VELOCITY AND TEMPERATURE FIELD INSIDE A PASSIVE SOLAR AIR COLLECTOR. PRELIMINARY RESULTS OF CFD ANALYSIS

J. Fiuk, K. Dutkowski

Technical University of Koszalin, Department of Heat Enginnering and Refrigeration, Koszalin, Poland

ABSTRACT

This paper presents preliminary results of a CFD analysis of passive solar air collector. The geometry used in simulation is a representation of a prototype solar air collector which has been investigated experimentally in laboratory of Heat Engineering and Refrigeration Department of Koszalin University of Technology. The external dimensions of collector are 2 x 1 m (heigth x width). The absorber plate is flat. The set of experimental results is used in validation the numerical approximations of the flow field. Most notably, the mean velocity at inlet and temperatures can be easily compared between these two data sets, leading to a comprehensive conclusions. The numerical results are in agreement with experiment. The results revealed a behavior of fluid in the discussed geometry that promotes backflows and may be the cause of momentum loss.

INTRODUCTION

Energy from the Sun is regarded as unlimited, due to the timescale of star life duration. This energy can be utilized in a wide range of appliances, mostly through heat processes (space heating, greenhouse heating, agricultural crop drying). Among the few existing methods, the photothermal conversion of solar energy is the least demanding one in terms of materials and additional energy input. Solar air collectors (or Solar air heaters) are devices which absorb incident solar radiation on a surface called the absorber. The heat is then transferred to air flowing near the wall of absorber. Increased temperature of the fluid is usable for selected application.

A significant amount of research is devoted to design the absorber surface shape, its artificial roughness, flow patterns over the surface in order to maximize the heat transfer coefficient, which in consequence can lead to increase of coefficient of thermal efficiency of the device (Aghaie 2015, Hamid 2015, Jin 2015, Kulkarni 2015, Kumar 2014, Singh 2015).

Thermodynamic estimation of thermal properties of solar collector relies on the estimation of mean heat transfer coefficients and transforming it into nondimensional Nusselt number. Thermal efficiency of solar collector can be calcualted using the well-known correlation of Hottel-Whillier-Blias. The flow in solar collector is inevitably bound to be affected by pressure drops due to frictional losses. From the fluid dynamics point of view, each design of solar collector can be characterized with a coefficient of frictional losses. Many correlations have been found for different shapes of surface extensions, fins, ribs, wires etc. It is proven that many exertions on the absorber surface aiming at increasing the heat transfer coefficient from solid to fluid increases the viscous friction, causes turbulence and in general provokes pressure drops along the flow path and may suppress the flow (Lanjewar 2015, Kumar 2014).

It is therefore of paramount importance to take into account this mutual relation between flow phenomenon and heat transfer . Computational Fluid Dynamics is well-known tool for solving various fluid heat transfer problems with prescribed accuracy. Every solution is an approximation of reality, due to the discrete nature of numerical equations and finite amount of computational time available. A system of equations of mass, momentum and energy are being solved with the appropriate boundary and initial conditions. (Hirsch, 2007) In the field of solar air collectors, some numerical research has been conducted using commercial CFD tools such as ANSYS FLUENT (Aghaie 2015, Hamid 2015, Jin 2015, Kumar 2014, Singh 2015) and ANSYS CFX (Kulkarni 2015). These works describe the effects of numerical investigations on convective heat transfer problems near absorber surface with artificial roughness, fins and ribs causing the flow to be turbulent.

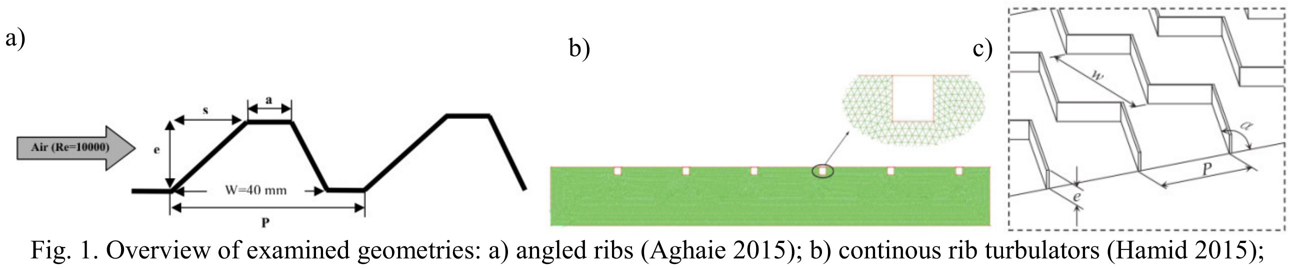


Fig. 1. Overview of examined geometries: a) angled ribs (Aghaie 2015); b) continous rib turbulators (Hamid 2015); c) multi v-shaped ribs (Jin 2015)

The presented paper deals with a natural convection phenomenon in a passive solar air collector. Results of a numerical simulation are presented and compared with experimental data from author's other work. (Dutkowski 2015, Fiuk 2016) Simulation has been conducted in ANSYS CFX commercial software.

NUMERICAL SIMULATION SETUP

An overview of the geometry used in simulation is presented in Fig. 2. Overall dimensions of the collector are: height - 2080 mm, width – 1040 mm, depth – 180 mm. Inlet and outlet are two circular ducts with respective diameter 130 mm and 110 mm. Each of the ducts is extruding from the main body for 500 mm. Inlet is located at the bottom and outlet at the top. This geometry represents the fluid domain in simulation.

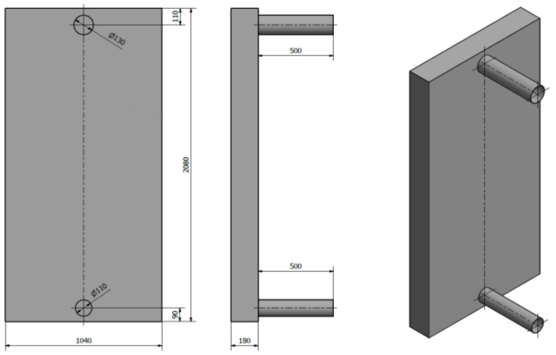


Fig. 2. Geometry overview and dimensions of the modelled solar air collector

The strategy for meshing was to create uniform mesh with slightly increased density of mesh in areas where high gradients of flow parameters are to be expected. These areas included mostly boundary wall areas. The mesh was unstructured with hexahedral elements used. The total number of finite volumes created was: 55873 with total number of nodes: 63366. An overview of mesh in the vicinity of outlet duct is shown on Fig. 3.

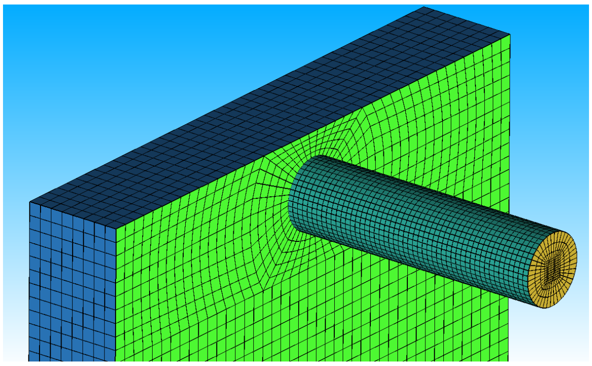


Fig. 3. An overview of meshing near outlet duct

Various surfaces of the mesh were specified in order to enable the creation of boundary conditions. In Fig. 4. all boundary conditions for this case are presented.

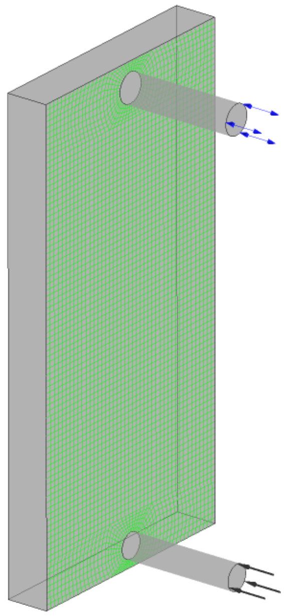


Fig. 4. An overview of boundary conditions

The boundaries are given as follows:

At the inlet a subsonic flow regime with static pressure 1 atm was given, the temperature of air at inlet is 20 ^{o}C.

At the outlet, the condition of opening is given, with static pressure 1 atm and ambient temperature of 25 ^{o}C.

On the absorber surface (Highlighted wall in Fig. 4.) along with no slip wall condition there is constant heat flux q = 300 W/m2

Rest of the walls are adiabatic with no slip condition and smooth surface.

Due to the fact that phenomenon of flow inside the passive solar air collector is driven by natural convection, there can be no "driving" boundary conditions at the inlet or outlet (such as forced mass flow or velocity). Instead, solver has to calculate the flow rate that is occurring due to density gradient in the collector plenum.

The simulation is a steady state case of buoyant flow. The addition of energy to the fluid occurs only through the constrained heat flux on the absorber surface. No external heat losses are modelled.

Turbulence model is k-Epsilon with scalable wall function.

RESULTS OF SIMULATION

The results of preliminary simulation shall be presented in subsequent section. In Fig. 5. a contour plot of temperature, velocity and density of air in the midsection of collector.

a) b) c)

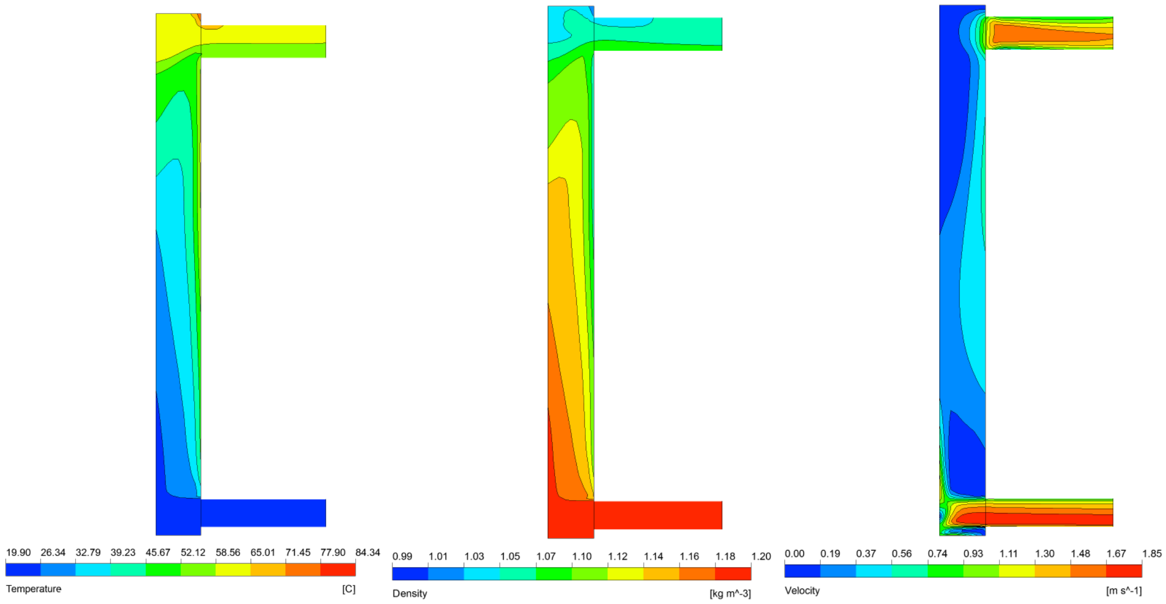
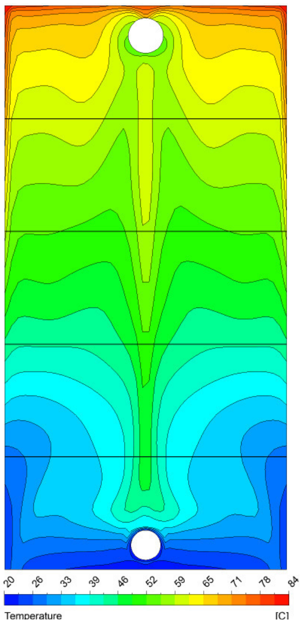


Fig. 5. Contour plots of a) temperature; b) density; c) velocity in the midplane of solar air collector



$\overline{T}_{1,2,3} = 60\ °C$

$\overline{T}_{4,5,6} = 52\ °C$

$\overline{T}_{7,8,9} = 44\ °C$

$\overline{T}_{10,11,12} = 35\ °C$

Fig. 6. Contour plot of temperature of the fluid near absorber wall with average temperatures calculated on shown lines

a)

b)

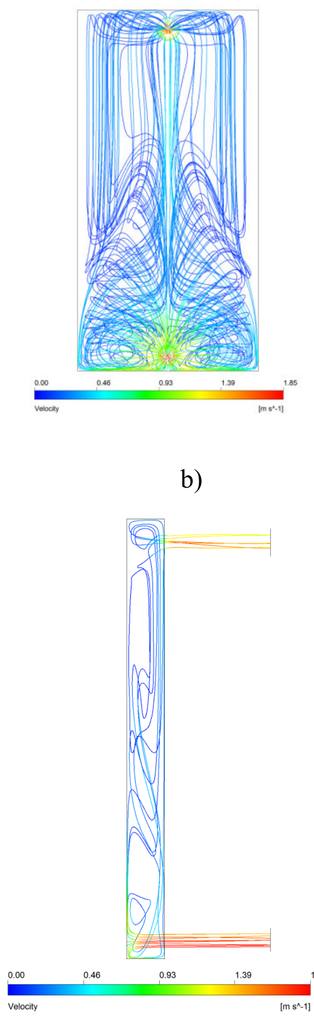


Fig. 7. Streamlines of the flow a) front view; b) side view

a) b)

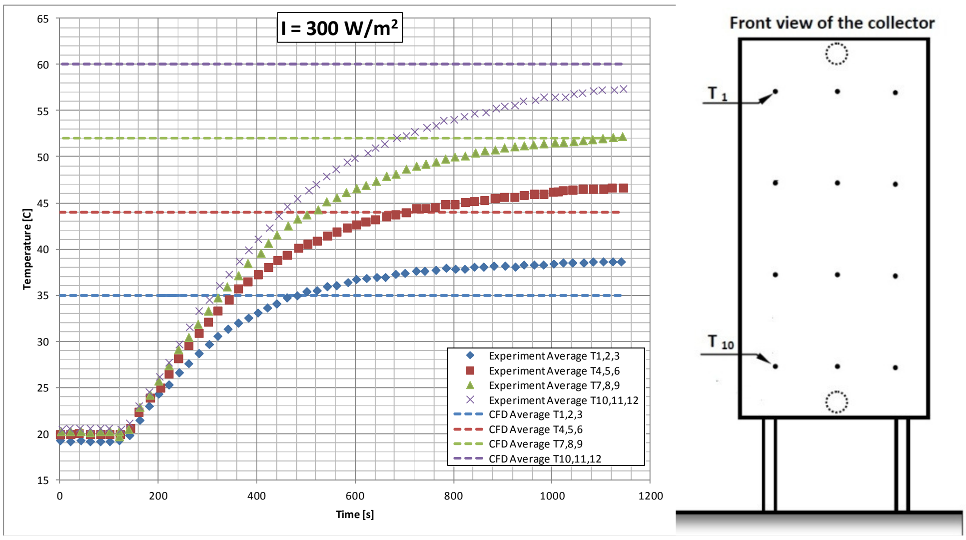


Fig. 8. a) Comparison of temperature profiles in an unsteady experimental dataset with steady state CFD results; b) locations for temperature measurements in experiment (Dutkowski, 2015)

For heat flux equal to 300 W/m^2 the increase of temperature of air flowing through the collector is about ΔT = 35 K, whilist the mean velocity at inlet is $w_{in,mean}$ = 1.53 m/s with air being at standard density ρ_{in} = 1.2 kg/m^3. At the outlet the mean velocity is $w_{out,mean}$ = 1.23 m/s with density ρ_{in} = 1.07 kg/m^3. The mass flow is conserved throughout the domain and equals m = 0.0175 kg/s.

In Fig. 6 it has been shown how temperature field of the fluid develops near the absorber wall. Progressively, travelling upwards the air becomes warmer as is expected of natural convection flow near flat vertical wall (absorber). In Fig. 7a the streamlines of the flow are displayed as viewed from the front. As it can be seen, path of the flow is highly convoluted and it gains momentum near hot wall, travelling upwards. This phenomenon is better observed from the side (fig. 7b). Air flowing near the cold wall is losing momentum and has tendency to flow downwards. The gap between absorber and cover surface is too wide as air has tendency to flow backwards. This behaviour contributes to overall losses of the flow.

COMPARISON WITH EXPERIMENTAL DATA

The results obtained numerically were compared with available data from a previous experimental research done by authors. (Dutkowski 2015, Fiuk 2016) In the experiment, temperature along the absorber was measured in both steady and unsteady state conditions. Fig. 8. shows a chart of temperature of the back side of absorber plate in an unsteady state. A comparison can be made with steady state results from numerical simulation (see Fig. 6). It can be seen that best agreement is for average temperature in taken from points 7, 8, 9, in other cases the agreement is at a reasonable level.

Experimental data includes mean velocity value at the inlet duct cross section. For irradiance I = 300 W/m^2 it is $w_{in,\ EXP}$ = 0.96 m/s as can be seen from Fig. 9. The CFD results are much higher, the area averaged velocity for the inlet duct is $w_{in,\ CFD}$ = 1.53 m/s. For the same values of heat transferred to the fluid the difference in velocities is rather high. This can be explained by the lack of wall roughness impact in CFD simulation. During the numerical setup it was assumed that no roughness is present on walls. Another possible explanation may be that no heat losses are being included in the model, as little as they may be it is possible that the loss of energy through aluminum casing or transparent cover may lead to lower inlet velocity. The volumetric flow at inlet in CFD results was calculated and is $V_{in,\ CFD}$ = 52.15 m^3/s, experimental results show the volumetric flow rate to be $V_{in,\ EXP}$ = 29 m^3/s. This discrepancy is a straightforward consequence of different inlet velocities.

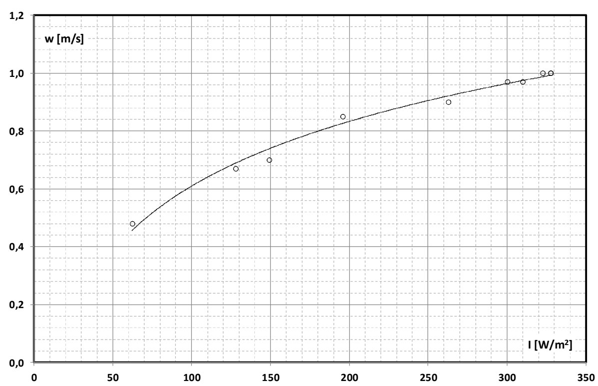


Fig. 9. Inlet mean velocity profile as a function of Irradiance (Dutkowski 2015)

CONCLUSIONS

The presented work is a preliminary insight into numerical computations of flow within passive solar air collector. A device designed to operate within natural convection regime, therefore a buoyant flow solver is required. ANSYS CFX software was used to perform all the calculations.

A model was created and meshed with hexahedral finite volumes (about 56 000). The mesh was deliberately small, so that computational time required to establish a converged solution would not be too high for available processing units. Grid independence tests are to be done in future endeavours, as well as further grid refinement and optimisation.

The obtained results show good agreement with available experimental data from previous research on solar air collectors in laboratory of Department of Heat Engineering and Refrigeration at Koszalin University of Technology. A comparison was made with unsteady state experimental data of temperature profiles at different heigth of the collector. Averaged temperature values were compared, showing that vertical temperature increase in CFD simulation is comparable with results obtained from investigating the prototype collector.

Mean velocity at the inlet duct is overestimated in CFD results, as the value calculated is $w_{in,\ CFD}$ = 1.53 m/s, while the experimental value is $w_{in,\ EXP}$ = 0.96 m/s. This discrepancy may be caused by inaccurate flow modelling, especially the wall roughness and wall friction impact on natural convection. It is worth investigating in further research.

The streamlines of the flow show multiple regions where backward flow can be observed. This implies that the geometry of the collector is not optimal and could be improved by removing such flow behaviour either by putting ribs, flow deflectors or fins.

The preliminary simulation shows great promise in designing solar collectors and improving its thermal efficiency.

NOMENCLATURE

$\dot{m}$	mass flow rate	kg/s
I	irradiance	W/m^2
q	wall heat flux	W/m^2
T	temperature	°C
$\dot{V}$	volume flow rate	m^3/s
w	velocity	m/s

Greek symbols

ρ	mass density	kg/m^3
ΔT	difference of air temperature	K

Subscripts

amb	ambient
CFD	CFD simulation result
EXP	data taken from experiment
in	inlet
mean	mean value
out	outlet

REFERENCES

Aghaie, A. Z., Rahimi, A. B., & Akbarzadeh, A., 2015, *A general optimized geometry of angled ribs for enhancing the thermo-hydraulic behavior of a solar air heater channel - A Taguchi approach*, Renewable Energy, 83, 47–54

Dutkowski K., Piątkowski P., 2015, *Experimental investigation of a prototype passive solar air collector with polycarbonate cellular cover*, Instal, 3, 360, 17–22

Fiuk J., Dutkowski K., 2016, *Experimental prototype research of passive solar air collector*, Ciepłownictwo Ogrzewnictwo Wentylacja, 47/4, 135-141

Hamid, M. O. A., & Zhang, B., 2015, *Field synergy analysis for turbulent heat transfer on ribs roughened solar air heater*, Renewable Energy, 83, 1007–1019

Hirsch, C., 2007, *Numerical computation of internal & external flows. Volume 1 Fundamentals of Computational Fluid Dynamics. Second Edition.*, Elsevier

Jin, D., Zhang, M., Wang, P., & Xu, S., 2015, *Numerical investigation of heat transfer and fluid flow in a solar air heater duct with multi V-shaped ribs on the absorber plate*, Energy, 89, 178–190

Kulkarni, K., Afzal, A., & Kim, K.-Y., 2015, *Multi-objective optimization of solar air heater with obstacles on absorber plate* Solar Energy, 114, 364–377

Kumar, A., 2014, *Analysis of heat transfer and fluid flow in different shaped roughness elements on the absorber plate solar air heater duct*, Energy Procedia, 57, 2102–2111

Kumar, A., Saini, R. P., Saini, J. S., 2014, *A review of thermohydraulic performance of artificially roughened solar air heaters,* Renewable and Sustainable Energy Reviews, 37, 100–122

Lanjewar, A. M., Bhagoria, J. L., Agrawal, M. K., 2015, *Review of development of artificial roughness in solar air heater and performance evaluation of different orientations for double arc rib roughness*, Renewable and Sustainable Energy Reviews. Vol. 43, p. 1214–1223

Singh, S., Singh, B., Hans, V. S., & Gill, R. S., 2015, *CFD (computational fluid dynamics) investigation on Nusselt number and friction factor of solar air heater duct roughened with non-uniform cross-section transverse rib,* Energy, 84, 509–517