Experimental and numerical investigations of a novel chimney system for power generation using the combination of fossil fuel power plant exhaust gases and ambient air

Bagher Mokhtari Shahdost1 | Fatemeh Razi Astaraei1 | Amir Ebrahimi-Moghadam2 | Mohammad Hossein Ahmadi2

Abstract
Fossil fuel power plants, one of the main sources of power generation, are the major emitters of exhaust gases into the environment at relatively high temperature and significant energy. One way to utilize these hot gases is to use wind turbines in the path of fluid flow that created by buoyancy (like a solar chimney). In this paper, the power generation from the gas of the power plant's chimney based on the buoyancy phenomenon using the combination of the power plant's hot output with ambient air is modeled and investigated at different discharge rates and temperatures of the power plant's hot output on a pilot scale. An experimental model, at the same scale of modeling, is constructed and used to validate the modeling results. The obtained results showed that due to the buoyancy phenomenon the combination of the power plant's exhaust gases with the ambient air creates a higher volume of the fluid with a good velocity inside the chimney compared with the exhaust gases of the power plant only. At the temperature of 350 K and velocity of 1 m/s for the hot inlet, the velocity of the fluid resulting from the buoyancy would be greater than 1 m/s for the air which enters the chimney's foot and its gas.

KEYWORDS
buoyancy, chimney, computational fluid dynamics, power plant, solar chimney power plant, turbulent flow

1 | INTRODUCTION
In recent years, the energy crisis and its related problems has become an important issue in the world, and the supply of energy resources and power generation methods are one of the most important issues in each country and have overwhelmed all political and economic issues. The limitation of fossil fuels, environmental problems, population growth, economic growth, and consumption ratios are critical issues that require appropriate solutions.1-6 Therefore, research on power generation methods,7,8 performance optimization,9,10 overall efficiency,11,12 reduction in environmental pollutants,13-15 thermal and energy recovery,16,17 and the use of new energies18 in power plants has a special significance, as these are used in the formulation of policies, performance improvement, and increasing the productivity of existing sectors as well as designing new systems. Power plants are one of the major sources of energy production. These plants ensure that the energy needs of human are fulfilled. The human health is at risk due to the
pollutants generated by these power plants. Therefore, one of the main problems in power plants is to control and prevent the excessive emission of pollutants, especially sulfur dioxide and nitrogen oxides. The exhaust gas from the power plant has a temperature of around 150°C, which is a significant energy source that can supply some of the thermal needs for the energy units in the power plant. Thus, the implementation of such a project results in significant amount of energy savings.

One of the methods of controlling air pollution is the recovery of heat energy from the exhaust gases from the chimney. This can be done by constructing high-length chimneys at the exhaust of power plants and by placing the wind turbines in the direction of fluid flow generated by the buoyancy phenomenon.

Therefore, the solar chimney technology can be introduced in this regard. The first concept of the solar chimney was introduced by Gunther in 1931. The solar chimney is a simple system that generates electricity based on the greenhouse effect and rise of the hot air using solar energy in the tower and launching the system's turbine. In the other words, this technology can be defined as using the solar energy and convection flow created by the density difference in wind production. Based on this model, a solar chimney with a height of 200 m was constructed and tested in 1980 in Manzanares, Spain; its energy equilibrium, cost, and design analyses are presented by Haaf et al. Bernardes et al developed a numerical model of a solar chimney power plant and compared its results with the experimental data from the Manzanares power plant. Ghorbani et al designed a novel system based on the combination of a solar chimney and a dry cooling tower for increasing the thermal efficiency of Rankin cycle steam power plant. Fluri et al analyzed several cost models for large solar power plants. Okoye and Atikol studied the feasibility of solar chimney location parameters. Guo et al performed a yearly performance analysis for a solar chimney. They used an analytical approach and numerical modeling to investigate the optimal turbine pressure drop in a solar chimney and showed that sunlight and ambient air temperatures are among the most important parameters affecting the rate of turbine pressure drop. Li et al investigated the effects of collector diameter and solar radiation on optimal turbine pressure drop and found a positive relationship between different parameters. Zhou et al provided a new combination of solar chimney with a hole in the mountain (mountain hollow), which excluded chimney construction and costs in that study. In another study by Zhou et al, a model was to verify the power output of a solar chimney power plant with regard to solar radiation, chimney height, and collector area. Patel et al applied computational fluid dynamics (CFD) simulation and examined the influence of the geometrical parameters to enhance the flow characteristics of a solar chimney power plant. Koonsrisuk et al evaluated the height and diameter of the solar chimney to obtain the maximum mass flow rate. In another study, Hamdan proposed a mathematical model for steady-state air flow inside a solar chimney using the Bernoulli equation with buoyancy effect; their results showed that the chimney height, collector radius, solar radiation, and turbine head are the main parameters for the design of a solar chimney. Shariatzaodeh et al conducted a feasibility study for a hybrid solar chimney/fuel cell power plant. Fasel et al presented a CFD modeling of a solar chimney using ANSYS Fluent software to conduct numerical study for some solar chimneys (including a real one at the University of Arizona) with different tower heights.

Khelifi et al developed a comprehensive analytical model for solar chimney with and without a turbine to calculate the temperature distribution of adsorbent and transparent coating and also air temperature. Therefore, the effect of chimney height and collector diameter on the performance of solar chimney power plant was investigated. Sakonidou et al presented a new mathematical model that estimated the optimum tilt angle of the solar chimney in order to obtain the maximum efficiency. Mehrpooya et al developed a two-dimensional axisymmetric solar chimney without a turbine; they used previously developed relationships to calculate the power of existing turbines over a year for this model. Kasaeani et al conducted a numerical and analytical study to optimize the geometry of a prototype solar chimney made at Tehran University, Iran. The results showed that the height and diameter of the chimney are one of the most important parameters for designing a solar chimney. In another work, Kasaeani et al applied a three-dimensional CFD simulation for modeling the prototype of the Manzanares solar chimney power plant (with turbine), and compared and verified their results with experimental data. Afterward, they investigated the effect of the turbine's blades number, chimney height, and collector diameter, and concluded that increasing the chimney height and collector diameter exacerbates the mass flow rate and output power. Ghalamchi et al investigated the effect of electrohydrodynamic systems with different electrode layouts in a pilot solar chimney. The results of their research showed that the electrohydrodynamic system increases air flow rate and improves the solar chimney performance. Ming et al reviewed new technologies about solar updraft power plant systems (SUPPSs) and considered a case study for investigating the effect of the ambient cross wind on a large-scale SUPPS. They studied three different designs for the studied system: a conventional horizontal canopy, a familiar sloped canopy, and a collector canopy with eight radial partition walls under it. Renaud de Richter et al proposed a review of combining photocatalytic reactor with a solar chimney. In these systems, by locating photocatalysis in the air flow path below the collector of the solar chimney power plan (SCPP), in addition to producing renewable electricity by turbines, the photocatalysis cleanse and absorb the greenhouse gases (GHGs). It was also stated in this study that the
The purpose of combining the TiO$_2$ photocatalysis with the solar SCPP was to cleanse the atmosphere of non-CO$_2$ GHGs.

The use of power plant's chimney to generate power using the buoyancy phenomenon is a new field for research. In this study, first, a novel scheme of the using power plant's chimney for the power generation with solar technology is presented. The proposed system, unlike solar chimney power plants, does not require a collector and an absorber; the system consists of three main parts: the exhaust of the power plant, chimney (tower), and turbines. This study composed of experimental and numerical studies. In the first step, the prototype of the chimney is constructed at the University of Tehran campus on the gas of the pilot-scale power plant. Afterward, an accurate CFD simulation is built and the results of the numerical model are validated with the obtained experimental data in the previous step. After validating the results, the numerical procedure is also applied to describe the flow characteristics (temperature, velocity, and pressure distribution) inside the chimney, and the effects of parameters such as the flow rate and temperature of the power plant's exhaust (the hot gases entering the chimney) are described.

2 | RESEARCH METHODOLOGY

As shown in Figure 1A, in the solar chimney power plant, air is heated underneath a transparent ceiling that passes through the solar radiation. It should be noted that the existence of this ceiling and the ground below it acts as a solar collector. A chimney or vertical tower is placed in the middle of this transparent ceiling, through which a vast amount of air enters from its bottom. The connection between the transparent ceiling and the tower must be “air locked” that is it should be devoid of any pores. The hot air moves up the tower because it is lighter than cold air; this movement causes suction at the bottom of the tower to drain more amount of the hot air inside the tower so that the surrounding cool air enters the transparent ceiling. Therefore, this work is based on the concept that solar radiation in the tower causes suction on the top which generates energy; this energy is converted into mechanical energy by the turbine embedded in the tower in several stages and then converted into electrical energy. In the solar chimney of the Manzanares power plant (Spain), at the radiation flux of 1000 W/m$^2$, the maximum temperature difference between the temperature of the air in the collector and the ambient air is about 20°C, which, creates an airflow with velocity of 15 m/s inside the tower due to buoyancy of air.$^{22}$

In the combined cycle power plant, the temperature at the chimney’s output is about 100-150°C, and it is about 400-450°C for the gas turbine plant, which is higher when compared to the ambient temperature. If the high-temperature output of the power plant is combined with the ambient air, there will be a difference in temperature of 20-30 degrees than the ambient temperature, and consequently, the volume of hot air will increase significantly. This concept has been used in the new proposed system in this study (see Figure 1B). As shown in Figure 1B, the hot gases from the exhaust of the power plant move toward the tower's foot. The hot gases flowing from the central tube into the chimney's (tower) foot are driven upward by the buoyancy force and this phenomenon causes a pressure drop at the tower’s foot; this pressure drop results in the suction of the ambient air from the central tube toward the tower’s foot. Finally, the ambient air entering the tower's foot is combined with the hot gases from the exhaust of the power plant and the resulting mixture moves toward the top of the chimney, and is continued in the solar chimney power plant. The kinetic energy of fluid is then converted into mechanical energy by
the embedded turbine in the tower and finally into electrical energy. The advantage of combining the power plant's hot exhaust gases with the ambient air entering the tower is that the volume of air increases significantly due to the buoyancy phenomenon compared with the power plant's hot exhaust gases only.

This system does not require collectors and absorbers in comparison to the solar chimney power plant system. This system consists of three main parts, namely the power plant output, chimney, and turbine. As shown in Figure 1B, the turbine for the generation of the power is placed in the path of the ambient air entering the chimney or at the exit of the chimney. If the turbine is not corrosion-resistant against the exhaust gases from the fossil fuel power plant, then the turbine can be placed at the inlet of the chimney. As the flow path of the inlet air into the chimney is circular, the turbines are placed in a circular path between the gas of the power plant's exhaust (high-temperature fluid inlet) and the chimney. If the turbine is corrosion-resistant against the exhaust gases from the power plant, the turbine can be placed in the direction of the mixed fluid at a certain distance from the chimney's foot or at the gas of the chimney because the cross section of the mixed fluid flow is larger compared to the inlet ambient air cross section entering the chimney's foot.

A new method for generating power from the power plant's exhaust utilizing a chimney without a turbine at the exhaust of a power plant in a pilot scale is proposed in this paper.

3 | GOVERNING EQUATIONS

The introduced system had just one chimney that is located at the output of the power plant's exhaust, where the energy level inside is higher than the surrounding environment. The air flow moves upward from the inside of the chimney based on the buoyancy phenomenon. The fluid flow inside the chimney can be described based on the governing equations. Except the buoyancy formula, the fluid properties are assumed to be constant along the chimney. Also, changes in air flow velocity, air compression, and heat dissipation from the chimney wall are negligible. The governing equations are presented as follows:

Continuity equation:
\[
\frac{\partial \rho}{\partial t} + \frac{\partial \left( \rho u_i \right)}{\partial x_i} = 0
\]  

(1)

Navier-Stokes Equation:
\[
\frac{\partial \left( \rho u_i \right)}{\partial t} + \frac{\partial \left( \rho u_i u_j \right)}{\partial x_j} = \rho g_i - \frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}
\]  

(2)

**Energy equation:**
\[
\frac{\partial \left( \rho c_p T \right)}{\partial t} + \frac{\partial \left( \rho c_p u_i T \right)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \lambda \frac{\partial T}{\partial x_i} \right) + \tau_{ij} \frac{\partial u_i}{\partial x_j} + \beta T \left( \frac{\partial p}{\partial t} + u_i \frac{\partial p}{\partial x_i} \right)
\]  

(3)

Considering the physics of the flow inside the chimney, the turbulent flow is assumed in this study. The standard k-ε turbulence model, the most common model which employs the buoyancy effect with good accuracy, is utilized for modeling turbulence in this work. This model uses two additional equations for modeling turbulence. These two equations are the turbulent kinetic energy (k) equation (Equation 4) and the turbulent dissipation rate (ε) equation (Equation 5).

\[
\frac{\partial}{\partial t} \left( \rho k \right) + \frac{\partial}{\partial x_i} \left( \rho k u_i \right) = \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + G_k + G_b + \rho \varepsilon + Y_M + S_k
\]  

(4)

\[
\frac{\partial}{\partial t} \left( \rho \varepsilon \right) + \frac{\partial}{\partial x_i} \left( \rho \varepsilon u_i \right) = \frac{\partial}{\partial x_i} \left( \mu_t + \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right) + C_{1\varepsilon} \frac{k}{\varepsilon} \left( G_k + C_3 \varepsilon \right) - C_{2\varepsilon} \rho \frac{k^2}{\varepsilon} + S_\varepsilon
\]  

(5)

where \( Y_M \) is the fluctuating dilation that occur in compressible turbulence, \( G_k \) is the turbulent kinetic energy generated due to the gradients of mean velocity, \( G_b \) is the turbulent kinetic energy generated due to the buoyancy, \( \mu_t \) is the turbulent viscosity, and \( S_k \) and \( S_\varepsilon \) are the user-defined source terms. All the mentioned parameters are defined in Equations 6-9).

\[
Y_M = 2 \rho \varepsilon M_i^2
\]  

(6)

\[
G_k = -\rho \mu_t u_i \frac{\partial u_i}{\partial x_i}
\]  

(7)

\[
G_b = \beta g_i \frac{\mu_t}{\rho T_i} \frac{\partial T}{\partial x_i}
\]  

(8)

\[
\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon}
\]  

(9)

In these equations, \( C_k = 0.09, C_{1\varepsilon} = 1.44, \) and \( C_{2\varepsilon} = 1.92 \) are the experimental constants. The parameters \( \sigma_k = 1.0, \) \( \sigma_\varepsilon = 1.3, \) and \( \beta = 0.85 \) are known as the turbulent Prandtl numbers for \( k, \varepsilon, \) and energy, respectively. Also, the parameter in Eq. (6) represents the turbulent Mach number, defined as \( M_i = \sqrt{G_{hk}/a^2} \) (where \( a \) is the speed of sound).

4 | CONSTRUCTING AN EXPERIMENTAL SETUP AND CFD MODELING

A pilot-scaled prototype of the chimney on the power plant output is constructed at the campus of the University of
Tehran. Figure 2 shows the CAD design model which consists of a chimney with a height of 3 m and diameter of 10 cm; the high-temperature output gases from the exhaust of the power plant enters the chimney foot through a pipe with 5 cm diameter. In order to assemble the chimney, three galvanized pipes with length of 1 m are used, which were covered with a layer of fiberglass for insulating purposes. For the high-temperature fluid section, acting as the gas of a power plant’s exhaust, a 1600 W hairdryer is used and the heated air flows through a polyethylene pipe (with a layer of fiberglass insulation) towards the tower’s foot. In order to control the temperature of the gas fluid from the hair dryer, its air intake is altered. In order to control the velocity of the heated fluid by the hair dryer, a three-way path is designated on the direction of the hot fluid toward the tower’s foot. The first path was that the hot fluid moves toward the tower’s foot and the second path was that it was discharged into the surrounding atmosphere. By changing the second path, the velocity of the hot fluid at the tower’s foot can be controlled.

A foundation is considered to hold the chimney and the hair dryer. Different views of the designed system on the pilot scale are shown in Figure 3. In order to measure the temperature and velocity of the fluid, thermal sensors (SMT-160 model) and hot wires (YK-2004AH model) are employed at different points.

Besides the experimental studies, an accurate numerical modeling is carried out using the ANSYS Fluent 16 software package to investigate the distribution of velocity, temperature, and pressure inside the chimney and some of the other parameters. In the CFD method, according to the symmetry of the system, the model is designed as a two-dimensional and axisymmetric swirl model to reduce the computational process time. The scale of the CFD model was exactly the same as the scale of the experimental setup with a radius of 5 cm and a height of 3 m, as well as a hot air entrance to the tower’s foot with a radius of 2.5 cm. The assumptions of the boundary conditions in the numerical modeling are the uniform velocity profile for the hot inlet fluid and the ambient air at the tower’s foot, and the pressure at the gas boundary condition (atmospheric pressure) is considered for the tower’s gas. Also, the numerical modeling was performed at various inlet conditions (including different flow rates and temperatures of the hot fluid entering the chimney’s foot) and their effects are analyzed in next sections. Details of the numerical simulation are listed in Table 1.

A sample of the grid generation is shown in Figure 4A. Due to the large gradient of the flow characteristics around the walls and also vortexes in these areas, considering a high-resolution mesh around the walls has a significant impact on the success of the simulation. In this study, all Quad meshes are utilized for the grid generation. Also, a high-quality mesh is considered near the walls. A grid independent study for analyzing the sensitivity of results to the computational cells number is carried out as shown in Figure 4B (all assumption is the same for different grid sizes in this analysis). As shown in this figure, there is a very negligible change in the modeling results for the geometries with more than 7000 cells. As shown in this figure, the percentage of the relative difference in the amount of the gas velocity is 0.06% between geometries with 7384 cells and 14 749 cells.

5 | RESULTS AND DISCUSSION

5.1 | Validation

To verify the modeling results, it is necessary to compare them with the experimental data obtained from the experimental model. The testing of the experimental model has been performed on a winter day with the ambient temperature of 290 K and a very low wind flow.

To extract the experimental results, the data of the speed and temperature were recorded by sensors for at least 10 minutes after the hair dryer was turned on (for letting the system reach the thermal equilibrium). In order to ensure that the accuracy of the experimental results and to reduce the risk of measurement errors and the error of the temperature and speed sensors, the temperature and speed measurements were performed for 3 days at the same environmental conditions and the same results were obtained. The percentage of relative difference (as error) between the experimental and numerical modeling is calculated as follows:

\[
\text{ERROR} (\%) = \left| \frac{\text{Experimental result} - \text{Numerical result}}{\text{Experimental result}} \right| \times 100
\]

(10)

The comparison between the numerical and experimental results has been performed for two different cases:
1. In the first case, the experimental model has been tested for measuring the velocity of the ambient air entering the chimney’s foot at a constant velocity of the hot inlet into the chimney (1.75 m/s) and at three different temperatures of 330, 350, and 370 K. The results of this validation process and the percentage of the relative difference between the experimental and CFD results are presented in Figure 5A, B, respectively. As shown in Figure 5A, there is a good agreement between the results of the experimental data and CFD results, and the average difference between these two results at the three different temperatures is approximately 9%.

2. In the first case, the experimental model has been tested for measuring the velocity of the ambient air entering the chimney’s foot at a constant temperature of the hot inlet into the chimney (350 K) and at three different velocities of 1, 1.75, and 2.5 m/s. The comparison between the experimental data and CFD modeling is shown in Figure 6A for three different speeds, with an average error of 6% (Figure 6B). The difference between the experimental and CFD modeling depends on various factors such as accuracy of measuring instruments, velocity and temperature, modeling accuracy, environmental airflow, thermal losses, and so on. Given the comparisons are performed at different conditions for the experimental and numerical results, the accuracy of the results of modeling can be verified with good accuracy.

5.2 | Analyzing the distribution of velocity, temperature, and pressure inside the chimney

The distribution of temperature, velocity, and pressure are the three main factors to be observed in the chimney. As mentioned in the previous section, the dimensions of the chimney modeled in the ANSYS Fluent software are same as the dimensions constructed at the University of Tehran (experimental setup). In the initial modeling, for investigating the velocity, temperature, and pressure distribution inside the chimney, the ambient temperature, the hot inlet temperature at the chimney’s foot, and hot inlet velocity are considered to be 290, 350 K, and 1.75 m/s, respectively. The temperature and relative pressure distribution for the fluid flow inside the chimney are drawn as contour plots in Figures 7 and 8, respectively. The temperature contour indicates that the hot
inlet at the chimney’s foot moves up the chimney due to the buoyancy phenomenon (Figure 7), and this causes a relative drop in pressure at the chimney’s foot (Figure 8). The pressure increases when the hot inlet moves upward, and when it reaches the chimney’s top, it will be equal to the ambient pressure at the exit (zero relative pressure). Considering the openness of the chimney’s foot and the created pressure drop, the ambient air enters the chimney’s foot and the hot and cold streams are combined together. Due to the high temperature of the created mixture, this flow continuously moves upward inside the chimney by buoyancy force.

The reason for creating buoyancy force inside the chimney is due to the higher temperature of the fluid inside the chimney compared to the ambient temperature. Variation in the fluid flow temperature inside the chimney along the radial direction is plotted in Figure 9 for three different heights from the chimney’s foot. As observed in Figure 9, when the fluid moves upward inside the chimney the temperature becomes more uniform. Analyzing the results indicate that the fluid flow leaves the chimney with a completely uniform temperature of 309 K.

Fluid velocity is one of the most important parameters in the chimney. As shown in Figure 10 (contour plot of the velocity inside the chimney) by moving upwards until the top of the chimney, the hot and cold streams are mixed together at different velocities, reaching a uniform velocity and temperature at the exit of the chimney. Figure 11 illustrates the fluid flow velocity along the chimney’s radius at three different heights from the chimney’s foot. As shown in this figure, the maximum velocity at the height of 25 cm from the chimney’s foot is 14.86% and 20.36% higher than its related

**FIGURE 4**  
A, A sample of the grid generation inside the solution domain; B, the effect of the computational cells number on the average velocity at the chimney’s gas

**FIGURE 5**  
A, Comparison of the experimental and CFD modeling results of the ambient air inlet velocity into the chimney at different hot inlet temperatures; B, relative difference between the results
Because at the chimney's inlet the flow is laminar, the fluid velocity in chimney's center increases by the formation of the boundary layer on the chimney wall and its growth. The boundary layer then leaves the laminar mode and becomes turbulent, resulting in a more uniform speed profile.

5.3 Investigating the effect of the hot inlet fluid entering the chimney's foot

To study the effect of hot inlet's temperature on the fluid flow velocity inside the chimney, the modeling has been carried out at three different temperatures of the hot inlet (including 330, 350 and 370 K) at a certain hot inlet velocity of 1.75 m/s and at ambient temperature of 290 K. Figures 11 and 12 displays the effect of the hot inlet temperature on the average temperature and relative pressure of the fluid flow inside the chimney, respectively. As shown in Figure 12, increasing the temperature of the hot inlet the average temperature inside the chimney increases, which results in an increase in buoyancy force and it results a higher pressure drop at the chimney’s foot (Figure 13). As shown in Figure 14, the increase in the pressure drop at the chimney’s foot causes a more volume of the ambient air entering the chimney, resulting in a higher velocity ratio (V.R). Also, as shown in Figure 15, increase in the velocity of the inlet air to the chimney’s foot at a constant hot inlet velocity increases the maximum velocity.
In order to investigate the effect of the volumetric flow rate of the hot inlet entering the chimney's foot, modeling has been performed at three different velocities of the hot inlet with a constant temperature of 350 K for the hot inlet and an ambient temperature of 290 K. The effect of the hot inlet velocity on the temperature, relative pressure, and velocity of the fluid flow along the chimney's height is presented in Figures 16-18, respectively. As shown in Figures 16 and 17, increasing the velocity of the hot inlet at the chimney's foot, the average temperature of the fluid flow along the chimney is also increased, resulting in increasing buoyancy and further pressure drop.
FIGURE 14  Variation in the inlet air velocity and velocity ratio (V.R) in terms of the temperature of the hot inlet entering into the chimney’s foot

FIGURE 15  Variation in the fluid velocity along the center of the chimney at three different hot inlet temperatures

FIGURE 16  Variation in the fluid temperature along the center of the chimney at three different hot inlet velocities

FIGURE 17  Variation in the fluid pressure along the center of the chimney at three different hot inlet velocities

FIGURE 18  Variation in the fluid velocity along the center of the chimney at three different hot inlet velocities

FIGURE 19  Variation in the inlet air velocity and velocity ratio (V.R) in terms of the velocity of the hot inlet entering into the chimney’s foot

from 1.75 to 2.5 m/s causes 17.43% pressure drop at chimney’s foot; but at the height of more than 1 m from the chimney’s foot, the same amount of pressure drop is observed for both of
these velocities, while the fluid temperature inside the chimney is not the same for the two states. From the above results, it can be concluded that the diameter of the chimney has a certain capacity and as the hot inlet velocity increases than a certain amount (eg, 1.75 m/s in Figure 17) the pressure drop at the top sections of the chimney (eg, higher heights than 1 m in Figure 17) will not decrease and the pressure of the fluid in the chimney will not change. Figure 18 shows that as the hot inlet velocity increases, the velocity of the fluid at the center of the chimney and at its output increases (Figure 18); also, the aforementioned results indicate that, for a given value of the hot inlet velocity, the velocity of the fluid in the chimney does not change from about 1 m in height to the next.

Variation in the inlet air velocity and V.R has been plotted in Figure 19 for different amounts of the hot inlet velocities. As shown in Figure 19, with the increase in the hot inlet velocity, the inlet air velocity increases and the velocity ratio decreases, respectively. These results show that decreasing the flow rate of the hot inlet at the chimney’s foot the V.R increases, so that at a hot inlet velocity of 1 m/s, the inlet velocity of the ambient air and the chimney’s output are 1.15 and 1.3 m/s, respectively.

6 | CONCLUSION

At the gas of the fossil fuel power plant’s chimney, the fluid enters the environment with a high temperature, which has a significant level of energy. Due to the limited volume and velocity of the buoyant air inside the chimney, the method proposed in this study can be applied. As demonstrated in the CFD modeling and experimental results, at the entry of the chimney’s foot, the ambient air and the output of the power plant can be combined with a specific ratio to achieve a certain velocity. The pilot-scale results can be extended to a real-size power plant of the same scale and flow rate. The results showed that the air entering the chimney’s foot can be faster than the hot stream flowing entering the chimney’s foot.

For the hot stream entering the chimney’s foot at speed of 1 m/s and temperature of 350 K, the maximum amount of the velocity of the chimney’s gas and the inlet air at the foot reaches 1.3 and 1.15 m/s, respectively; this causes a significant increase in the volume of fluid flow.

Analyzing the results indicates that increasing the temperature of the hot section at a constant flow rate leads to an increase in the velocity of the inlet air into the chimney. Also, increase in the flow rate of the hot section at a constant temperature causes decrease in the V.R. By analyzing the results, the following advantages can be achieved for the proposed system in this study compared to solar chimney systems:

- The system occupies very little space (no need for a collector) compared to the solar chimney. Furthermore, a considerable reduction in the system cost is observed (due to the elimination of the collector’s high cost).
- Unlike the solar chimney systems, it can produce power during the night when the power plant is operational.

NOMENCLATURE:

- \( G_k \): Turbulent kinetic energy generation due to mean velocity gradients
- \( G_b \): Turbulent kinetic energy generation due to buoyancy
- \( k \): Kinetic energy
- \( Y_M \): Fluctuating dilation in compressible turbulence
- \( \varepsilon \): Rate of dissipation of kinetic energy
- \( \mu \): Dynamic fluid viscosity
- \( \mu_t \): Eddy viscosity
- \( \lambda \): Thermal conductivity
- \( T \): Temperature
- \( P \): Pressure
- \( t \): Time or unsteady items
- \( u \): Velocity
- \( C_{1e}, C_{2e}, C_{3e} \): Constants
- \( S_k \) and \( S_\varepsilon \): User-defined constant
- \( \beta \): Volume coefficient of expansion
- \( \rho \): Density
- \( \tau \): Shear stress caused by viscosity
- \( g \): Acceleration of gravity
- \( C_p \): Specific heat at constant pressure
- \( \sigma_k \): Turbulent Prandtl number for \( k \)
- \( \sigma_\varepsilon \): Turbulent Prandtl number for \( \varepsilon \)
- \( Pr_t \): Turbulent Prandtl numbers for \( k \)
- \( a \): Speed of sound
- \( M_t \): Turbulent Mach number

ENDNOTE

\[ \text{Velocity Ratio: V.R} = \frac{\text{velocity of the inlet air}}{\text{velocity of the hot inlet into the chimney's foot}} \]

ORCID

Mohammad Hossein Ahmadi [ORCID ID: https://orcid.org/0000-0002-0097-2534]

REFERENCES

1. Deymi-Dashtebayaz M, Ebrahimi-Moghadam A, Pishbin SI, Pourramezan M. Investigating the effect of hydrogen injection on
natural gas thermo-physical properties with various compositions. *Energy*. 2019;167:235-245.

2. Farzaneh-Gord M, Mohseni-Gharyehsafa B, Ebrahimi-Moghadam A, Jabari-Moghadam A, Toikka A, Zvereva I. Precise calculation of natural gas sound speed using neural networks: an application in flow meter calibration. *Flow Meas Instrum*. 2018;64:90-103.

3. Ebrahimi-Moghadam A, Moghadam AJ. Optimal design of geometrical parameters and flow characteristics for Al2O3-water nanofluid inside corrugated heat exchangers by using entropy generation minimization and genetic algorithm methods. *Appl Therm Eng*. 2019;149:889-898.

4. Dormohammadi R, Farzaneh-Gord M, Ebrahimi-Moghadam A, Ahmadi MH. Heat transfer and entropy generation of the nanofluid flow inside sinusoidal wavy channels. *J Mol Liq*. 2018;269:229-240.

5. Mohseni-Gharyehsafa B, Ebrahimi-Moghadam A, Okati V, Farzaneh-Gord M, Ahmadi MH, Lorenzini G. Optimizing flow properties of the different nanofluids inside a circular tube by using entropy generation minimization approach. *J Therm Anal Calorim*. 2018;135:801-811.

6. Ebrahimi-Moghadam A, Mohseni-Gharyehsafa B, Farzaneh-Gord M. Using artificial neural network and quadratic algorithm for minimizing entropy generation of Al2O3-EG/W nanofluid flow inside parabolic trough solar collector. *Renew Energy*. 2018;129:473-485.

7. Sadaghiani MS, Ahmadi MH, Mehrpooya M, Pourfayaz F, Feidt M. Process development and thermodynamic analysis of a novel power generation plant driven by geothermal energy with liquefied natural gas as its heat sink. *Appl Therm Eng*. 2018;133:645-658.

8. Kasaeian AB, Amirifard M, Ahmadi MH, Kasaeian F. Investigation of the effects of ambient temperature and dimensional parameters on the performance of solar chimney power plants. *Int J Low-Carbon Technol*. 2017;12:335-348.

9. Ahmadi MH, Jokar MA, Ming T, Feidt M, Pourfayaz F, Astaraei FR. Multi-objective performance optimization of irreversible molten carbonate fuel cell–Braysson heat engine and thermodynamic analysis with ecological objective approach. *Energy*. 2018;144:707-722.

10. Ahmadi MH, Ahmadi MA, Pourfayaz F. Thermodynamic analysis and evolutionary algorithm based on multi-objective optimization performance of actual power generating thermal cycles. *Appl Therm Eng*. 2016;99-1005.

11. Sadatsakkak SA, Ahmadi MH, Bayat R, Pourkiaei SM, Feidt M. Optimization density power and thermal efficiency of an endoreversible Braysson cycle by using non-dominated sorting genetic algorithm. *Energy Convers Manag*. 2015;93:31-39.

12. Ahmadi MH, Sayyaadi H, Dehghani S, Hosseinzade H. Designing a solar powered Stirling heat engine based on multiple criteria: maximized thermal efficiency and power. *Energy Convers Manag*. 2013;75:282-291.

13. Farzaneh-Gord M, Pahlevan-Zadeh MS, Ebrahimi-Moghadam A, Rastgar S. Measurement of methane emission into environment during natural gas purging process. *Environ Pollut*. 2018;242:2014-2026.

14. Ebrahimi-Moghadam A, Farzaneh-Gord M, Arabkoohsar A, Moghadam AJ. CFD analysis of natural gas emission from damaged pipelines: correlation development for leakage estimation. *J Clean Prod*. 2018;199:257-271.

15. Ebrahimi-Moghadam A, Farzaneh-Gord M, Deymi-Dashtebayaz M. Correlations for estimating natural gas leakage from above-ground and buried urban distribution pipelines. *J Nat Gas Sci Eng*. 2016;34:185-196.

16. Noroozian A, Mohammadi A, Bidi M, Ahmadi MH. Energy, energy and economic analyses of a novel system to recover waste heat and water in steam power plants. *Energy Convers Manag*. 2017;144:351-360.

17. Mohammadi A, Bidi M, Ahmadi MH. Economic evaluation of different scenarios for gas turbine waste heat recovery to produce water and power. *Int J Ambient Energy*. 2017;38:727-734.

18. Ahmadi MH, Mehrpooya M, Abbasi S, Pourfayaz F, Bruno JC. Thermo-economic analysis and multi-objective optimization of a transcritical CO2 power cycle driven by solar energy and LNG cold recovery. *Therm Sci Eng Prog*. 2017;4:185-196.

19. Günther H. *In Hundred Years-Future Energy Supply of the World*. Stuttgart, Germany: Stuttgart Kosmos Fr Verlagshandlung; 1931.

20. Zhou X, Wang F, Fan J, Ochieng RM. Performance of solar chimney power plant in Qinghai-Tibet Plateau. *Renew Sustain Energy Rev*. 2010;14:2249-2255.

21. Haaf W. Solar chimneys. *Int J Sol Energy*. 1984;2:141-161.

22. Haaf W, Friedrich K, Mayr G, Schlaich J. Solar chimneys part I: principle and construction of the pilot plant in Manzarnas. *Int J Sol Energy*. 1983;2:3-20.

23. dos S, Bernardes MA, Voß A, Weinrebe G. Thermal and technical analyses of solar chimneys. *Sol Energy*. 2003;75:511-524.

24. Ghorbani B, Ghashami M, Ashjaee M, Hosseinizadeh G. Electricity production with low grade heat in thermal power plants by design improvement of a hybrid dry cooling tower and a solar chimney concept. *Energy Convers Manag*. 2015;94:1-11.

25. Fluri TP, Pretorius JP, Van Dyk C, Von Backström TW, Kröger DG, Van Zijl GPAG. Cost analysis of solar chimney power plants. *Sol Energy*. 2009;83:246-256.

26. Okoye CO, Atikul U. A parametric study on the feasibility of solar chimney power plants in North Cyprus conditions. *Energy Convers Manag*. 2014;80:178-187.

27. Guo P, Li J, Wang Y, Wang Y. Evaluation of the optimal turbine pressure drop ratio for a solar chimney power plant. *Energy Convers Manag*. 2016;108:14-22.

28. Li J, Guo H, Cheng Q, Huang S. Optimal turbine pressure drop for solar chimney-aided dry cooling system in coal-fired power plants. *Energy Convers Manag*. 2017;133:87-96.

29. Zhou X, Yang J, Wang J, Xiao B. Novel concept for producing energy integrating a solar collector with a man made mountain hollow. *Energy Convers Manag*. 2009;50:847-854.

30. Zhou X, Yang J, Xiao B, Hou G. Simulation of a pilot solar chimney thermal power generating equipment. *Renew Energy*. 2007;32:1637-1644.

31. Patel SK, Prasad D, Ahmed MR. Computational studies on the effect of geometric parameters on the performance of a solar chimney power plant. *Energy Convers Manag*. 2014;77:424-431.

32. Koonsrisuk A, Lorente S, Bejan A. Constructal solar chimney configuration. *Int J Heat Mass Transf*. 2010;53:327-333.

33. Hamdan MO. Analysis of solar chimney power plant utilizing chimney discrete model. *Renew Energy*. 2013;56:50-54.

34. Joneydi Shariatzadeh O, Refahi AH, Abolhassani SS, Rahmani M. Modeling and optimization of a novel solar chimney cogeneration power plant combined with solid oxide electrolysis/fuel cell. *Energy Convers Manag*. 2015;105:423-432.

35. Fasel HF, Meng F, Shams E, Gross A. CFD analysis for solar chimney power plants. *Sol Energy*. 2013;98:12-22.
36. Khelifi C, Ferroudji F, Ouali M. Analytical modeling and optimization of a solar chimney power plant. *Int J Eng Res Africa*. 2016;25:78-88.

37. Sakonidou EP, Karapantsios TD, Balouktsis AI, Chassapis D. Modeling of the optimum tilt of a solar chimney for maximum air flow. *Sol Energy*. 2008;82:80-94.

38. Mehrpooya M, Shahsavan M, Sharifzadeh MMM. Modeling, energy and exergy analysis of solar chimney power plant-Tehran climate data case study. *Energy*. 2016;115:257-273.

39. Kasaeian A, Ghalamchi M, Ghalamchi M. Simulation and optimization of geometric parameters of a solar chimney in Tehran. *Energy Convers Manag*. 2014;83:28-34.

40. Kasaeian A, Mahmoudi AR, Astaraei FR, Hejab A. 3D simulation of solar chimney power plant considering turbine blades. *Energy Convers Manag*. 2017;147:55-65.

41. Ghalamchi M, Kasaeian A, Ghalamchi M, Fadaei N, Daneshazarian R. Optimizing of solar chimney performance using electrohydrodynamic system based on array geometry. *Energy Convers Manag*. 2017;135:261-269.

42. Ming T, Wu Y, de Richter RK, Liu W, Sherif SA. Solar updraft power plant system: a brief review and a case study on a new system with radial partition walls in its collector. *Renew Sustain Energy Rev*. 2017;69:472-487.

43. de_Richter R, Ming T, Davies P, Liu W, Caillol S. Removal of non-CO2 greenhouse gases by large-scale atmospheric solar photocatalysis. *Prog Energy Combust Sci*. 2017;60:68-96.

44. Ming T, Gui J, de Richter RK, Pan Y, Xu G. Numerical analysis on the solar updraft power plant system with a blockage. *Sol Energy*. 2013;98:58-69.

45. Versteeg HK, Malalasekera W. *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, 1st edn. Harlow: Longman Scientific & Technical; 1995.

46. Ebrahimi-Moghadam A, Farzaneh GM, Deimi D, Bayaz M. Develop an equation to calculate the amount of gas leakage from buried distribution gas pipelines. *Iran J Mech Eng*. 2016;18:64-86.

---

**How to cite this article**: Mokhtari Shahdost B, Razi Astaraei F, Ebrahimi-Moghadam A, Ahmadi MH. Experimental and numerical investigations of a novel chimney system for power generation using the combination of fossil fuel power plant exhaust gases and ambient air. *Energy Sci Eng*. 2019;7:764-776. [https://doi.org/10.1002/ese3.306](https://doi.org/10.1002/ese3.306)