CFD Analysis of Airflow Through Prism Obstacles Inside Solar Air Heater Channel

L Diana¹, A G Safitra¹, D Ichsani², S Nugroho¹

¹ Mechanics and Energy Department, Politeknik Elektronika Negeri Surabaya, Jalan Raya ITS Sukolilo 60111, Indonesia
² Mechanical Engineering Department, Institut Teknologi Sepuluh Nopember, Jalan Teknik Kimia ITS Sukolilo 60111, Indonesia

Abstract. This study presents an analysis of airflow through prism obstacles inside a trapezoidal solar air heater channel using a computational fluid dynamics method. The purpose is to now the best configuration of gap ratio (S/H) 0.7, 1, 1.3 for 30° obstacle folded angle. The air is 2.62 m/s and the air temperature is 299 K. The channel wall boundary condition is heat flux with a constant value of 620 W/m² is attached alongside the wall of the channel. The equation is Pressure-velocity coupling performed with SIMPLE. Renormalization-group (RNG) k-ε is chosen as a turbulent model. The simulation result presents temperature distribution, airflow vector in the middle plane of a channel. The obstacle can create backflow. It can increase the air temperature in a channel.

1. Introduction

Solar air heater has many purposes such as to drying fruits, meat, seeds, and vegetables [1][2]. The working principle of solar air heater is solar radiation heat transmitted by a glass than absorbed by an absorber plate. The absorber plate will heat the air inside a channel. The hot air is used to drying process. Some researches about solar air heater have been done to increase performance. Some research change shape of absorber plate to increase heated area. The performance of a solar air heater can be increased with add effective area of absorber plate. A v-corrugated absorber has more effective area than flat-plate absorber [3]. In other research, a trapezoidal air heater channel can absorb much heat and give better efficiency if it compared to the other absorber shapes that had been researched previously [4]. The other way to increase the performance of solar air heater is adding obstacles. It can raise the heat transfer inside an absorber channel, the experiment was performed by inserting aluminum cans inline and staggered in order to improve solar air heater performance [5]. The highest efficiency is obtained when the cans were arranged staggered with a mass flow rate of 0.05 kg/s. Research on v-corrugated absorber with an angle of 20° and folded triangular obstacle were also added inside the channel. The research reported that folded angle’s obstacle with 30° has the most optimum efficiency compared to obstacle with folded angle’s obstacle of 0° to 80° [6].

Other research method about solar air heater not only an experiment but also a simulation. A simulation is a method to present some phenomena that difficult to catch eyes in actual experiment conditions such as temperature and pressure contour, velocity vector, etc. A simulation study using computational fluid dynamics software is done to know performance analysis of solar air heater duct provide with artificial roughness [7]. Some computational fluid dynamics study such as solar air heater with v-plate on channel surface and solar air heater with transverse wire ribs in upper channel surface chose Renormalization-group (RNG) k-ε as turbulent model, pressure-velocity coupling, performed with SIMPLE because the result for Nusselt number has small gap with experiment result and Dittus-
Boelter equation [8][9]. The simulation about trapezoidal solar air heater is investigated to know airflow and air temperature distribution [10]. The advantages of simulation: the cost is cheaper than an experiment because it can simulate a lot of variation. The running time is short because some variations can be done at the same time. The simulation design can be a reference for the experiment model. There are research and standard that can use for validation. This study will analysis airflow through prism obstacles inside a trapezoidal solar air heater channel using a computational fluid dynamics method. This study uses 30° for obstacle folded angle. The gap ratio (S/H) is used as dimensionless parameter. The variation of S/H are 0.7, 1, 1.3. This study has purpose to know best variation of S/H. The simulation result will present analysis of temperature distribution, airflow vector.

2. Computational Fluid Dynamics Method

2.1 Geometry
The geometry of trapezoidal channel is 900 mm long, 30 mm wide and 85 mm high. The lower angle of the trapezoidal channel is 89°. The channel is made of aluminum plate. The obstacles in the form of triangular prism are arranged in one line inside the channel. The obstacles are 18 mm wide and 51 mm high with the folded angles 30° as in Figure 1. The variation of the gap ratio (S/H) 0.7, 1, 1.3. The H is the height of the obstacles and S is distance of obstacle.

2.2 Boundary condition
The boundary condition for simulation can be seen in figure x. The air inlet velocity is 2.62 m/s. The air inlet temperature is 299 K. The channel wall boundary condition is heat flux with constant value of 620 W/m² is attached alongside the wall of the channel. The boundary condition can be seen in Figure 1.

![Figure 1. Boundary condition and gap ratio](image)

2.3 Meshing
The mesh uses hex core-native mesh type with inflation near to obstacle wall. The mesh can be seen in Figure 2.
2.4 Parameter Set up

This study uses 3D-model, and double precision. Renormalization-group k-ε model for turbulent viscous model. A second-order upwind scheme is chosen for energy and momentum equations. The convergence criterion of $10^{-6}$ is used for the residual of the continuity, velocity components and residual of the energy. The detail of pro-processing set up can be seen in Table 1.

| Table 1. Pro-processing set up |
|--------------------------------|
| **Model solver** | Solver | Pressure based |
| | Formulation | Implicit |
| | Space | 3D |
| **Energy equation** | **Type** | **Value** |
| **Model viscous** | Inlet = Velocity inlet | V= 2.62 m/s (Re =4000) |
| | T in = 299 K | |
| | Outlet = Outflow | - |
| | Wall | Heat flux = 620 W/m² |
| **Boundary condition** | **Material** | **Value** |
| | Density | 1.16607 kg/m³ |
| | Cp | 1006.98 J/kg.K |
| | K | 0.02622 W/mK |
| | Viscosity | 1.842 x 10⁻⁵ kg/ms |
| **Pressure-velocity coupling** | SIMPLE | |
| **Discretization** | Pressure | Second order |
| | Momentum | Second order |
| | Turbulent kinetic energy | Second order |
| | Turbulent dissipation rate | Second order |

3. Result and Discussion

Grid independence is validation step. It is needed to produce accurate simulation result. This study compares five mesh A, B, C, D and E with different number of cells, faces and nodes for S/H 0.7. To decide which mesh that can use for simulation, there are some parameter. They are $Y^+$, difference temperature $\Delta T$, drop of pressure $\Delta P$ and useful energy $Qu$. Grid independence result for obstacle with a folded angle of 40° shown in Table 2. Base on it, mesh D is chosen as reference of number cells that used for simulation. It is because mesh D result has small difference value with mesh E for all parameter and also consider computer capacity and running time. It is also used for S/H 1 and 1.3.

| Table 2. Grid independence |
|---------------------------|
| **Mesh** | **Cells** | **Faces** | **Nodes** | **$Y^+$** | **$\Delta T$ (K)** | **$\Delta P$** | **Qu** |
|---------------------------|----------------|-----------|----------|---------|-----------------|-----------|-----|

3
Figure 3 shows the air temperature distribution in the middle of a channel. The temperature distribution is not uniform. The channel with S/H 0.7 has the lowest air temperature in the outlet section. It is showed in a dark blue color. The channel with S/H 1.3 has the highest air temperature in the outlet section. It is showed in a light blue color.

|    | A     | B     | C     | D     | E     |
|----|-------|-------|-------|-------|-------|
|    | 401241| 797885| 1259435| 1524571| 1987955|
|    | 858536| 1825441| 2823965| 3756452| 4960493|
|    | 103710| 354277 | 435141 | 557689 | 679428 |
|    | 18.04 | 17.28 | 14.27 | 11.32 | 10.08 |
| Pa |       |       |       |       |       |
|    | 14.37 | 15.08 | 15.73 | 16.42 | 16.93 |
|    | 3.94  | 4.18  | 5.61  | 6.36  | 6.62  |
| Watt|      |       |       |       |       |
|    | 112.54| 114.96| 118.66| 122.36| 124.01|

Figure 3. The air temperature distribution in middle of channel.

Figure 4 shows the air velocity distribution in the middle of the channel. Base on the figure the channel with S/H 0.7 the backflow happens near the inlet position or in front of the second obstacle. It because the obstacle arrangement is nearest than other S/H variation. The backflow is showed blue color.

|    | A     | B     | C     | D     | E     |
|----|-------|-------|-------|-------|-------|
|    | 401241| 797885| 1259435| 1524571| 1987955|
|    | 858536| 1825441| 2823965| 3756452| 4960493|
|    | 103710| 354277 | 435141 | 557689 | 679428 |
|    | 18.04 | 17.28 | 14.27 | 11.32 | 10.08 |
| Pa |       |       |       |       |       |
|    | 14.37 | 15.08 | 15.73 | 16.42 | 16.93 |
|    | 3.94  | 4.18  | 5.61  | 6.36  | 6.62  |
| Watt|      |       |       |       |       |
|    | 112.54| 114.96| 118.66| 122.36| 124.01|
Figure 4. The air velocity distribution in middle of channel.

Figure 5 shows the air velocity vector in the middle of the channel. The air velocity vector is needed to show airflow direction in the channel. In the channels with obstacles, a velocity vector of airflow does not go to the outlet smoothly. It is because of backflow. The backflow happened because airflow crash into obstacles walls.
The backflow can increase the air temperature in the channel. It is because air goes back to heat other air in before position. It is showed in Figure 6. In this condition, the flow will be turbulent. The turbulent flow can increase heat transfer in a channel. Base in Figure 6, the air temperature increases along length Z. The gap ratio S/H 1.7 has the highest air temperature 319 K and the lowest air temperature when S/H 0.7 the air temperature 316 K.

Figure 5. The air velocity vector in the middle of channel

Figure 6. The air temperature in length Z

4. Conclusion
The obstacle in a solar air heater channel can increase temperature and heat transfer. The prism obstacles can increase temperature because the function of prism obstacles is to direct airflow to touch
upper and side absorber plate so air will be heated. The obstacle can create backflow. The backflow can increase the air temperature in the channel. It is because air goes back to heat other air in before position.

5. References

[1] Ivanova D and Andonov K 2001 *Analytical and Experimental Study of Combined Fruit and Vegetable Dryer* Energi conversion and management 42, pp 975-983

[2] Abene A Dubois, V Le Ray M and Oagued A 2004 *Study of A Solar Air Flat Plate Collector: Use of Obstacle and Application for The Drying of Grape*, Journal of food engineering, Vol. 65, pp 15-22

[3] Karim and M A Hawlader, M N A 2006 Performance Investigation of flat Plate, V-Corrugated and finned Air Collector, Energy 31, pp.452–470

[4] Lakshmi, D V N Apurba L & P Muthu K 2016 Performance Analysis of Trapezoidal Corrugated Solar Air Heater with Sensible Heat Storage Material

[5] Ozgen F and Esen, H 2009 Experimental Investigation of Thermal Performance of Double-Flow Solar Air Heater Having Aluminium Cans, Renewable Energy, vol. 30, pp.2594-2601

[6] Handoyo Ekadewi 2014 Experimental Studies on A Solar Air Heater Having V-Corrugated Absorber Plate with Obstacles Bent Vertically. Applied Mechanics and Materials, Vol. 493, pp 86-9

[7] Kumar Sharad and Saini Rp 2009 CFD Based Performance Analysis of A Solar Air Heater Duct Provided with Artificial Roughness, Renewable Energy, 34

[8] Yadav, A. S and Bhagoria, J.L 2013 A CFD (Computational fluid Dynamics) Based Heat Transfer and Fluid Flow Analysis of A Solar Air Heater Provided with Circular Transverse Wire Rib Roughness on The Absorber Plate, Energy 55, pp. 1127-1142

[9] Jin Dongxu and Zhang Manman Numerical Investigation of Heat Transfer and Fluid Flow in A Solar Air Heater Duct with V-Shaped Ribs on The Absorber Plate, Energy 89, pp. 178-190, 2015.

[10] L. Diana, A. G. Safitra and G. Muhammad, "Numerical Investigation of Airflow in a Trapezoidal Solar Air Heater Channel," 2019 International Electronics Symposium (IES), Surabaya, Indonesia, 2019, pp. 248-253.

Acknowledgments

Authors wishing to acknowledge to Politeknik Elektronika Negeri Surabaya for financial support in local research and all collega for support.