HUNGARIAN AGRICULTURAL ENGINEERING
N° 32/2017 29-37
Published online: http://hae-journals.org/
HU ISSN 0864-7410 (Print) / HU ISSN 2415-9751(Online)
DOI: 10.17676/HAE.2017.32.29
Received: 2017.10.17. - Accepted: 2017.12.07.

PERIODICAL OF THE COMITTEE OF AGRICULTURAL AND BIOSYSTEM ENGINEERING OF
THE HUNGARIAN ACADEMY OF SCIENCES
and
SZENT ISTVÁN UNIVERSITY
Faculty of Mechanical Engineering

AXIAL FLOW TURBINE FOR SOLAR CHIMNEY

Author(s):
W. M.A Elmagid[1] – I. Keppler[2]

Affiliation:
[1]Institute of Environmental Systems, Szent István University, Páter Károly street 1, Gödöllő, H-2103, Hungary
[2]Institute of Mechanics and Machinery, Szent István University, Páter Károly street 1, Gödöllő, H-2103, Hungary

Email address:
w.abdelmaged@Aswu.edu.eg, keppler.istvan@gek.szie.hu

Abstract

Producing sufficient amount of energy is critical issue over the entire world. A rather new method of harvesting renewable energy is taken into the focus of our research to cover the increase in electricity demand. The solar chimney power plant (SCPP) is one of some solutions to produce electrical energy by utilizing renewable source. The combination of solar energy and chimney effect is the renewable source that delivers the SCPP its energy. In this study, the redesign of an axial turbine for SCPP of Manzanares prototype is presented to simulate SCPP overall with using radiation model. Additionally, the investigation of flow inside the turbine is carried out by using the three dimensional CFD model. The CFD model solves Reynolds-averaged Navier–Stokes equations (RANS equation) using K-ε turbulent model. The solar radiation is calculated by using two different radiation models according to the physical state. P1 radiation model is applied in the fluid zone, and Monte Carlo model is applied in the solid zone. The comparison of the CFD results and previous experimental results show a good agreement, which validates our CFD model.

Keywords

Solar Chimney, Computational Fluid Dynamics, Turbine

1. Introduction

Utilization of renewable resources is dramatically essential for reducing fossil fuel effect on the human environment. Thus, the application of renewable energy has therefore been the goal of numerous researchers to improve the cost of energy production. To that end, many innovative concepts have been proposed to capture more energy from the renewable energy sources such as solar, wind, and hydroelectric energy. The most promising paths towards the sustainable development are solar energy, especially in energy production.

One of the options that will help in global electricity production by using renewable energy resource is the solar chimney power plant (SCPP). The Solar Chimney Power Plant System (SCPPs) is a natural driving power generating system. It can convert solar energy first into thermal energy then into kinetic energy and finally into electrical power. The operation of a solar chimney power plant (SCPP) is based on a simple principle: when air is heated by the greenhouse effect under the transparent roof of a collector, buoyancy forces arise as a consequence of density variation, this less dense hot air rises up a chimney, which installed at the centre of the collector. At the base of the chimney, the air flows through a turbine to produce mechanical energy for driving a generator. It combines the concept of solar air collector and central updraft chimney to generate a solar induced convective flow which drives wind turbines to generate electricity. The SCPP consists of a greenhouse roof collector, wind turbine, and updraft chimney that is located at the centre of the greenhouse roof collector, as shown in Figure 1. The SCPPS has been proposed as a device to economically generate electricity from solar energy in commercial-scale in the future.

Cabanyes (1903) first proposed the solar chimney power technology concept. In 1978, Schlaich again presented the technology in a congress [1].During the two-year period between 1981 and 1982, this technology has been verified by the successful construction and operation of the 50 kW Manzanares, Spain SCPP pioneer prototype [2]. This prototype has a 194.6 m high chimney, a radius of the collector is

122 m, and a vertical axis single-rotor turbine without inlet guide fans configuration installed at the base of the chimney. This prototype designed to operate with the peak power lying at about 50 kW for eight years, whereas in fact, the output measured was 36 kW [3].

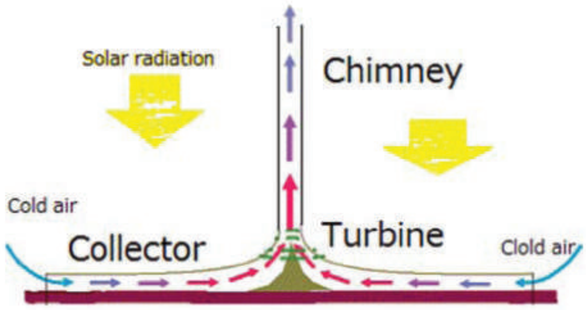

Figure 1. Schematic diagram of SCPP [4].

Since then, many prototypes of the SCPP had been built by experts in various countries. Australia intends to construct the largest SCPP in world at present with the generation capacity reaching up to 200MW in New South Wales of Australia. The chimney in the plant will be 1000m in height and the collector is 7 km in diameter, the system would cover a ground area of 38 km2 [5]. Significant research effort has been put into the construction, simulation and operation of the solar chimney collector. An analytical model has been presented to study detailed performance for main plant elements (collector, chimney and wind turbine). Mullett [6]developed analysis for the SCPP, aimed particularly to calculate overall efficiency and performance. He concluded that the solar chimney is essentially a power generator of large scale. Overall efficiency is directly related to the height of the chimney and is shown to be about 1% for a height of 1000 m. This resultants is confirmed by Yan et al [7].

Experimental and numerical calculation method can be used to study the performance of the SCPPs. But the large scale system is hard to establish. Computational fluid dynamic (CFD) take an effective role in a solar chimney technology research. Subsequently, with the development of computer technology and CFD software techniques, both temperature and pressure distribution in the large system can easily be predicted by numerical calculation method [8]. Rafiuddin et al [9] study the optimization a geometry of the major components of the SCPP using a computational fluid dynamics (CFD) technique within software ANSYS-CFX to investigate and improve the flow characteristics inside the SCPP. Based on the CFX computational results, the best configuration was achieved using the chimney with a divergence angle of 2° and the temperature inside the collector is higher for the lower opening resulting in a higher flow rate and power. Additional, other researchers is used FLUENT software to optimize the SCPP by changing the collector inlet opening and outlet diameter of the chimney, and concluded that the available power was virtually unresponsive to the variation of collector inlet opening, as [10].

Pasumarthi1 and Sherif [11] used experimental and mathematical model to study the effect of various parameters on the air temperature, air velocity, and power output of the solar chimney. Two experimental configuration were tried on the collector: increasing the collector base diameter and introducing an intermediate absorber. Enhancing the air temperature is resulted from the former modification, while the latter contributed to increasing the air temperature as well as the mass flow rate inside the chimney. Both enhancements helped to increase the overall chimney power output. Comparison of mathematical model results to published data of the solar chimney system built in Manzanares, Spain has been used to validate the calculated results. Also, an economic assessment of the system costs are presented in reference [12].

Pretorius and Kröger [13] have developed comprehensive models to solve the governing conservation and draught equations simultaneously, and Bernardes et al [14] presented same techniques to predict the SCPP performance. The results show that the height of chimney, the factor of pressure drop at the turbine, the diameter and the optical properties of the collector are important parameters for the design of solar chimneys. Gannon and Von Backsröm [15] adapted the standard gas turbine cycle to define a standard solar chimney cycle, and the adaption includes chimney friction, turbine system and exit kinetic energy losses in the analysis.

The objective of this study is to accurately analyze the SCPP system by using CFD simulation model, with fewer assumptions are used in the theoretical calculation, but more detailed descriptions of pressure and flow field could be obtained. A 3D approach for SCPP prototype is carried out by using ANSYS CFX v18 with axial vertical flow turbine. The turbine is designed of using free vortex and the matrix throughflow method .The 3D numerical simulation incorporating the radiation models and turbine models. Results from the mathematical model were compared with Manzanares experimental results for model validation. Based on the proposed numerical approach, the effects of solar radiation, pressure extracted at the turbine, and mass flow rate on the SCPP system performance were investigated in detail.

2. Description of Solar Chimney

A typical SCPP consists of three major components, namely, a solar collector, a chimney and wind turbines, as shown in Figure 1. In a SCPP, solar radiation collected by the solar collector and greenhouse effect, which heats up the ambient air entering the collector; then the hot air flows into the chimney through the collector exit, which is also the entrance of the chimney. In the chimney, the air density difference between the inside and outside of the chimney causes a large pressure difference between the system and the ambient air (chimney effect), which will drive the wind turbine installed at the chimney base to generate electricity (wind turbine effect). The SCPP has no adverse effect on the environment, needs no cooling water and has extremely low maintenance costs. Furthermore, the SCPP can operate even at nights since the soil under the collector works, as a natural heat storage system. These distinct advantages have made SCPP an attractive option for utilizing solar energy.

Principle of solar collector

The major component of a solar chimney power station is the solar collector. Solar energy collectors are special kind of heat transfer that transform solar radiation energy to internal energy of the ambient air (transport medium). The collector is the part of the chimney that produces hot air by the greenhouse effect. It has a roof made up of plastic film or glass plastic film. This covering admits the short wave solar radiation component and retains long wave radiation from the heated ground. Thus, the ground under the roof heats up and transfers its heat to the air flowing radially above it from the outside to the chimney. The height of the roof increases adjacent to the chimney base so that the air is driven to the chimney base with minimum friction loss. However, the structure of a collector change according to consideration of architectural and civil design [16] .

Principle of updraft chimney

The chimney is the main characteristic of the solar chimney station. The tower, which acts like a large chimney, is located at the centre of the greenhouse canopy (collector). The tower utilize a temperature differential between the cool air at the top and the heated air at the bottom. The temperature difference create difference in density. This creates the chimney effect, which sucks air from the bottom of the tower out of the top. The chimney of the plant is extremely high and will need a stable base while still allowing free flow of air through the turbine. It would also be advantageous to have the turbine as low as possible in the chimney to make its construction simpler.

Principle of air turbines

The turbine of the solar chimney is key component of the plant as it extracts the kinetic energy from the air and transmits it to the generator. It has significant influence on the plant as the turbine pressure drop and plant mass flow rate are coupled. The specifications for solar chimney turbines are in many aspects similar to those ones for large wind turbines. They both convert large amounts of energy in the airflow to electrical energy. However, there are also various important differences. The following characteristic are typical for solar chimney turbines in contrast to wind turbines.

The typical solar chimney turbine is of the axial flow type. It has characteristics between those of wind turbines and gas turbines: it has more blades than the typical two or three of wind turbines, but not as many as gas turbines. The rotor blades are adjustable, like those of wind turbines, but as in gas turbines, the flow is enclosed, and some of solar chimney turbine may have radial vanes of inflow inlet guide. The main function of the turbine is the efficient conversion of fluid power to shaft power. A secondary function of solar chimney turbines is flow and output power control by adjustment of its blade angles [17].

This Characteristic of solar chimney turbines leads the researchers to take different method during the design of the turbine. Gannon and Von Backström [17], Denantes and Bilgen [18], and Fluri and von Backström [19] used the free vortex design. The free vortex approach is used in an axial flow gas turbine stage annulus and is assumed to be fully cascaded aerofoil (two dimensions) when the flow parameters are functions of two space coordinates, which the radial effect on the flow is ignored. On the other hand, Y Zhou et al. [20]used Wilson design theory to design of the SCPP turbine. Wilson design is classical theory to design horizontal axis wind turbine.

3. Turbine Design of the SCPP

Solar chimney literature has little to say about factors affecting efficiency of the turbines, but merely assumes various fixed values of efficiency in the range 40-80%, according to Backström & Gannon, [17] and Mullett [6]. In this study, a new approach is present for redesign of the solar chimney turbine. The Matrix Throughflow Method (MTFM) usually design the axial flow turbine and fan. The MTFM is a two-dimensional analysis tool that is effective in the design phase of turbine of SCPP. It simulates the machine as an axisymmetric duct with the blade rows represented by actuator discs or volumes. The MTFM considers the free vortex method assumed that all flow changes occurred within the blade row while in a real machine the velocity profiles change in the space ahead and behind the blades as well. Its main use in this analysis is to calculate the correct gas inlet and outlet angles to produce the correct amount of absorbed power from the fluid. In the next design step, profile shapes are designed based on these gas inflow and outflow angles. Using the MTFM method, the flow in the duct is also analysed to check and see that there are no adverse pressure gradients as the flow turns into the turbine as this could lead to separation at parts of the wall. This is achieved by using the interpolation method that allows values on a local orthogonal grid from the quasiorthogonal grid to be calculated using a simple linear interpolation method, by (Harms et al. [21] .This approach is used in an almost identical form in the application of the streamline through flow method (STFM) and detail of the implementation is given, which done in thesis presented by Gannon [22].

The Vista AFD under ANSYS Workbench v 18 is used to design the turbine of the SCPP. It is applied MTFM to design the axial fan. However, the design of the turbine is carried by entering the inlet flow condition as output of fan, and outlet flow condition is an inlet of fan. This condition is correct under free vortex design for 0.5 degree of reaction. Figure 2 shows the velocity triangle of free vortex at 0.5 degree of reaction in a turbine stage, where the blade angle is β, absolute flow angle is α, relative velocity is w, absolute velocity is c, and U is the blade speed. This condition allows reversing the turbo machine with same efficiency. The input aerodynamic parameter to Vista AFD is calculated from free model of the SCPP. The geometry parameter of the SCPP is also need to design the turbine, as shown in Table 1.

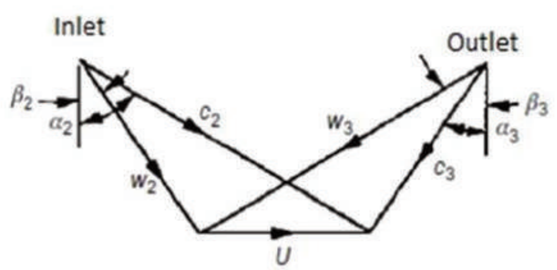

Figure 2. Velocity profile of half degree of reaction of the blade [23].

4. Simulation Model

A solar chimney power plant has many physics principles. The heating collector is work as air heater by solar energy, where all terms of heat transfer are applied to predict his performance. To analysis of chimney, the heat transfer is neglected but buoyancy force (fluid mechanics) is taken into account. The turbo machine theories should be applied to design and analyse the turbine.

Table 1. Boundary condition for Vista AFD

Input aerodynamic parameter	
Rotational speed (Ω)	100 rpm
Air Velocity (U_∞)	9 m/s
Inlet total pressure	92930 pa
Inlet total temperature	291.5 K
Total head rise	90 pa
Input geometry parameter	
Outer diameter	5 m
Hub\tip rotor	0.15
Number of the blades	4

The Computational Fluid Dynamic (CFD) involves the numerical solution of the differential governing equations of fluid flows and heat transfer, with the help of computers. This technique has a wide range of engineering applications. In the field of solar energy research, this technique has become increasingly important and it is prominent for studying the SCPP.

ANSYS-CFX v18 was used for simulation purpose in this research project. ANSYS-CFX v18 uses unsteady Navier– Stokes equation in their conservation form to solve set of equations. The instantaneous equation of mass (continuity), momentum, and energy conservation are presented below:

Mass conservation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{\upsilon}\right) = 0$$

where ρ is density of the fluid, $\vec{\upsilon}$ is the velocity vector, and t is time

Momentum conservation:

$$\frac{\partial\left(\rho \vec{\upsilon}\right)}{\partial t} + \nabla \cdot \left(\rho \vec{\upsilon} \vec{\upsilon}\right) = -\nabla p + \nabla \cdot (\overline{\overline{\tau}}) + \rho \vec{g} + \vec{F}$$

where p is the static pressure, $\overline{\overline{\tau}}$ is the stress tensor, $\rho \vec{g}$ is the gravity force and $\vec{F}$ are the other external body forces.

Energy conservation:

$$\rho \frac{De}{Dt} = \frac{\partial(\rho e)}{\partial t} + \nabla \cdot (\rho e \vec{\upsilon}) + \nabla \cdot (\vec{\upsilon} \cdot \tau) + \vec{\upsilon} \cdot S_m$$

where e is energy of the fluid, which write by total enthalpy $h + \upsilon^2/2$, the term $\nabla \cdot (\vec{\upsilon} \cdot \tau)$ represents the work due to viscous stresses and is called the viscous work term. This models the internal heating by viscosity in the fluid, and is negligible in most flows, and the term $\vec{\upsilon} \cdot S_m$ represents the work due to external momentum sources and is currently neglected.

In commercial codes, a friendly interface gives the user the possibility of easy setting the various options and analyzes the results. As an example, ANSYS CFX v18 that is used in present study. When approaching the study of fluid dynamics problems, the mathematical model is based on the fundamental mass, momentum and energy conservation principles. The Reynolds's Averaged Navier-Stokes (RANS) equations for the compressible fluid flow are included the equations of the conservation of mass and momentum. Therefore, the conservation laws are invoked in the following, in the vectorial notation and conservative form for unsteady, three–dimensional compressible flow [24].

For the prediction of solar chimney performance in this study, the k- ε turbulence models have been chosen which type of Two-equation models. Two-equation models have been the most popular models for a wide range of engineering analysis and research. These models provide independent transport equations for both the turbulence length scale, or some equivalent parameter, and the turbulent kinetic energy. With the specification of these two variables, two-equation models are complete; no additional information about the turbulence is necessary to use the model for a given flow scenario. While this is encouraging in that these models may appear to apply to a wide range of flows, it is instructive to understand the implicit assumptions made in formulating a two-equation model [25].

As mentioned, the SCPP unit consists of ground, cover collector, chimney and fluid zone. All component geometry is drawn as 3D using SOLIDWROKS, as shown in Figure 3 (light green is Cover collector, blue is Fluid zone and yellow is ground area). The ANSYS Design Modeler is used to enter CAD file, which is a part of ANSYS Workbench 18. It able to read CAD file and transfer data reading to grid generation program such as Meshing or turbo grid. However, ANSYS Meshing is used that suitable for SCPP geometry. This technique has many advantages, reading shape with high accuracy and save time and labor needing to enter data.

Grid generation converts the geometry into a format that can be understood by the CFD solver. It is often the most time consuming and tedious jobs in achieving the CFD solution. Grid generation is the most important before pre-processing step. It is very important to generate accurate grids for the solver to obtain correct results. The accuracy of the CFD solution depends on the quality of the grid used to perform the calculations. Figure 4 show the overall model gird.

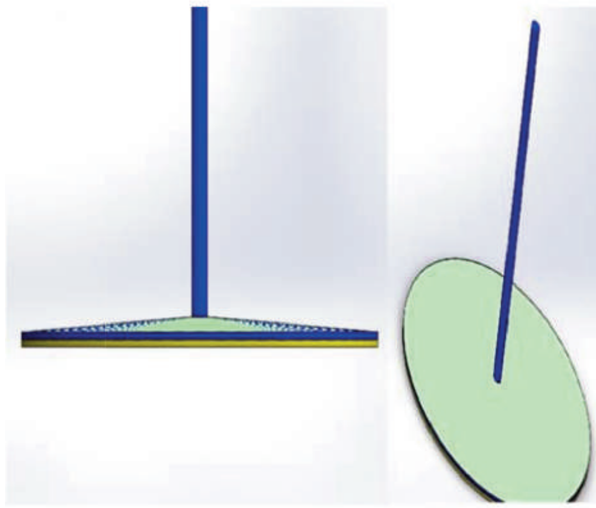

Figure 3. The 3D geometry of difference zone for the solar chimney unit.

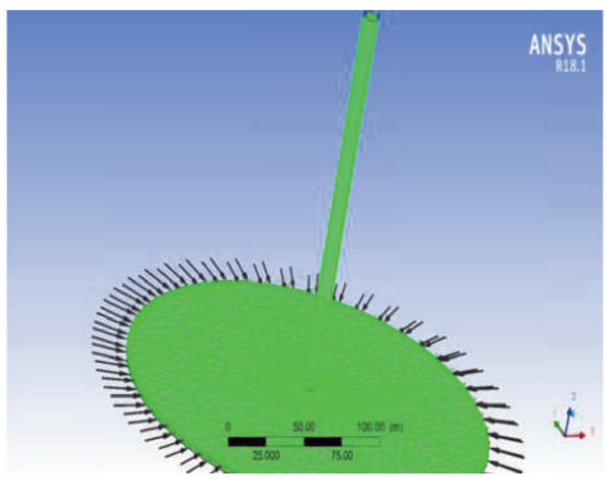

Figure 4. Grid of the solar chimney unit.

For generation turbine grid, turbo grid program is used that is specialized in turbo machine grid generation. One passage of flow is generated and the flow regions hub, shroud, inlet and outlet are defined. Figure 5 shows the turbo grid passage that used H/J/C/L-grid to make the flow region and O-grid in close to blade surface. This passage has rotating boundary condition.

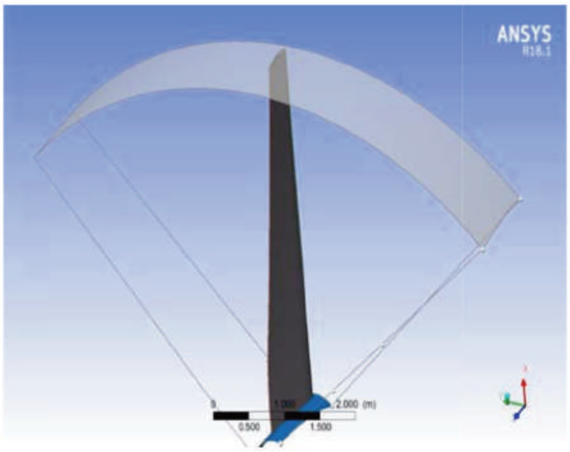

Figure 5. Turbo grid flow passage.

The interface between these two meshes is the GGI interface. The frozen rotor model is type of GGI interface, which available to use in ANSYS CFX v18.The two frozen rotor interfaces are used in this model. The first one is used between the passage inlet and side outlet from collector. The second interface is used between the outlet of passage and under surface of the chimney.

5. Results and Discussion

The axial turbine is designed by using the operation condition in the Table 1. Vista AFD software is carried out free vortex analysis to find a first approximation of the blade configuration. The next step of the analysis is to perform an axisymmetric simulation of the turbine. A Matrix Throughflow Method (MTFM) is used for this purpose, which able to handle the radial inflow and axial outflow that are found inside the solar chimney turbine. The airflow inflow and outflow angles from the turbine is then calculated by the MTFM approach. The values are slightly different from the free vortex prediction. From this calculation, the blade angle and chord distribution can be obtained for the most efficient design possible over the required operating range. Figure 6 show the 3D rotor geometry of the designed SCPP turbine. The BladeGen program under ANSYS Workbanch v18 is used to present and modify the thickness distribution of the designed blade.

Beta angle is airflow angle of the designed blade, which is shown at leading and trialing edge along of the blade by Figure 7. In hub section generate (Range 0 to 0.5 of the blade length), the sharp change of the angle specifically at leading edge, the high twist angle of the blade. The blade with high twist angle has some difficult in his fabrications. Thus, more optimization will be required to reach the efficient performance and easy to fabrication.

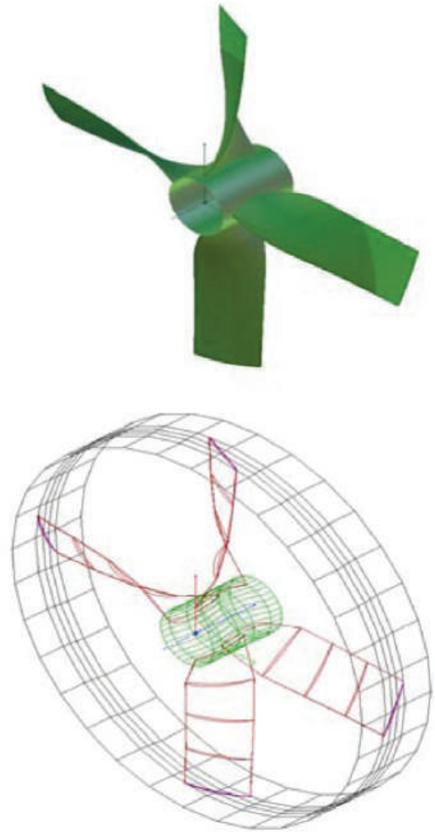

Figure 6. 3D axial turbine for the SCPP unit.

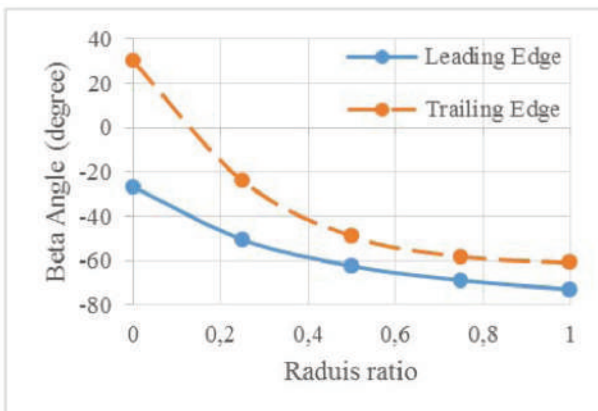

Figure 7. Beta angle at leading and trailing edge along the blade.

The blade thickness has serious influence in the turbine performance and toughness of the blade. NACA 0006 is selected to able the blade to perform well. Another factors are shape of NACA 0006 is easy to manufacture and the ratio of lift to drag coefficient in the work range of the SCPP turbine. The NACA 006 thinness is applied to the designed turbine by BladeGen software. Table 2 show the final chord distribution and twist angle of the design blade. The rotor is consisted from 4 blades that is described in Table 2.

Table 2. Designed blade configuration.

Span of the blade	Chord length (mm)	Twist Angle (degree)
0.00	1439.45	1.8
0.25	1442.91	-38.0
0.50	1441.65	-56.1
0.75	1452.01	-63.9
1.00	1470.66	-67.5

To validate the CFD model, calculated results are compared with the experimental results of the prototype from Manzanares, Spain. The measured data on September 2nd, 1982 are adopted from Reference (Haaf, 1984). The plant dimensions are given in Table 3. The comparisons between the calculation results and the experimental values are presented in Table 4. There is good agreement between measurement and calculation results, which are acceptable value.

Table 3. Geometrical dimensions of the pilot plant in Manzanares, Spain.

Mean roof radius	122 m
Average roof height	1.85 m
Tower height	194.6 m
Tower radius	5.08 m

Table 4. Comparison between measured data and mathematical model results.

	Measured	Calculated
Difference temperature at collector ,°C	17.1	14.25
Upwind Velocity , m/s	9.045	9.11784
Power, P , kW	36	36.1552

The air velocity is calculated at different locations in the all domain of the SCPP unit. The upstream and downstream lines of the air velocity are calculated. As shown in Figure 8, the maximum velocity according at near close to the chimney entering, and uniform airflow in the collector region. The range of the velocity in the chimney is approximately 4 times the range of velocity in the collector. To capture more output power, the turbine should be located at high air velocity, which is achieved in near enter of the chimney.

Figure 8 illustrates contours of the pressure distribution at different zones of the chimney in a vertical intersection. The pressure around turbine has significant influence of the SCPP performance. Figure 9 shows that the pressure increases gradually as the air is flowing inside the chimney. It indicates also that the minimum static pressure is reached near the chimney base at the exit of the turbine due to extract the energy from the flow by the turbine.

The blade of an axial turbine flow depends on the airfoil theory. The pressure distribution around the airfoil surface controls in the lifting properties of the airfoil. Figure 10 show the pressure distribution at different lengths of the blade. As shown, the tip section of the blade has enough large pressure difference between the upper and lower surface. On other hands, the hub section has less pressure difference between its surfaces. Consequently, static pressure increases from root to tip. The degree of reaction control of the pressure distribution, which increases from root to tip according to free vortex assumption. However, the power production is constant a long of the blade. The energy production and degree of reaction is an agreement of the free vortex principle.

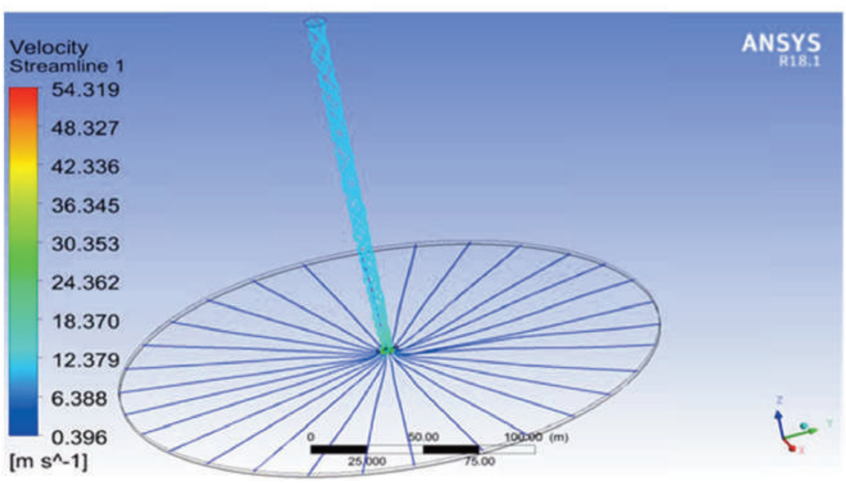

Figure 8. Streamline of air in SCPP model.

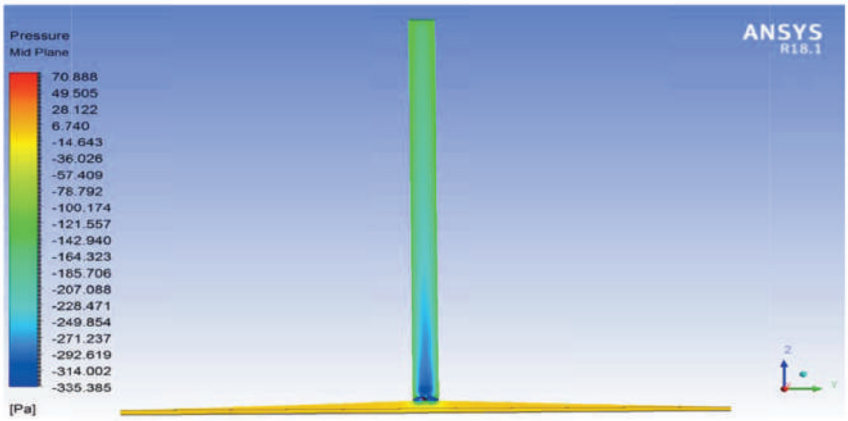

Figure 9. Contours of static pressure at different zones of the chimney.

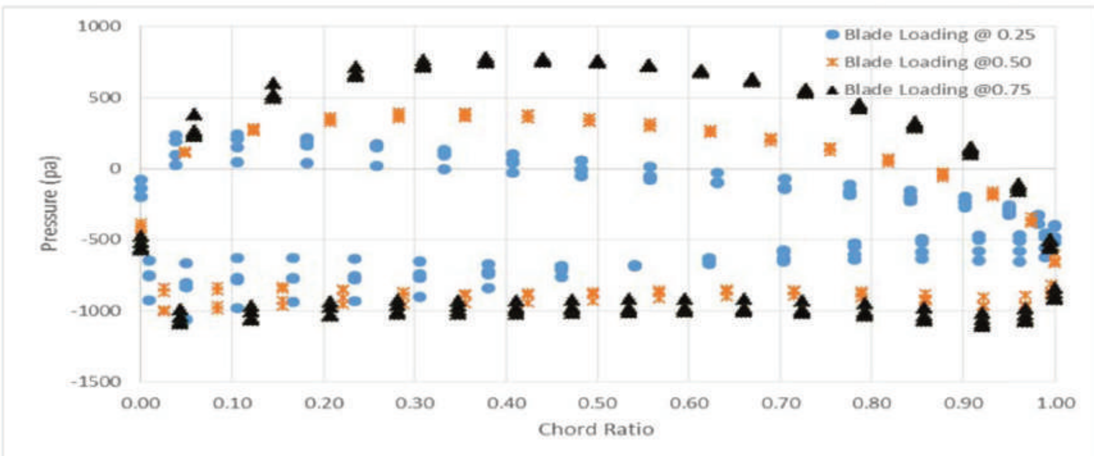

Figure 10. Pressure distribution at different length of the blade.

6. Conclusion

In this study, the main aims were: design and analysis of the axial flow turbine performance for solar chimney power plant. The advantage of free vortex design is used to calculate the geometric shape (chord distribution and twist angle) for the turbine blade. To simulate the turbine of the solar chimney, the standard k-ε turbulent and radiation model within ANSYS-CFX v18 is chosen, which predicts its performance. The comparison the CFD result and previous experimental data reveals a good agreement.

The results have shown that free vortex theory and Matrix Throughflow Method have reasonable accuracy to obtain the good blade shape. The designed blade can be easily manufactured. However, the hub section needs more work in the fabrication phase. NACA 0006 airfoil is very close to thickness distribution calculation of MTFM. The characteristics of the NACA 0006 match the working condition of the SCPP. Thus, makes it is suitable to design the turbine for the SCPP.

The present study conclude the capability of CFD model, as a powerful research tool and engineering analysis method for analysis of complex thermal and aerodynamic system, such as the solar chimney power plant. The theoretical modeling approach can help to create an experimental study of a solar chimney plant to make it more effective and economical.

Acknowledgement:

"This work was supported by the Stipendium Hungaricum Programme and by the Mechanical Engineering Doctoral School, Szent István University, Gödöllő, Hungary."

References

[1] Zhou X., Wang F., Ochieng R. M.: 2010. A Review of Solar Chimney Power Technology. Renewable and Sustainable Energy Reviews, Vol. 14, No. 8, pp. 2315-2338. http://dx.doi.org/10.1016/j.rser.2010.04.018

[2] Haaf W., Friedrich K., Mayr G., 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. http://dx.doi.org/10.1080/01425918308909911

[3] Haaf W.: 1984. Solar chimneys; Part II: Preliminary Test Results from the Manzanares Pilot Plant. International Journal of Solar Energy, Vol. 2, No. 2, pp. 141-161. http://dx.doi.org/10.1080/01425918408909921

[4] Mekhail T., Elmagid W. M,, Fathy M., Bassily M., Harte R.: 2016. Theoretical Investigation of Solar Chimney Power Plant Installed in Aswan City. In International Symposium on Industrial Chimneys and Cooling Towers, Rotterdam, Germany, October 5-8, 2016.

[5] Lu F., Zhang H., Yao X.: 2006. Review of key technologies for solar chimney power generation, East China Electric Power, 2006.

[6] Mullett L. B.: 1987. The Solar Chimney—overall efficiency, Design and performance. International Journal of Ambient Energy, Vol. 8, No. 1, pp. 35-40. http://dx.doi.org/10.1080/01430750.1987.9675512

[7] Yan M. Q., Sherif S. A., Kridli G. T., Lee S. S., Padki M. M.: 1991. Thermo-fluid Analysis of Solar Chimneys. In Industrial Applications of Fluid Mechanics, Atlanta, GA, 1991.

[8] Huang H., Zhang H., Huang Y., Lu F.: 2007. Simulation Calculation on Solar Chimney Power Plant System. In International Conference on Power Engineering, Hangzhou, China, 2007.

[9] Patel S. K., Prasad D., Ahmed 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. http://dx.doi.org/10.1016/j.enconman.2013.09.056

[10] Vieira R. S., Garcia C., Junior I. C. A., Souza J. A., Rocha L. A. O., Isoldi L. A., dos Santos E. D.: 2015. Numerical Study of The Influence of Geometric Parameters on The Avaliable Power in a Solar Chimney. Thermal Engineering, Vol. 14, No. 1, pp. 103-109.

[11] Pasumarthi N., Sherif A. S.: 1998. Experimental and Theoretical Performance of a Demonstration Solar Chimney Model-Part I: Mathematical Model Development. International Journal of Energy Research, Vol. 22, pp. 277-288. http://dx.doi.org/10.1002/(SICI)1099114X(19980310)22:3<277::AID-ER380>3.0.CO;2R

[12] Pasumarthi N., Sherif A. S.: 1998. Experimental and Theoretical Performance of Demonstration Solar Chimney Model- part II: Experimental and Theoretical Results and Economic Analysis. International Journal of Energy Research, Vol. 22, pp. 443- 461. http://dx.doi.org/10.1002/(SICI)1099114X(199804)22:5<443::AID-ER381>3.0.CO;2-V

[13] Pretorius J., Kröger D.: 2006. Solar chimney power plant performance. Journal of Solar Energy Engineering, Vol. 3, No. 128, p. 302–311. http://dx.doi.org/10.1115/1.2210491

[14] Bernardes M., Voss A., Weinrebe G.: 2003. Thermal and technical analyses of solar chimneys. Solar Energy, Vol. 75, pp. 511–524. http://dx.doi.org/10.1016/j.solener.2003.09.012

[15] Gannon A., von Backström T.: 2000. Solar chimney cycle analysis with system loss and solar collector performance. Journal of Solar Energy Engineering, vol. 122, p. 133–137. http://dx.doi.org/10.1115/1.1314379

[16] Dhahri A., Omri A.: 2013. A Review of solar Chimney Power Generation Technology. International Journal of Engineering and Advanced Technology (IJEAT), Vol. 2, No. 3, pp. 1-17.

[17] Gannon A. J., von Backström T.: 2002. Solar Chimney Turbine Part 1 of 2: Design. In 2002 International Solar Energy Conference, Reno, Nevada, USA, June 15-20, 2002.

[18] Denantes F., Bilgen E.: 2006. Counter-Rotating Turbines for Solar Chimney. Renewable Energy, Vol. 31, pp. 1873–1891. http://dx.doi.org/10.1016/j.renene.2005.09.018

[19] Fluri T., von Backström T.: 2008. Comparison of Modelling Approaches and Layouts for Solar Chimney Turbines. Solar Energy, Vol. 82, No. 3, pp. 239–246. http://dx.doi.org/10.1016/j.solener.2007.07.006

[20] Zhou Y., Gao B., Dong H. R., Hao K.: 2016. Design for the Turbine of Solar Chimney Power Plant System with Vertical Collector. IOP Conf. Series: Earth and Environmental Science, Vol. 40, No 1. 012085. http://dx.doi.org/10.1088/17551315/40/1/012085

[21] Harms T., von Backström T., du Plessis J.: 1996. Simplified Control Volume Finite-Element Method. Numerical Heat Transfer, Vol. B, No. 30, pp. 179-194. http://dx.doi.org/10.1080/10407799608915078

[22] Gannon A. J.: 2002. Solar Chimney Turbine Performance. University of Stellenbosch (PhD Thesis), Matieland, South Africa, March 2002.

[23] Dixon B. S. L., Hall C. A.: 2014. Fluid Mechanics and Thermodynamics of turbomachinery (Seventh Edition), Oxford, UK: Elsevier Inc., pp. 556, 2014.

[24] ANSYS CFX Introduction User Help.

[25] Celik I. B.: 1999. Introductory Turbulence Modeling," Mechanical & Aerospace Engineering Dept. West Virginia University (Lectures Notes), Morgantown, Dec 1999.