DOI: 10.14621/ce.20160102

Numerical Modelling of a Solar Chimney Power Plant

Filip V. Stojkovski[1], Marija Chekerovska[2], Risto V. Filkoski[1]*, Valentino Stojkovski[1]

[1]Faculty of Mechanical Engineering, University ”Sts Cyril and Methodius” 1000 Skopje, Macedonia; risto.filkoski@mf.edu.mk
[2]Faculty of Mechanical Engineering, University “Goce Delchev”, Shtip, Macedonia

Abstract

A solar chimney power plant is a relatively new concept for power generation, based on renewable energy, combining the greenhouse effect with the chimney suction. The present study involves mathematical modelling of the system, based on the computational fluid dynamics (CFD) approach. The technical features of solar chimney power plant are analysed by use of CFD technique, as a way for effective optimisation of the object’s geometry and thermo-fluid aspects. The created numerical domain represents the complete volume of the object under consideration, with total height of 100 m, chimney’s base radius of 6.25 m, chimney’s top radius of 10.5 m and roof radius of 100 m. The numerical grid consists of 276000 volume cells, 729242 faces and 190156 nodes. The governing equations for mass, momentum and energy are solved using a commercial CFD code. The computation is performed using the assumption of steady 3-D flow and the turbulence is taken into consideration with the k-ε realizable model. The discrete ordinates (DO) model is selected as thermal radiation model, since it represents properly the physicality of the radiation energy transfer phenomenon and due to the opportunity of applying a solar load directly to the radiation model. The obtained initial results demonstrate the capability of the CFD, as a powerful research and engineering tool for analysis of complex aerodynamic and thermal systems.

Keywords:	Solar energy; Solar chimney; Heat transfer; Convection; Thermal radiation; Power generation

Article history:	Received:	31 July 2015
	Revised:	19 October 2015
	Accepted:	29 January 2016

1. Introduction

The solar energy represents inexhaustible nonpolluting energy resource. The solar chimney power plant is one of the relatively new methods aimed for large-scale generation of electricity, based on quite simple that combines solar energy and the greenhouse effect with the chimney suction. Basically, as it is illustrated in Figure 1 (modified from [1]), air and soil, or other layer, water-filled pipes, bags etc., are heated underneath the collector transparent roof by solar radiation. As a consequence, in combination with the chimney effect, a strong upward air draft is created due to a density difference that drives a turbine connected to an electrical generator.

The idea of solar chimney power plant initially was proposed by the German engineers Jorg Schlaich and Rudolf Bergermann in 1976 [2]. The solar chimney consists of three essential components: a solar roof collector, a chimney and a turbine, which have been familiar from time immemorial, but in this case, combined in a new way [2]. Research efforts on solar chimney are characterised with a number of theoretical studies, but with insufficient experimental work. The tests conducted on the first prototype in Manzanares, Spain, (a chimney radius 5 m, a height 195 m and a collector radius of 120 m), with a designed peak output of 50 kW, have shown that the concept is technically viable [3, 4].

Some of the works, conducted in the meantime, are directed towards development of mathematical models, in combination with experimental research [5, 6]. Others, such as [7], are devoted to development of specific modelling approach, in order to study important effects, such as wall friction, internal drag and area change. Pretorius and Kröger in [8] solved a convective heat transfer equation, evaluated a more accurate turbine inlet loss coefficient and the effects of various types of soil on the performance of a large-scale solar chimney power plant. The paper [9] presents some

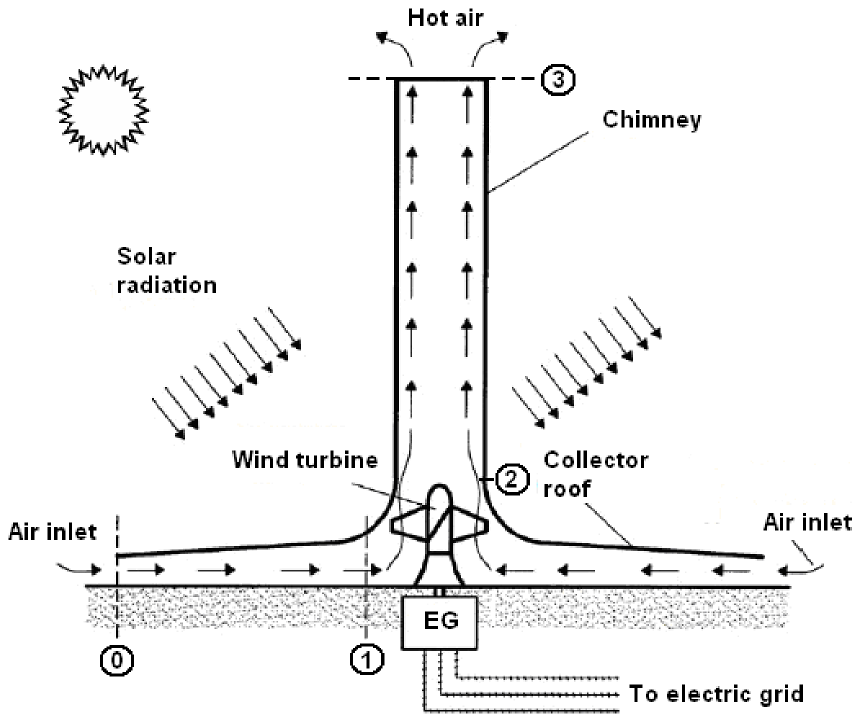

Figure 1: Schematic layout of a solar chimney power plant (modified from [1])

theory aspects, technical issues, practical experience and basic economy analysis of solar updraft towers, giving results from designing, construction and operating of a small scale prototype in Spain. The feasibility of solar chimney power plants as an energy source for settlements, islands and remote places of countries in regions with specific weather conditions, is analysed in the works [1, 10]. The accuracy of different theoretical models for solar chimney power plants, as well as the dependence of the plant's power output and efficiency on the geometry: chimney height and radius, collector radius and roof height is analysed in [11].

The common general findings of all the works regarding the efficiency can be summarised as follows:

- The plant overall efficiency is very low, but it increases with the plant size [11];
- The investment cost per MW installed power and the levelised electricity cost decrease with the plant size [9];
- In particular, a significant reduction of electricity generation cost is associated with increasing the plant size, leading to values comparable to the ones for electricity generated in conventional power plants [9];
- The optimisation of the plant geometry is essential in order to design an economically and technically practical system [3, 4, 9, 11].

The present work is a continuation of the study presented in [12], with the main objective to investigate a solar chimney power plant operation by use of a computational fluid dynamics (CFD) approach. The CFD technique, as a proven powerful engineering tool, has been extensively used for modelling and investigation of operational behaviour of aerodynamics and thermal energy systems, helping researchers and engineers in performing their work more efficiently. In combination with experimental research and on-site measurements, CFD offers multiple benefits, such as time and cost reduction, possibility to reproduce the operating conditions, analysis of geometry variations etc. The Fluent CFD software is used in the present study for performing the numerical modelling and simulation procedure [13].

2. Methodology of the research

2.1. Theoretical background

The operation principle of a solar chimney plant is shown in Fig. 1 (modified from [1]). Air enters the space under a low circular transparent or translucent roof that is open at the periphery and receives heat by solar radiation. The covering roof and the ground below it form a solar air collector. The height of the collector roof varies from the inlet to the junction with the chimney in accordance to the law of meridian flow theory. As a consequence, the air under the roof does not accelerate,

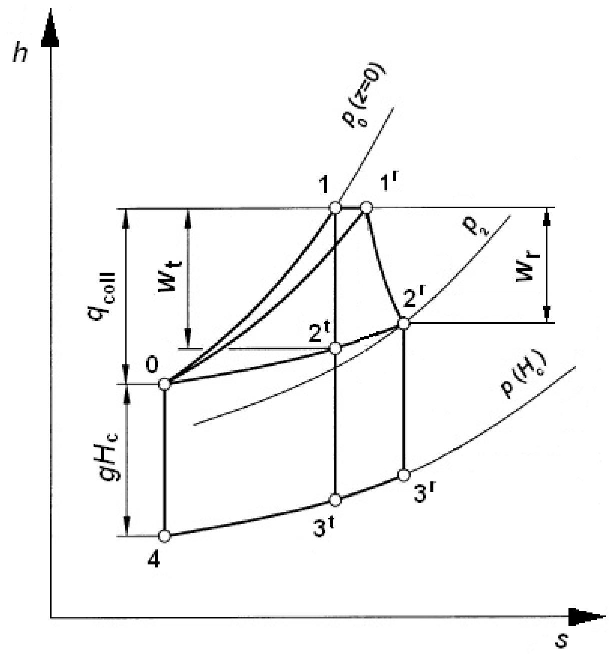

Figure 2: The ideal and real air process in solar chimney power plant

so that the hydraulic energy losses are minor. A vertical chimney, also called tower, with large air inlets at its base, is placed in the middle of the roof. The chimney directs and intensifies the air flow to the top, causing air acceleration in the zone behind the wind turbine, which results with a pressure drop and making it suitable for placing the turbine at that point.

Hot air flows up the tower due to its smaller density, compared to the cold air. Then, more hot air from the collector is drawn in by the tower suction, and cold air comes in from the outer perimeter. Continuous day-and-night operation can be achieved by placing a heat accumulator, water-filled tubes or tight bags, on the ground. In that case, the water heats up during day-time and releases heat at night, causing a relatively constant updraft in the tower. The energy contained in the air updraft is converted into mechanical energy by pressure-staged turbines located at the base of the chimney, which is then transformed into electrical energy by conventional generator.

The theoretical and real processes of air in a solar chimney power plant are illustrated in the enthalpy-entropy chart, Fig. 2 (slightly modified from [1]). The process 0-1-2^t-3^t-4-0 represents the ideal Brayton cycle. Due to aerodynamic losses, there is a small pressure drop in the real process, represented with the cycle 0-1^r-2^r-3^r-4-0. The annotation used in the chart has the following meaning: 0 – the state of air at the inlet; 1 – theoretical state of air at the collector outlet; 1-2^t – theoretical expansion in the turbine(s); 2^t-3^t – air flow to the top of the chimney; 3^t-4 – heat rejection; 4-0 – isentropic downdraft air movement to the inlet; 1^r – real state of air at the collector outlet; 1^r-2^r – real expansion in the turbine(s); 3^r – real state of air at the top of the chimney; 3^r-4 – real heat rejection.

2.2. Model set-up and simulation

This sub-section presents briefly the mathematical model, describing the solar chimney system geometry, the modelling and the numerical approach. The governing equations for mass, momentum and energy are solved using the commercial Fluent CFD code [13]. The object geometry is presented in Figure 3. The dimensions of the solar chimney under consideration are as follows: the collector roof inlet height is 2 m, the collector radius is 100 m, the chimney height is 100 m, the chimney's base radius is 6.25 m and the chimney's top radius is 8.75 m. The numerical mesh is given in Figure 4, consisting of 276003 volume cells, 729242 faces and 190156 nodes.

The numerical simulations were carried out using segregated implicit pressure based solver for steady state conditions. The governing differential equations for mass and momentum are solved for steady incompressible flow. Turbulence is taken into account by the realizable k-ε model, with inclusion of standard wall function for near wall treatment. The velocity-pressure coupling has been effectuated through the SIMPLEC algorithm. Second order upwind scheme was chosen for the solution scheme.

The interpolation formulae for the air thermo-physical properties, used in the calculations, depending on absolute temperature, are given in Table 1.

Two modelling approaches were considered within the present research, one taking into account natural convection without radiant heat transfer within the envisaged system (model 1) and the second one taking into consideration natural convection with radiant heat transfer (model 2).

In the framework of the model 1, it was assumed that heat is coming from the collector roof and the heat accumulation system under the collector roof, situated at the ground surface. The adopted pressure values at the inlet and the chimney's outlet are p_1=p_2=0 [Pa], at atmospheric pressure of p_{atm}=101325 [Pa]. In this case, the solver is density based, using the Boussinesq approximation theory for natural convection [13]. The density is set to change by the ideal gas law. The heat flux given at the collector roof surface and the heat accumulation is set as Φ = 1000 W/m^2.

In the model 2, in the present study, the discrete ordinates (DO) model is selected as a thermal radiation model, due to the opportunity of applying a solar load directly to the DO model [13, 14, 15]. The DO radiation model considers the radiate transfer equation (RTE) in the direction **s** as a field equation:

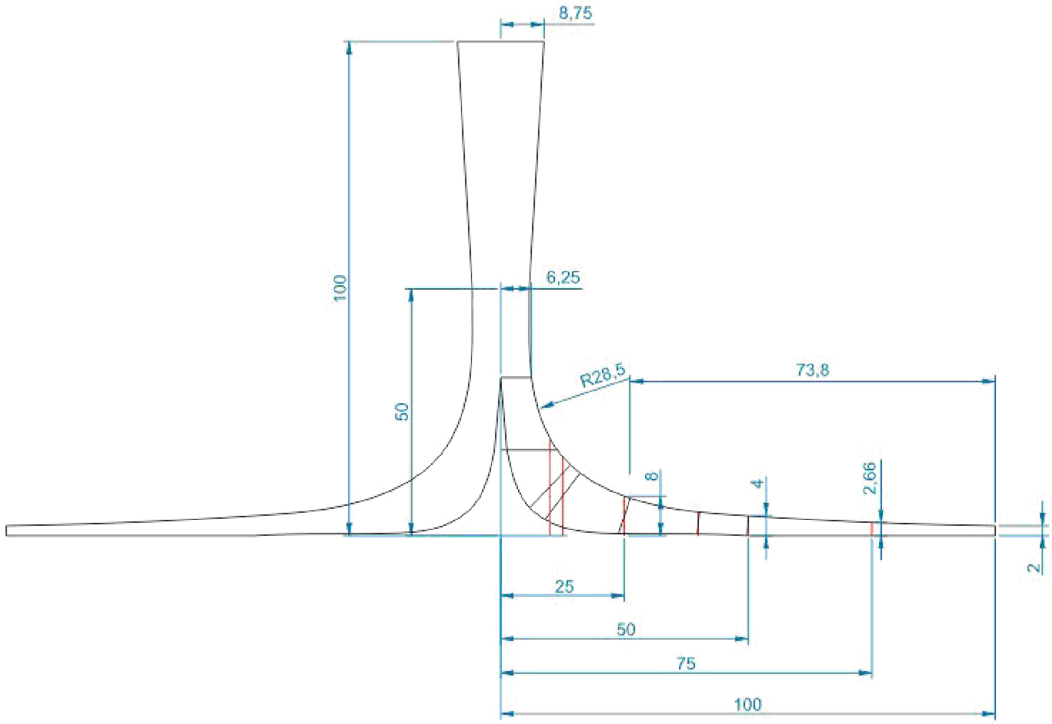

Figure 3: The object geometry

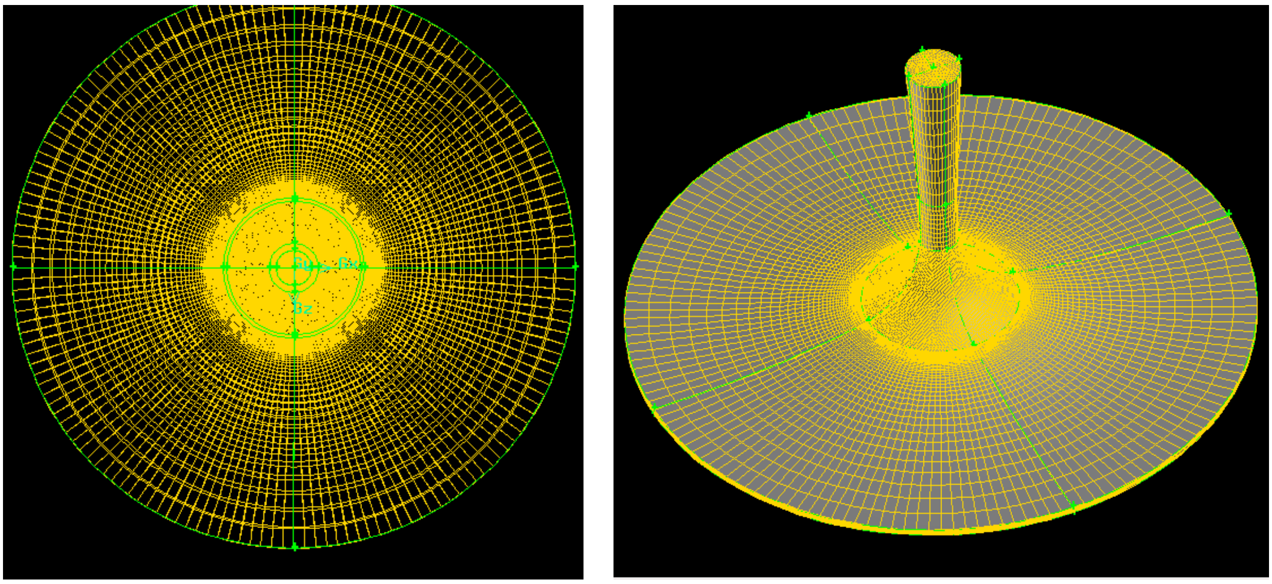

Figure 4: The numerical mesh of the object

Table 1: Thermo-physical properties of the air (*T* in K)

Property	**Interpolation function**
Density, kg/m^3	$\rho = 2.829 - 0.0084T + 0.00001T^2 - 5{\cdot}10^{-9}T^3$
Dynamic viscosity, kg/ms	$\mu = 5{\cdot}10^{-8} + 4{\cdot}10^{-10}T$
Specific heat capacity, J/kgK	$c_p = 1014.7 - 0.0005T + 0.0001T^2$
Thermal conductivity, W/mK	$\lambda = 0.0228 + 8{\cdot}10^{-5}T$

$$\frac{dI(\mathbf{r},\mathbf{s})}{ds}+(a+s_s)I(\mathbf{r},\mathbf{s})=$$

$$an^2\frac{\sigma_0 T^4}{\pi}+\frac{\sigma_s}{4\pi}\int_0^{4\pi} I(\mathbf{r},\mathbf{s'})\Phi(\mathbf{s}\cdot\mathbf{s'})d\omega' \qquad (1)$$

where: $I_l(\mathbf{r},\mathbf{s})$ [W/m^{2}srad] is the spectral intensity, λ is the wavelength, a [m^{-1}] is the spectral absorption coefficient, **r** [-] is the location vector, **s** [-] is the direction vector (defined as **s**= $\mu\vec{i}+\eta\vec{j}+\xi\vec{k}$), **s**' [-] is the vector of scattering direction, σ_s [m^{-1}] is the scattering coefficient at wavelength λ, σ_0 [W/m^{2}K^4] is the Stefan-Boltzmann constant, σ_0 =5,672·10^{-8} W/m^{2}K^4, ω' [-] is the solid angle, Φ is the phase function, which represents the probability that a ray with frequency v' from the direction **s** in a finite discrete solid angle $d\omega'$ will veer in the direction **s**' inside the angle $d\omega$, with frequency v.

The irradiation flux is applied directly to the semi-transparent walls (the collector roof) as a boundary condition, and the radiative heat transfer is derived from the solution of the DO radiative transfer equation. The DO radiation method considers the radiative transfer equation (RTE) as a field function. The RTE is solved for a finite number of discrete solid angles, each associated with a vector direction **s**, fixed in the Cartesian system. The fineness of the angular discretization can be changed accordingly and the DO model solves as many transport equations as there are directions **s**. In this case, the so-called S6 approximation was applied, corresponding to 48 flux approximations [13, 14, 15]. This approach gives sufficiently reasonable results for the amount of the numerical work. The higher-order approximations, such as the S8, with 80 flux approximations, require considerably more numerical effort.

Imposing appropriate boundary conditions on the numerical domain is very important step in the creation of the CFD model. A 'pressure inlet' boundary condition is specified for the air inlet at the periphery bellow the collector roof and for the chimney outlet, an 'outflow' condition is specified. The collector roof is defined as semi-transparent cover, to which the irradiation flux is applied directly, as a boundary condition. The solar irradiation to the collector roof was changed in the range 400-1000 W/m^2, based on actual weather conditions for selected season and period of the day.

The grid independence test was performed in order to check the impact of the mesh density on the numerical solution, for meshes in range between 160000 and 330000 cells. Since the refinement from 276000 to 330000 cells did not change the results by more than 0.5 %, it was concluded that the influence of eventual further grid refinement would be negligible and,

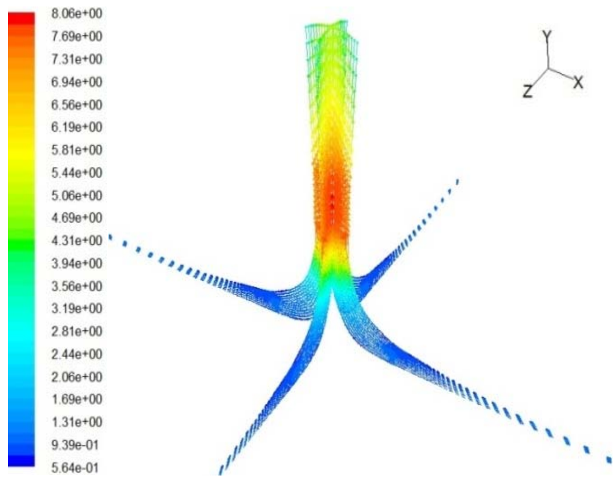

Figure 5: Velocity vectors obtained with the model 1

therefore, the present mesh quality was taken as appropriate for computation.

3. Results and discussion

The established mathematical model was used to conduct numerical simulations for different seasons of the year, periods of day and various weather conditions. Regarding the meteorological conditions, it was assumed that the object considered in this study is located in area with latitude 41° 32' and longitude 22° 07' (Krivolak region, Republic of Macedonia). Some results of the provided investigation, derived presuming average weather conditions during middle of April, are presented with the illustrations given in the next sub-sections.

3.1. Results of the model 1

The results of the model 1 show that the heat is evenly distributed under the collector roof and the velocity profile of the flow is uniform. The maximum velocity of the air in the plant, obtained by the simulations, is 8.06 m/s. The velocity vectors in two vertical cross-sections in this case are shown Figure 5.

However, by applying different heat fluxes at the surfaces, there was no significant change of the intensity of the air flow, as the heat energy appeared to be insufficient to intensify the flow and to generate higher rates of natural convection, which leads to a conclusion that this modelling approach is not sufficiently accurate.

3.2. Results of the model 2

The model 2 is considered as successful, because by changing the heat intensity in the model, there was a

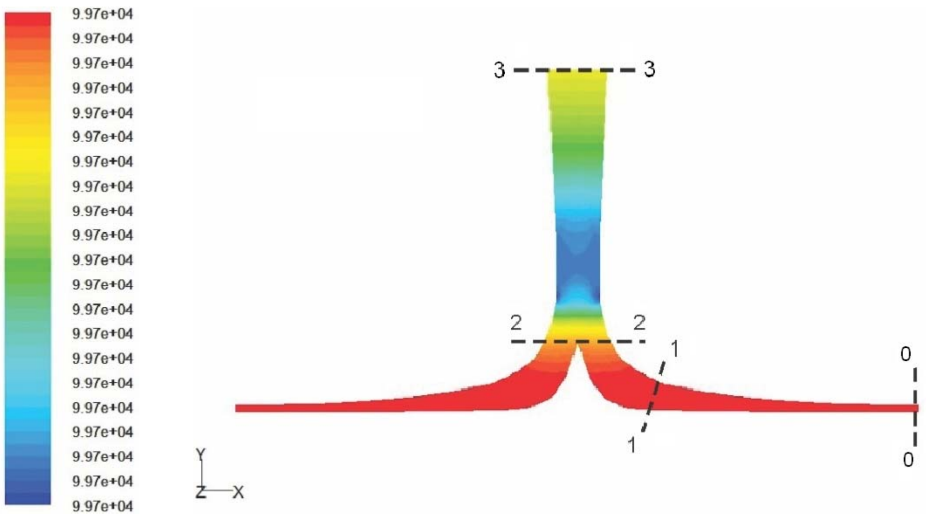

Figure 6: Contours of static pressure (in Pa)

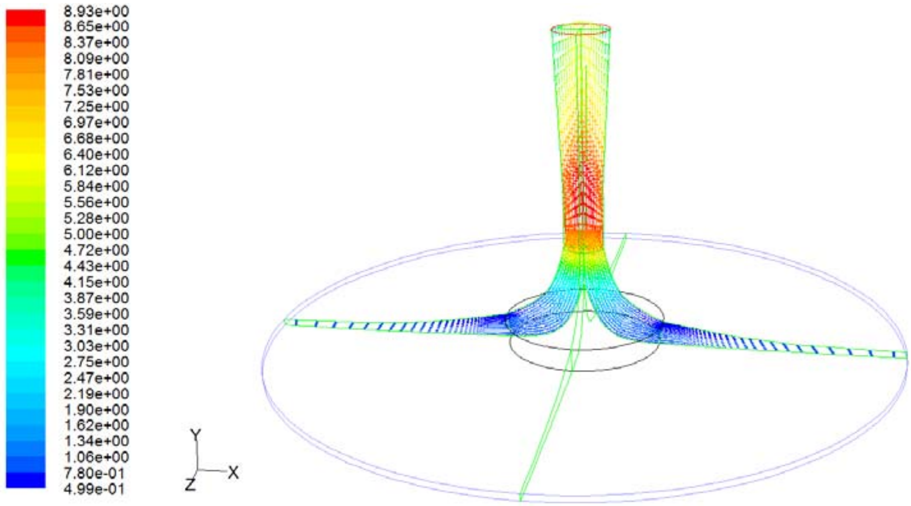

Figure 7: Velocity vectors coloured by velocity magnitude (in m/s) in central vertical intersection

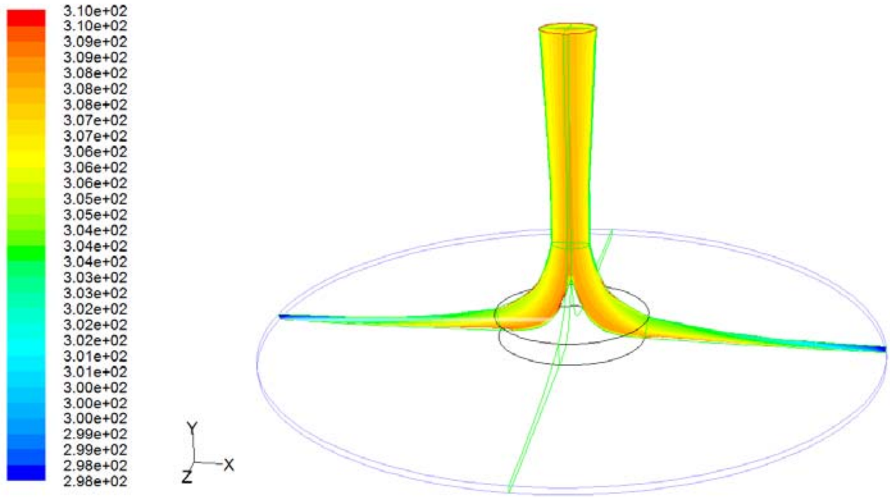

Figure 8: Contours of static temperature (in K), with implemented ground heat accumulator

significant change of the velocity increasing rate, which is regarded as a proof that the natural convection effect was established numerically.

Figure 6 shows contours of static pressure in a central axis vertical intersection. Velocity vectors coloured by the velocity magnitude in a vertical intersection are given in Figure 7. Contours of static absolute temperature, in a case with implemented ground heat accumulator, are presented in Figure 8.

The comparison between the results regarding the air velocity, obtained by CFD simulation and calculations based on the Boussinesq approximation theory for natural convection [13], are given in Table 2 and Chart 1.

It can be concluded from Chart 1, that the trend of the velocity calculated with CFD is following the curve patter calculated from the Boussinesq approximation theory, so the results can be assumed as correct.

The total enthalpy, as an energy "input" in the system, is transformed into enthalpy of air, pressure energy, kinetic energy, potential energy and energy loss. The maximum value of the total enthalpy in this case was E_{max} = 16253 J/kg. By far the largest share of the energy is transformed into enthalpy of air (over 93 %) and just a small part is utilised in the wind turbine, which results with output electric power between 15 and 25 kW, depending on the air flow rate.

Table 2: Comparison between the air velocities obtained by CFD simulation and calculations based on the Boussinesq approximation

T-coll. (K)	Mass flow (kg/s)	v_0 (m/s)	v_2 (m/s)	v_max - CFD (m/s)	v_max - Boussinesq (m/s)
350	993.92	0,787	4,281	8.378	9.881
345	981.34	0,777	4,223	8.259	9.748
340	969.01	0,766	4,166	8.141	9.613
335	956.54	0,756	4,108	7.979	9.475
330	944.11	0,746	4,05	7.905	9.335
325	931.55	0,736	3,991	7.785	9.192
320	918.63	0,725	3,932	7.660	9.047
315	905.74	0,714	3,871	7.539	8.898
310	892.62	0,703	3,809	7.417	8.744
305	879.05	0,692	3,743	7.294	8.585
300	864.87	0,681	3,674	7.165	8.421

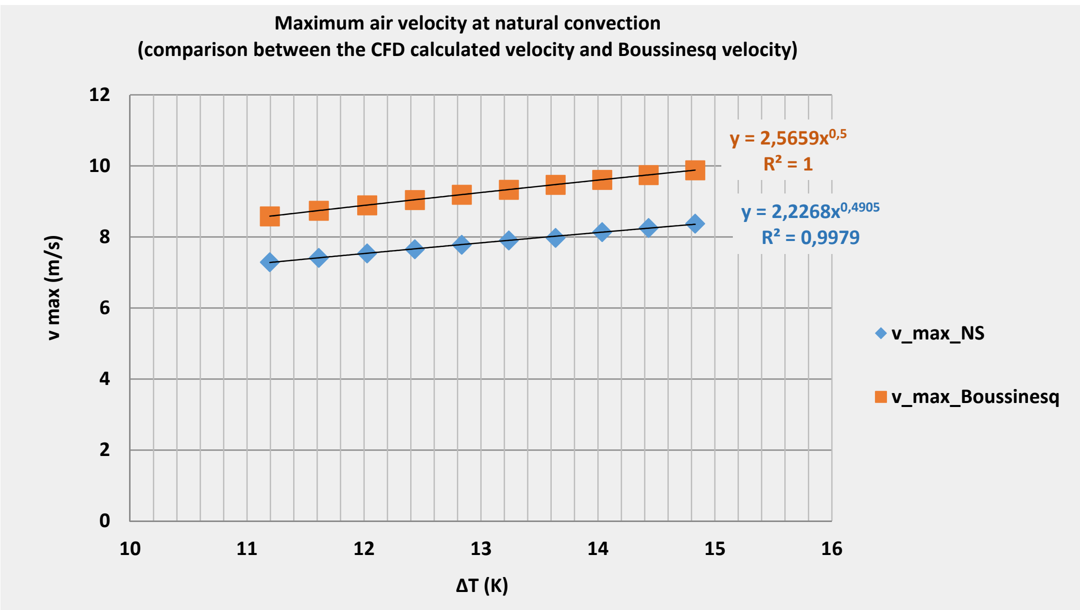

Chart 1: Comparison of the CFD calculated velocity and Boussinesq velocity at natural convection

4. Closing remarks

The present study demonstrates the capabilities of the CFD technique, as a powerful research and engineering tool for analysis of complex aerodynamic and thermal systems, such as the solar chimney power plants. The CFD approach should enable an experimental study of a solar chimney power plant to be simpler and more economical, so this approach can be further developed and upgraded for more detailed analysis.

Comparison between the two models applied in the present study, clearly shows that for further CFD investigation of such objects, a proper radiant heat transfer model should be included in the overall model, changing the air thermo – physical properties with other recommended values, or even changing the aerodynamic geometry of the object can lead to obtaining more accurate results.

Objects in this study are assumed to be achieved, so that a successful numerical model for a solar chimney power plant is developed, i.e. the natural convection phenomena is obtained and proved with the CFD simulations (figure 9) which follows the trend of the Boussinesq theoretical calculated velocity.

The accuracy of the results is relevant for this subject, but not so important, because the effort is aimed for developing a successful CFD simulation which can be used for further investigation of the flow and convection phenomena and energy capacities of such objects.

Although, it can be concluded from this CFD approach, from the efficiency viewpoint, that it is much better to include the heat accumulation (water pipes, water bags or other layer on the ground) that will release heat from the ground during the night time, enabling significant increase of the overall plant efficiency, which is otherwise quite low and the natural convection can not be developed.

References

[1] Nizetic S., Ninic N., Klarin B., Analysis and feasibility of implementing solar chimney power plants in the Mediterranean region, *Energy*, Vol. 33, (2008), pp. 1680-1690.

[2] Schlaich J., Schiel W., Solar Chimney, *Encyclopaedia of Physical Science and Technology*, Third Edition, 2000.

[3] Haaf W., Friedrich K., Mayr G., Schlaich J., Solar chimneys. Part I: Principle and construction of the pilot plant in Manzanares, *International Journal of Solar Energy*, 2, (1983), pp. 3–20.

[4] Haaf W., Solar chimneys. Part II: Preliminary test results from the Manzanares plant, *International Journal of Solar Energy*, 2, (1984), pp. 141–161.

[5] Pasumarthi N., Sherif S. A., Experimental and theoretical performance of a demonstration solar chimney model – Part I: Mathematical model development, *International Journal of Energy Research*, 22, (1998), 3, pp. 277–288.

[6] Pasumarthi N., Sherif S. A., Experimental and theoretical performance of a demonstration solar chimney model – Part II: Experimental and theoretical results and economic analysis, International Journal of Energy Research, 22, (1998), 5, pp. 443–461.

[7] von Backström T.W., Gannon A. J., Compressible flow through solar power plant chimneys, *ASME Journal of Solar Energy Engineering*, 122, (2000), 3, pp. 138–145.

[8] Pretorius, J.P., Kröger, D.G., Solar chimney power plant performance, *Journal of Solar Energy Engineering*, 128, (2006), pp. 302–311.

[9] Schlaich J., Bergermann R., Schiel W., Weinrebe G., Design of Commercial Solar Updraft Tower Systems – Utilization of Solar Induced Convective Flows for Power Generation, *Trans. of the ASME, Journal of Solar Energy Engineering*, 127, (2005), pp. 117-124.

[10] Hamdan M. O., Analysis of a solar chimney power plant in the Arabian Gulf region, *Renewable Energy*, Vol. 36, (2011), pp. 2593-2598.

[11] Koonsrisuk A., Chitsomboon T., Accuracy of theoretical models in the prediction of solar chimney performance, *Solar Energy*, Vol. 83, (2009), pp. 1764-1771.

[12] Filkoski V. Risto, Stojkovski V. Filip, Stojkovski Valentino, A CFD study of a solar chimney power plant operation, *Proceedings*, (Editor A. Olabi), 6th Int. Conf. on Sustainable Energy and Environmental Protection SEEP 2013, Maribor, Slovenia, 2013, pp. 631-636.

[13] Fluent Inc., Fluent 6.2 User's Guide, Lebanon NH 03766, USA, 2005.

[14] Fiveland W. A., Three-dimensional radiative heat-transfer solutions by the discrete-ordinates method, *Journal of Thermophysics*, Vol. 2, No. 4, pp. 309-316, 1988.

[15] Chekerovska Marija, Filkoski V. Risto, Efficiency of solar-tracking liquid flat-plate solar energy collector, Accepted for publication in (Int. Journal of) *Thermal Science*, (2015).