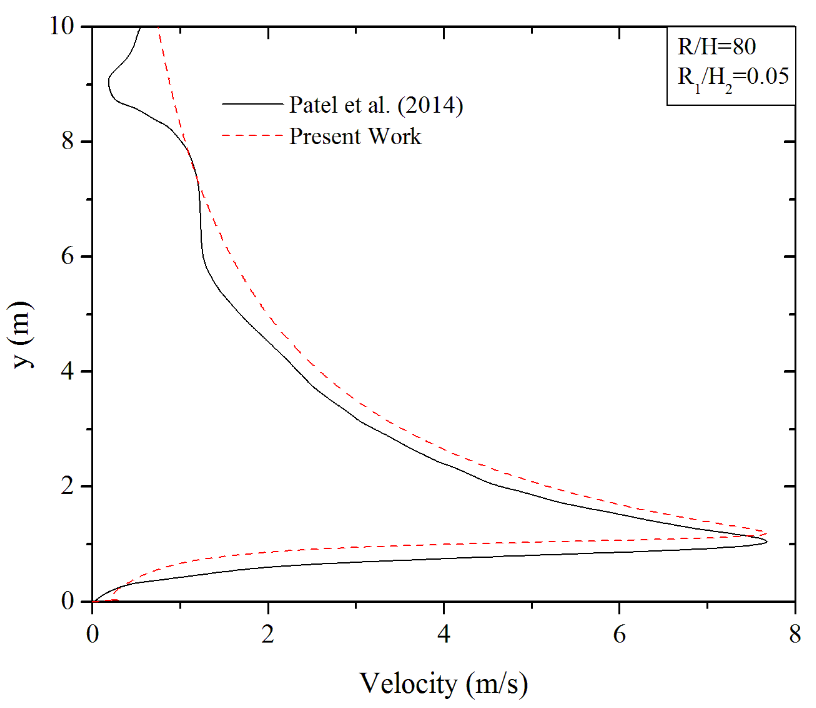


Figure 3. Verification of the proposed computational model.

Figure 4 shows the effect of the degree of freedom *R/H* on the available power to the four different ground temperatures evaluated here (T_S = 308 K, 313 K, 318 K and 323 K). It can be seen that the ground temperature and the degree of freedom *R/H* have large influence over the available power in the device. The ground temperature has a great influence because through the natural convection mechanism, the hot ground surface heats adjacent air leading to an augmentation of momentum in collector and turbine region of SCPP. Results showed that an increase of T_S = 308 K to T_S = 323 K, i.e., an increase of ΔT from 5 K to 20 K, it is possible to improve the available power in nearly 2.4 times. Concerning the effect of *R/H* over the available power, it can be noticed that for the highest magnitude of T_S (T_S = 323 K) the once maximized available power is P_m = 28.2W for an optimal ratio of $(R/H)_o$ = 800, it is also noticed that there is a stabilization of available power for ratios of $R/H \geq 700$. As the superficial temperature decreases the ratio which leads to stabilization decreases, e.g., for T_S = 318 K the ratio R/H = 800 does not conduct to a better performance than that achieved for R/H = 600. For the lowest values of T_S (T_S = 308 K and 313 K) the optimal geometry is not obtained for the highest ratio of R/H, showing the influence of the mechanism driven force over the design of SCPP. For T_S = 313 K there is an intermediate optimal ratio $(R/H)_o$ = 550 while for T_S = 308 K this optimal ratio decreases for $(R/H)_o$ = 300. In the latter case it is also noticed a decrease of available power for ratios higher than R/H = 300. Results also showed that, even for the lowest TS, the ratio between the available power for optimal and the worst shapes is nearly two times.

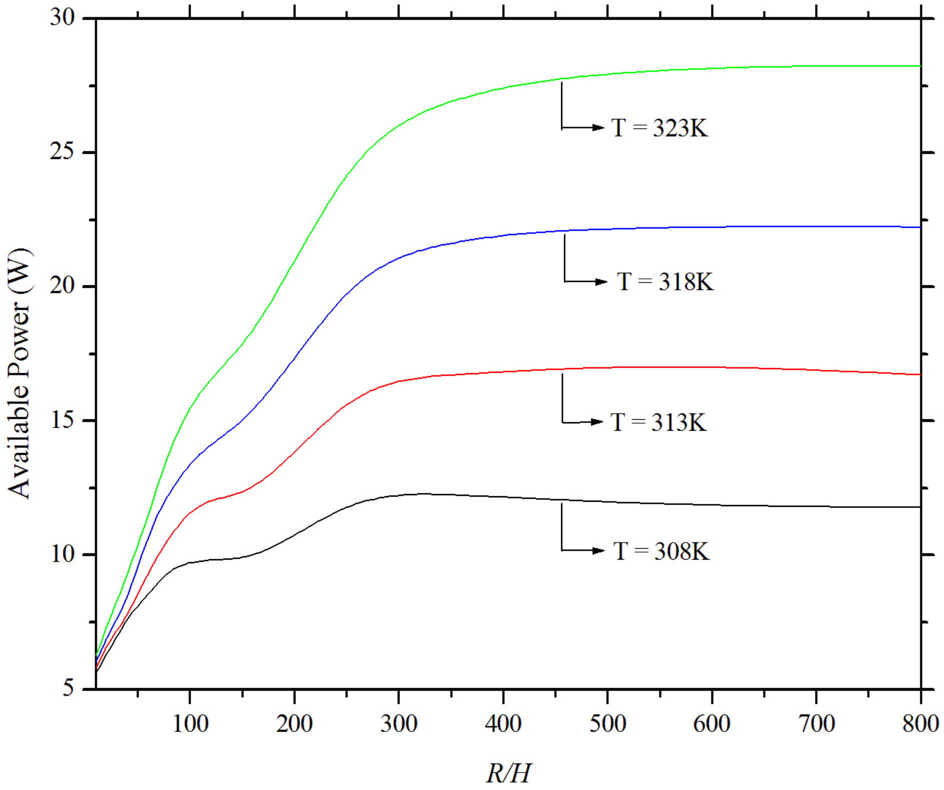


Figure 4. Ground temperature effects in available power.

5 CONCLUSION

In this work a numerical study was conducted to investigate the applicability of constructal design for geometrical evaluation of a Solar Chimney Power Plant (SCPP). In the present work it was investigated the effect of the ratio R/H (ratio between the collector radius and collector inlet height) over the available power. A sort of secondary purpose was to verify whether the numerical model is suitable for simulation of main operational principle of SCPP.

In all numerically simulated cases it was considered incompressible, turbulent, mixed convective flow at the steady state in an axisymmetric two-dimensional domain. The mass, momentum and energy conservation equations are solved numerically using the finite volume method, more precisely employing FLUENT software (Versteeg and Malalasekera, 2007; FLUENT, 2007). To address the closure modeling for turbulence, it was employed the RANS $k - \varepsilon$ model (Launder and Spalding, 1972).

In the first step, results obtained with the simplified proposed numerical model were in agreement with those presented by Patel *et al.* (2014), verifying the computational modeling and allowing future studies of geometrical evaluation, since the computational effort was strongly reduced in comparison with simulations of three-dimensional flows. Concerning the study about the influence of the ground temperature on the SCPP available power indicates that the increase of ground temperature can significantly improve the available power. It was also observed that the optimal ratio (R/H)o decreases with the decrease of ground temperature (TS), showing the importance to evaluate the SCPP design for different driven forces and conditions of external climate. To summarize, results showed the importance of employment of Constructal Design for geometrical evaluation of this kind of problem. Even for the lowest magnitude of T_S, the ratio between the available power for optimal and the worst shapes was nearly two times.

ACKNOWLEDGEMENTS

The author R. S. Vieira thanks CAPES for master science scholarship. C. Garcia thanks CNPq for scientific initiation scholarship. The authors E. D. dos Santos, L. A. Isoldi, and L. A. O. Rocha thank CNPq for financial support.

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, 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.