Prediction of flow field in a solar chimney using ANFIS technique

Minh-Thu T Huynh*, Tri Q Truong, Thinh N Doan, Trieu N Huynh, Tung V Nguyen, Viet T Nguyen, Y Q Nguyen
Faculty of Engineering, Van Lang University, 45 Nguyen Khac Nhu St., Co Giang Ward, Dist. 1, Ho Chi Minh City, Viet Nam

*thu.htm@vlu.edu.vn

Abstract. Solar chimneys have been intensively studied as an effective method for natural ventilation of buildings. Though numerical methods, such as Computational Fluid Dynamics (CFD), have been widely utilized in such studies, they usually require extensive computational resources. Moreover, experimental study is quite complicated and costly. In recent years, machine learning has started to be used as a tool in the thermal-fluid field. In this study, in order to save time and cost, Adaptive Neuro-Fuzzy Inference System (ANFIS) technique, a class of adaptive networks that incorporate both neural networks and fuzzy logic principles, is combined with CFD. A simulation model was first validated by experiment from another study in the field. The result was documented as a dataset using CFD code ANSYS Fluent (Academic version 2020 R2). Then, they are used to train and validate the ANFIS model. In particular, the study is to predict the fluid flow field in a 2-dimensional typical solar chimney when heat flux changes in the range of 400 to 1000 W/m². Inputs of the ANFIS model are position and heat flux, while outputs are temperature and velocity at that location. As a result, the 2 ANFIS models could achieve $R^2$ values of 0.997, 0.97 (training set) and 0.994, 0.9715 (testing set); RMSE are 1.009, 0.00224 (training set) and 1.074, 0.0204 (testing set) for outputs of temperature and velocity, respectively. Those results are acceptable. By using the ANFIS model, large amounts of flow fields with different scenarios can be estimated simultaneously. Therefore, it is expected that engineers and architects can have a quick tool in the process of design.

Keywords: solar chimney, CFD, machine learning, ANFIS

1. Introduction
Solar chimney is one of the most interesting passive methods in ventilation for buildings. Chimney effect due to higher temperature air heated by solar energy makes air flow out of the channel; hence, induces natural ventilation. Since the first study on solar chimneys carried out in 1993 by Bansal et al. [1], ventilation potential is proven to increase in a well-designed solar chimney. The study of Zavala-Guillen et al. [2] indicated that when the absorber-partitioned air channel solar chimney (SC-AP) is attached to a building, it is able to generate the air changes per hour (ACH) complying with the requirements recommended by ASHRAE. Studying glazed solar chimney walls combined with other solar chimney wall and roof configurations, Chantawong et al. [3] proved that high air ventilation rates can be induced, allow the substitution of stagnant room air with fresh outside air for a healthy and comfortable interior environment, and maintain indoor temperature at comfortable level. Comparing between a room in
Northern China during summer, with and without Trombe wall, a kind of solar chimney, the temperature in the room with a Trombe wall decreased by 0.5–3 °C and wind velocity increased to approximately 0.2–0.5 m/s [4]. Charvat et al. [5] also showed that with the increase of thermal mass, air velocity increases during the night; while during day the air velocity is increased by 25%. In addition, energy-saving is another advantage of solar chimney [6-7]. Up to 50% of the annual ventilation energy required can be saved for a Tokyo’s office building [8]. Because of the above benefits, a variety of numerical and experimental studies in solar chimney have been done so far [9-12]. However, those studies are for limited scenarios due to either limited resources in computation or expensive experimentation.

Recently, Machine Learning (ML) techniques have attracted intensive research interest in Fluid Dynamics problems. By using limited data obtained from the CFD or experiment as the input, the ML model is trained to learn the dynamical system’s characteristics. Different computational intelligence approaches, i.e. artificial neural network (ANN), genetic programming (GP), and adaptive neuro-fuzzy inference system (ANFIS) have been evaluated and compared for predicting the energetic performance of a building integrated photovoltaic thermal system [13]. Likewise, ANN and ANFIS model are used to predict performance of solar chimney power plants [14]. In the study of Tian et al. [15], the permeability is predicted based on pore structure parameters of porous media using ANN with genetic algorithm as an optimization technique. Pourtousi [16] and his team have combined CFD and ANFIS for the bubble column hydrodynamics study. Likely, Beigzadeh [17] used CFD data for Adaptive Neuro-Fuzzy to study the heat transfer in flat and discontinuous fins.

In this research, we employed an ANFIS model to predict the flow and temperature fields in a solar chimney, which have not been studied in the literature. A CFD model was built, validated, and run to provide data for training the ANFIS model, which was then used to predict the flow and temperature fields in other cases. Thus, engineer or architect can use as a quick tool to optimize their design without complicated computation or experiment. Then, comparisons between the ANFIS and CFD results were then conducted and validated the ANFIS model.

2. Modeling setup

2.1. Modeling in ANSYS

Solar chimney model in this study is sketched in Figure 1. The major dimensions of the solar chimney are the gap (G) and the height (H), which respectively are 0.1 m and 1.025 m. The left wall is heated with uniform heat flux I. The right wall receives radiative heat transfer from the left one. Atmospheric pressure is set for air flow at inlet (lower end of the air channel) and outlet (the upper end).

RANS (Reynolds Averaged Navier – Stokes equations) method in CFD is used to estimate the governing equations of air flow and heat transfer. It consists of continuity, momentum, and energy equations. The computational domain and mesh are coincided the air channel as in figure 1. The mesh was structured and finer as it is closer the walls. For all the cases, the Mesh – refinement tests lead the numerical results changed less than 1.0% when the non–dimensional distance y+ of the first grid points near the solid surface was less than 1.5. Finite Volume Method are also deployed by the CFD code ANSYS Fluent (Academic version 2020 R2), with following assumptions and setups:

- steady and two-dimensional flow and heat transfer; incompressible air flow in the channel.
- radiative heat transfer from the heated wall to the opposite wall in the air channel described by the S2S model;
- using RNG k – ε turbulence model;
- using SIMPLE method for the coupling between the continuity equation and the momentum equation;
- using PRESTO! Method for interpolation of the pressure on the staggered mesh, second order upwind scheme for momentum equation and energy conservation equation, and first order upwind scheme for the equation for k and ε.
Since mass flow rate is the most important parameter to evaluate performance of solar chimney, it is used to validate the solar chimney simulation model. Figure 2 shows that mass flow rate’s discrepancy for simulation and experiment in the research of Burek and Habeb [9] is less than 10%, which is acceptable due to other minor factors during experimentation, such as heat loss through walls.

![Diagram](image)

**Figure 1.** Modelling and meshing in ANSYS.

![Graph](image)

**Figure 2.** Computed (CFD) and measured (Expt.) flow rate in the experiment by Burek and Habeb [9] at different gaps and heat fluxes.

2.2. **ANFIS technique**

The general ANFIS structure [19], assuming to comprise two inputs \( x, y \) and 5 layers, is illustrated in Figure 3. Note that, the output is marked as the fifth layer. Based on the Sugeno fuzzy model [20], the model is defined by 2 rules as follows:

- Rule 1: If \( x \) is equal \( A_1 \) and \( y \) is equal \( B_1 \) then \( f_1 = p_1 x + q_1 y + r_1 \)
- Rule 2: If \( x \) is equal \( A_2 \) and \( y \) is equal \( B_2 \) then \( f_2 = p_2 x + q_2 y + r_2 \)

where \( p, q \) and \( r \) are ANFIS prediction parameters of each rules.
At the Layer 1, we define 4 membership functions that can be transformed the two inputs into linguistic labels. The membership function in the current study is selected as Bell-shaped functions:

\[ \mu_i = \frac{1}{1 + \left( \frac{|x - c_i|}{a_i} \right)^{2b_i}} \]

where \( a_i, b_i \) and \( c_i \) with \( i = (1, 2) \) are the fuzzy set parameters.

The Layer 2 performs the combination of two membership functions expressed by two inputs. Subsequently, the Layer 3 is used to calculate the ratio of the \( i \)-th rule’s firing strength to the sum of all rules’ firing strengths. The ratio is labeled as \( \bar{w}_i \) with \( i = (1, 2) \). The Layer 4 gives the weighted output combining the previous layer and first-order based on Sugeno fuzzy model as \( \bar{w}_i f_i \) with \( i = (1, 2) \). The Layer 5 contains one node presenting the output of the network as the sum of all weighted output in Layer 4.

To evaluate the accuracy of produced outputs, two statistical indices of \( R^2 \) and RMSE are used.

![Diagram](image)

**Figure 3.** The general ANFIS structure.

### 3. Results and discussion

#### 3.1. ANFIS model training

Dataset is generated using ANSYS with the above validated CFD model to include: location, \( x \) (m), \( y \) (m); heat flux, \( I \) (W/m\(^2\)); temperature \( T \) (K) and velocity \( V \) (m/s). The velocity is assumed as velocity on \( y \)-direction since the velocity on \( x \)-direction is very small; hence negligible.

As in all machine learning methods, training a successful network requires a lot of training data. More than 130 values of \( I \) in the range of 400 – 1000 W/m\(^2\) have been simulated. In each case, the total number of mesh nodes is 20,300 (100 x 200 mesh elements).

The inputs of the ANFIS are \( x, y \) and \( I \), while 2 outputs are \( T \) and \( V \). The 4 G-bell functions are used for membership functions of all 3 inputs. From dataset of more than 2.8 million points created from ANSYS, we randomly chose a part of them to train the ANFIS model. By increasing the number of data points used for training, we found that a training set of 100,000 data points offered the best \( R^2 \) and RMSE.

The fuzzy set parameters of each membership function are in Table 1.

The training results, respectively for temperature \( T \) and velocity \( V \), give RMSE of 1.009 and 0.00224; and \( R^2 \) of 0.99666 and 0.97. Those results are quite reasonable for the outputs in this study.
### Table 1. ANFIS model inputs

| Input       | MF    | Type of MF | a    | b    | c    |
|-------------|-------|------------|------|------|------|
| $x$ (m)     | in1mf1| Gbellmf    | 0.007| 1.200| -0.015|
|             | in1mf2| Gbellmf    | 0.029| 2.000| 0.027 |
|             | in1mf3| Gbellmf    | 0.028| 2.000| 0.072 |
|             | in1mf4| Gbellmf    | 0.010| 2.000| 0.115 |
| $y$ (m)     | inf2mf1| Gbellmf | 0.171| 2.000| 0.000 |
|             | inf2mf2| Gbellmf | 0.171| 2.000| 0.341 |
|             | inf2mf3| Gbellmf | 0.171| 2.000| 0.683 |
|             | inf2mf4| Gbellmf | 0.171| 2.000| 1.025 |
| $I$ (W/m$^2$) | inf3mf1| Gbellmf | 61.530| 2.000| 421.90 |
|             | inf3mf2| Gbellmf | 61.532| 2.000| 545.003|
|             | inf3mf3| Gbellmf | 61.532| 2.000| 668.067|
|             | inf3mf4| Gbellmf | 61.532| 2.000| 791.130|

### 3.2. ANFIS model validation

In order to validate the above ANFIS model, some values of heat fluxes in the remaining dataset after training are used for test. In this section, the results for three heat fluxes of 429, 532 and 615 W/m$^2$, which were selected randomly in the tested range, are presented. Table 2 shows RMSE and $R^2$ of $T$ and $V$ for each heat flux. It can be seen that the result for all 3 cases are acceptable.

### Table 2. RMSE and $R^2$ for testing data.

| $I$ (W/m$^2$) | Outputs | RMSE  | $R^2$ |
|---------------|---------|-------|-------|
| 429           | $T$ (K) | 1.00697| 0.99454 |
|               | $V$ (m/s)| 0.01703| 0.97730 |
| 532           | $T$ (K) | 1.11894| 0.99411 |
|               | $V$ (m/s)| 0.02014| 0.97225 |
| 615           | $T$ (K) | 1.09676| 0.99334 |
|               | $V$ (m/s)| 0.02411| 0.96372 |

Figure 4 shows comparisons of the temperature and velocity distributions in the channel obtained from CFD and ANFIS models for the cases presented in table 2. It is seen that the ANFIS model was able to capture well the main flow and temperature fields in the channel, particularly near the wall, at all heat fluxes. However, there are slight deviations among the two results near the channel center. The ANFIS model slightly over-predicted the temperature fields and under-predicted the velocity distributions. Since the channel center had the lowest temperature and velocity, the differences between the two results are seen in accord with the RMSE values in table 2.
Figure 4. CFD vs. ANFIS model for temperature and velocity distributions with different heat fluxes (vertical and horizontal directions are not at the same scale).
4. Conclusion

This study attempted to introduce a test on combination of Adaptive Neuro-Fuzzy Inference System (ANFIS) model for evaluating the flow field in a solar chimney. By comparing flow fields with several testing heat fluxes between CFD and ANFIS model, it can be confirmed that the ANFIS model is validated with acceptable RMSE and $R^2$. This ANFIS model was able to predict flow field at any point in the tested solar chimney channel for any heat flux in the range of 400 – 1000 W/m². Then, the result can provide a fast tool for engineers or architects in their design process.

In future works, the present model can be extended to either different dimensions of the solar chimney or other types of solar chimney, such as inclined or combine vertical – inclined solar chimneys. In addition, computational time should also be compared between CFD and ANFIS models.

References

[1] Bansal N K, Mathur R and Bhandari M S 1994 A study of solar chimney assisted wind tower system for natural ventilation in buildings J. Building Environ. 3 373–377
[2] Zavala-Guillén I, Xamán J, Hernández-Pérez I, Hernández-Lopéz I, Jiménez-Xamán C, Moreno-Bernal P and Sauceda D 2018 Ventilation potential of an absorber-partitioned air channel solar chimney for diurnal use under Mexican climate conditions J. Appl. Therm. Eng. 149 807-821
[3] Chantawong P, Hirunlabh J, Zeghmati B, Khedari J, Teekasap S and Win M M 2006 Investigation on thermal performance of glazed solar chimney walls Solar Energy 89 (3) 288-297
[4] Zhou Y, Wang Z, Yang C, Xu L and Chen W 2019 Influence of Trombe wall on indoor thermal environment of a two-story building in rural Northern China during summer Science and Technology for the Built Environment 25 (4) 438–449
[5] Charvat P, Jicah M and Stetina J 2004 Solar chimneys for ventilation and passive cooling Denver (USA): World Renewable Energy Congress.
[6] Al Touma A and Ouahran D 2018 Performance assessment of evaporatively-cooled window driven by solar chimney in hot and humid climates Solar Energy 169 187–195
[7] Briga Sá A C, Martins A, Boaventura-Cunha J, Carlos Lanzinha J and Paiva A 2018 An analytical approach to assess the influence of the massive wall material, thickness and ventilation system on the Trombe wall thermal performance Journal of Building Physics 41 (5) 445–468
[8] Miyazaki T, Akisawa A and Kashiwagi T 2016 The effects of solar chimneys on thermal load mitigation of office buildings under the Japanese climate, Renew. Energy 31 (7) 987–1010
[9] Burek S A M and Habeeb A 2007 Air flow and thermal efficiency characteristics in solar chimneys and Trombe Walls J. Energy Build. 39 128–135
[10] Chen Z D, Bandopadhyay P, Halldorsson J, Byrjalsen C, Heiselberg P and Li Y 2003 An experimental investigation of a solar chimney model with uniform wall heat flux J. Building Environ. 38 (7) 893–906
[11] Khanal R and Lei C 2015 A numerical investigation of buoyancy induced turbulent air flow in an inclined passive wall solar chimney for natural ventilation J. Energy Build. 93 217–226
[12] Shi L, Ziem A, Zhang G, Li J and Setunge S 2020 Solar chimney for a real building considering both energy-saving and fire safety – a case study J. Energy Build. 221 110016
[13] Gao W, Moayedi H and Shahsavaran A 2019 The feasibility of genetic programming and ANFIS in prediction energetic performance of a building integrated photovoltaic thermal (BIPVT) system Solar Energy 183 293-305
[14] Amirkhani S, Nasirivatan S, Kasaeeian A B and Hajinezhad A 2015 ANN and ANFIS models to predict the performance of solar chimney power plants Renew. Energy 83 597-607
[15] Tian J, Qi C, Sun Y, Yaseen Z M and Pham B T 2020 Permeability prediction of porous media using a combination of computational fluid dynamics and hybrid machine learning methods Engineering with Computers
[16] Pourtousi M, Sahu J N, Ganesan P, Shamshirband S and Redzwan G 2015 A combination of
computational fluid dynamics (CFD) and adaptive neuro-fuzzy system (ANFIS) for prediction of the bubble column hydrodynamics Powder Technology 274 466–481

[17] Beigzadeh R 2019 The CFD Providing Data for Adaptive Neuro-Fuzzy to Model the Heat Transfer in Flat and Discontinuous Fins Iranian J. Chem. Engineering 16 (2) 57-69

[18] Enayatollahi H, Fussey P and Nguyen B K 2020 Modelling Evaporator in Organic Rankine Cycle using Hybrid GD-LSE ANFIS and PSO ANFIS Techniques Thermal Science and Engineering Progress 19 100570

[19] Jang J-S R 1993 ANFIS: adaptive-network-based fuzzy inference system IEEE Transactions on Systems, Man, and Cybernetics 23 (3) 665-685

[20] Takagi T and Sugeno M 1985 Fuzzy Identification of Systems and Its Applications to Modeling and Control IEEE Transactions on Systems, Man, and Cybernetics SMC-15 (1) 116-132.