CFD model to estimate the Effect of Tilt and Height on the Natural Air Flow inside a Solar Chimney

T. D. KARAPANTSIOS[1,*], A. I. BALOUKTSIS[2], D. CHASSAPIS[2] and M.D. PETALA[1]

[1]: Division of Chemical Technology,
Aristotle University of Thessaloniki,
University Box 116, GR-54124, Thessaloniki,
GREECE

[2]: Department of Mechanical Engineering,
Technological Educational Institution of Serres,
End of Magnesia's Str, GR-62100, Serres,
GREECE

Abstract:
CFD calculations are employed to estimate the average air velocity inside a solar chimney as well as the average temperatures of the glazing, of the black absorber wall and of the chimney air. These parameters are computed as a function of chimney's tilt and height. Given the narrow geometry of typical chimneys, 2D CFD calculations are considered adequate based on the assumption of uniform temperature distributions across the chimney width. The effect of the onset of turbulence is examined by comparing simulations performed with the laminar and a turbulent model at different chimney heights. CFD results are in good agreement with the predictions of a recent engineering model and with experimental results obtained from 1 m solar chimney operated at different tilt positions.

Key-words: solar chimney, chimney height, chimney tilt, natural ventilation, CFD, modeling, solar air heater.

1 Introduction

Solar chimneys differ from conventional chimneys in that one side wall is replaced by a transparent sheet, i.e. glazing, that allows the collection and use of solar irradiation. Solar chimneys have been traditionally used in agriculture for air renewal in barns, silos, greenhouses and in drying of crops, grains, fruits or wood as well as for natural ventilation, climatization and energy conservation in buildings, e.g. [1-3].

Most published works deal with solar chimneys fixed at a specific inclination, usually vertical, as these are easier to construct and operate. Several authors derived approximate analytical expressions or/and employed CFD codes to simulate the air flow and heat transfer in chimneys of varying gap width, e.g., [4-6] .Evidence was provided that for chimneys with gap-to-length ratio close or less than 1:10, the temperature can be assumed uniform across the chimney gap and so 2D models can give reasonably accurate predictions.

Solar chimneys employing inclined collectors can evidently exploit more the incident irradiation to enhance air flow in the chimney. As the inclination of the chimney varies, two things occur that work in opposite directions with respect to the air flow rate. A higher inclination results in a higher exposure of the wall to solar irradiation and hence yield higher heat utilization and more intense buoyant airflow. On the other hand, tilting the chimney reduces the effective pressure head of the chimney and so diminishes air flow. It is apparent that there must be an optimum tilt that leads to the highest flow rate, compromising these two effects.

Recently, Sakonidou et al. [7] examined systematically the effect of inclination for chimneys where the absorbed heat flux depends on the diurnal and seasonal variations of solar

irradiation. They proposed a simple engineering model adequate for design purposes and field applications which does not employ detailed 2D/3D calculations. Their model was capable of estimating the optimum tilt of a solar chimney that maximizes natural air flow on an hourly basis starting from data of monthly average daily total solar irradiation on an horizontal plane and taking into account the variation of the optical properties (transmittance and absorptance) of the glazing with the tilt.
This work aims to compare detailed CFD calculations with results of the engineering model and experiments communicated in [7]. The subtle issue of the onset of turbulence for chimneys above a certain length is examined. In the following, the setup of the CFD model is presented first. Next, a parametric analysis of the problem over a useful range of heights and tilts is presented. Finally, CFD results are compared and discussed against theoretical predictions and experiments by Sakonidou et al. [7].

2 Model formulation

The commercial CFD code Fluent 6.1.18 is employed to simulate the heat transfer and fluid flow inside the chimney. Given the narrow geometry of usual solar chimneys (gap-to-length ratio less than 1:10), a 2D CFD model is considered adequate based on the assumption of uniform temperature distributions across the chimney width. The employed geometrical domain has a variable length (1-12 m) as the first dimension and a fixed gap depth (0.11 m) as the second one. The third dimension (width=0.74 m) is used only for estimation of total flow rates. The selected gap depth and width values are those of the experimental solar chimney employed in [7]. The chimney tilt is also treated as variable in the range 30 - 90° (angles from the horizontal plane).
The computational grid is a pure map mesh with the cells clustered towards the black wall and the glass. In such problems the heat transfer from the walls is the driving mechanism of the flow. As the computed heat transfer coefficient depends on the local air velocity and temperature profiles it is very important to resolve very accurately the boundary layer. Extra care was taken with some initial simulations to estimate and ensure a proper cell size near the wall, as an improper one would result in wrong velocity and temperature boundary layer estimation by the code. A y+ value of around 2 is established at the walls suitable for the turbulence model chosen for the simulations (see below). The final grid for the 1 m high chimney consists of 500 cells along the chimney and 55 cells across the gap (27500 quad cells in total), with an average size of 2 mm, Figure 1. For the taller chimneys the grid size is increased proportionally in the length dimension to maintain the same spatial resolution.

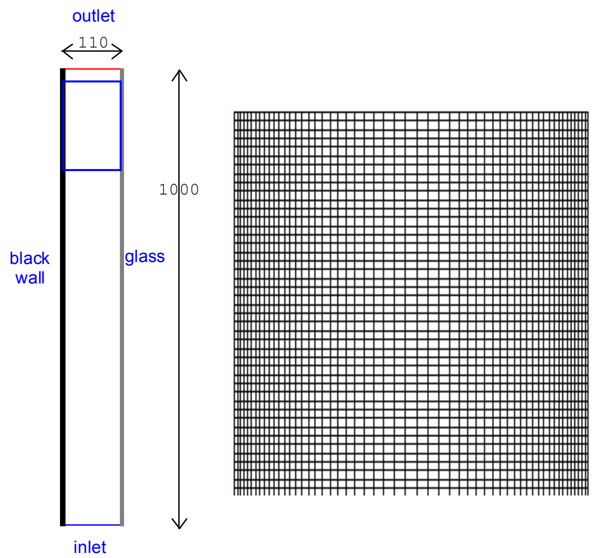

Figure 1. schematic of computational domain (left) and close up of grid near outlet (blue square) showing clustering of cells near the walls (right).

The Reynolds number of the air flow at the exit of the chimney duct is calculated (based on a hydraulic diameter) between 1000 and 4000 depending on height and tilt. Rigorously speaking such average Reynolds numbers are not appropriate for the lower air velocities since then the flow is chiefly inside boundary layers along the chimney walls. However, they can still give a fast gross estimation of the flow field. In any case, these are conditions where transition from laminar to turbulent flow occurs within the chimney and therefore, simulations are performed with both the laminar and turbulent models. For the latter, the shear-stress transport (SST) k-ω model with the transitional flows option active is used [8] which is suitable for low Reynolds turbulent flows. This model combines the traditional two-layer turbulent zonal model with enhanced wall functions.
The energy equation is employed to model the heat transfer phenomena with the Boussinesq approximation to hold for the density of air with a specified constant heat expansion coefficient.
Irradiation modeling is implemented using the Surface-to-Surface model [8] which accounts for the irradiation exchange in an enclosure of gray-diffuse surfaces. The energy exchange between two

surfaces depends in part on their size, separation distance, and orientation. These parameters are accounted for by a geometric function called a "view factor", calculated by the code. The main assumption of the Surface-to-Surface model is that any absorption, emission, or scattering of radiation by the air (between the surfaces) can be ignored; therefore, only "surface-to-surface" radiation need be considered for the analysis.

To allow meaningful comparisons with the engineering model proposed by Sakonidou et al. [7] it is decided not to use the CFD code to compute the heat flux absorbed by the black wall and the glass cover from solar irradiation components (direct, diffuse, ground-reflected) but instead employ the hourly average values absorbed by the solar chimney of varying tilt and height as calculated by their model based on available monthly average daily values of total irradiation on a horizontal plane. To estimate heat loses, the heat transfer coefficient from the black wall and the glass cover towards the ambient air is 0.9 W/m^2K and 9 W/m^2K, respectively. These are the values proposed in [7]. In the present analysis, it is assumed that the incident solar irradiation is sufficient to bring the chimney's body to its steady state temperature. The imposed boundary conditions for the two chimney walls, (glazing and absorbing black wall) are that they both have zero slip and internal emmitance of 0.95.

Data for monthly average daily total irradiation and monthly average ambient temperature are taken from ELOT, [9] - the Greek Organization of Standardization - for Serres, a city in North Greece (latitude 41° 07′, longitude 23° 34′, altitude 32 m).

3 Results and discussion

Due to space limitations only simulations at a summer day are presented: day 196 (mid July), monthly average daily total irradiation on a horizontal plane 23.1 MJ/m^2 and monthly average daily ambient temperature 28.9°C.

Initial runs are performed with the aim to evaluate and calibrate the appropriate models and boundary conditions. On this account, the results of the laminar model are compared with those of the k-ω SST turbulence model. Interestingly enough, for the 1m high chimney both models produce similar results as shown in Figure 2. The velocity magnitude consists of two peaks very close to both walls while in the middle of the chimney velocities are very low. The temperature fields are also alike.

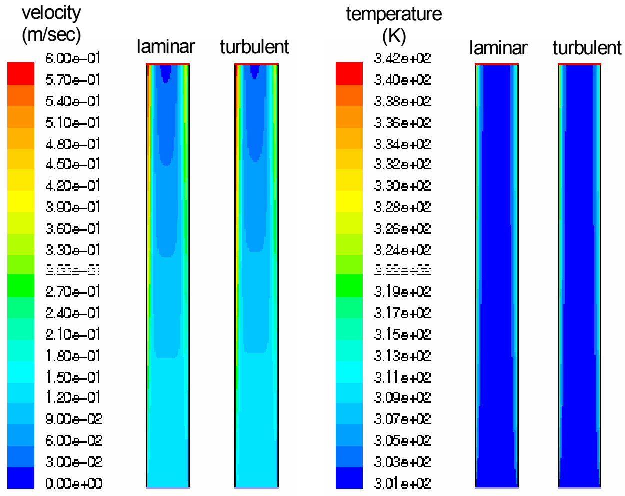

Figure 2: Contours of velocity and temperature. Comparison of laminar and turbulent simulations for a vertical 1 m high chimney.

However, for the 4m high chimney the results appear to deviate, Figure 3. As the chimney gets taller more heat is transferred to the flow leading to higher velocities. As the velocities increase so does the tendency of the flow to transit to turbulent. The laminar flow model is not capable of capturing such a phenomenon and so no change is found with the laminar simulation. On the contrary, the k-ω SST turbulence model is capable of predicting this as well as its consequences in the mixing and expansion of the thermal boundary layers.

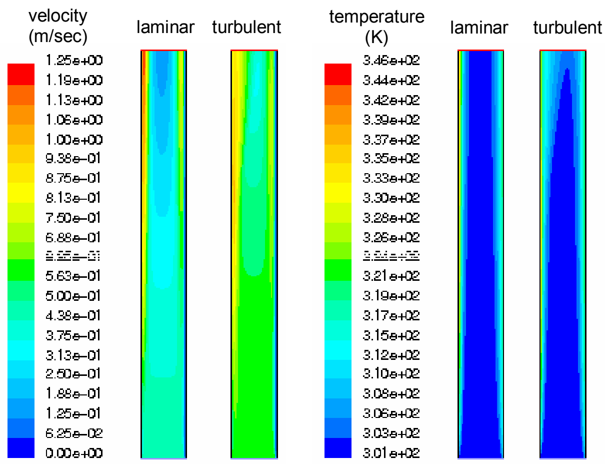

Figure 3. Contours of velocity and temperature. Comparison of laminar and turbulent simulations for a vertical 4m high chimney. (The image has been compressed by a factor of 4 in the vertical direction to allow comparison with 1m chimney).

This transition is also depicted in a plot of temperature of the black wall along its height,

Figure 4. The temperature predicted by the turbulent model shows a “kink” at a little below 3 m height. This is where transition to turbulence takes place. Due to turbulence the heat transfer coefficient at the wall increases, and the wall gradually cools down towards the exit. In contrast, the laminar simulation can not capture this feature. Turbulent flow increases air mixing and temperature uniformity across the chimney and so the resulting velocity profiles are also more uniform in the turbulent simulations. Based on the above, all simulations henceforth are performed using this turbulence model.

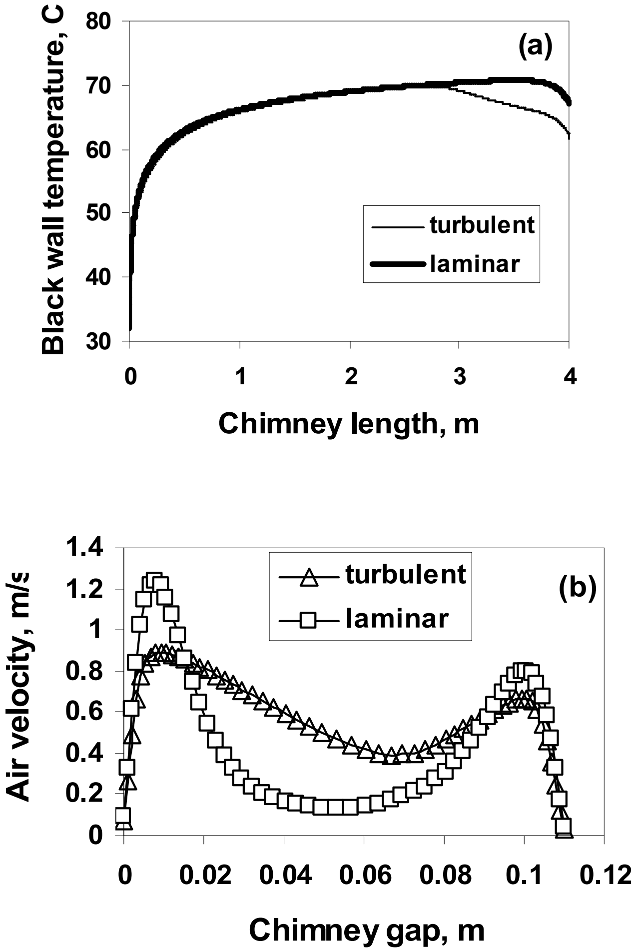

Figure 4. Predicted temperature along the length of the black wall (a) and velocity profiles at the outlet (b) for laminar and turbulent simulations of a 4m high chimney.

Figure 5(a) shows the air velocity profile across the chimney gap at the exit of a vertical chimney, with chimney length as a parameter. The shape of the velocity profiles for the two smaller lengths (1 and 2 m) is typical of non-interacting boundary layers. Two local maxima are observed near these walls (the higher for the hotter absorber wall) whereas at the centre of the chimney the velocity is close to zero. For higher chimneys the two boundary layers start to interact leading to less pronounced local maxima and a smoother velocity front with appreciable velocities at the center of the chimney. Figure 5(b) displays the corresponding mass-weighted - “cup-mixing” - air temperature profiles. As expected, the higher air temperatures are near the black absorber wall which seems to be the main heat supplier of the system.

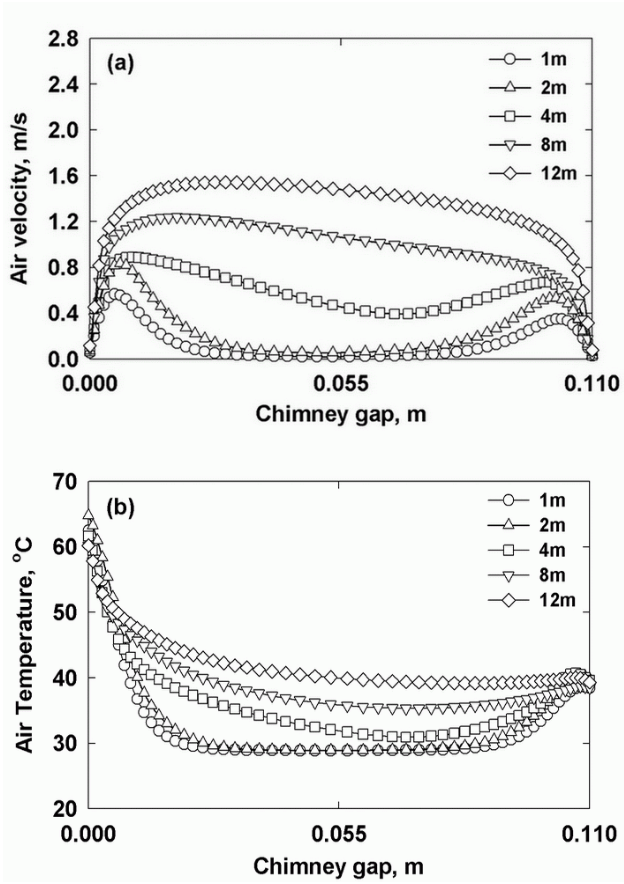

Fig. 5. Air velocity profile (a) and air temperature profile (b) across the chimney gap for different chimney lengths at the vertical position.

Figure 6 shows the influence of the tilt on (a) air velocity and (b) mass- weighted air temperature across the chimney gap for a 1m chimney. The main features of both the velocity and temperature profiles do not seem to vary with tilt. The air temperature in contact with the walls for the vertical chimney is appreciably lower than the values for the other angles but this is not so for the velocity. This manifests a different influence of tilt on heat transfer and fluid flow in the chimney with the consequence that the maximum energy uptake not to coincide with the maximum air flow rate.

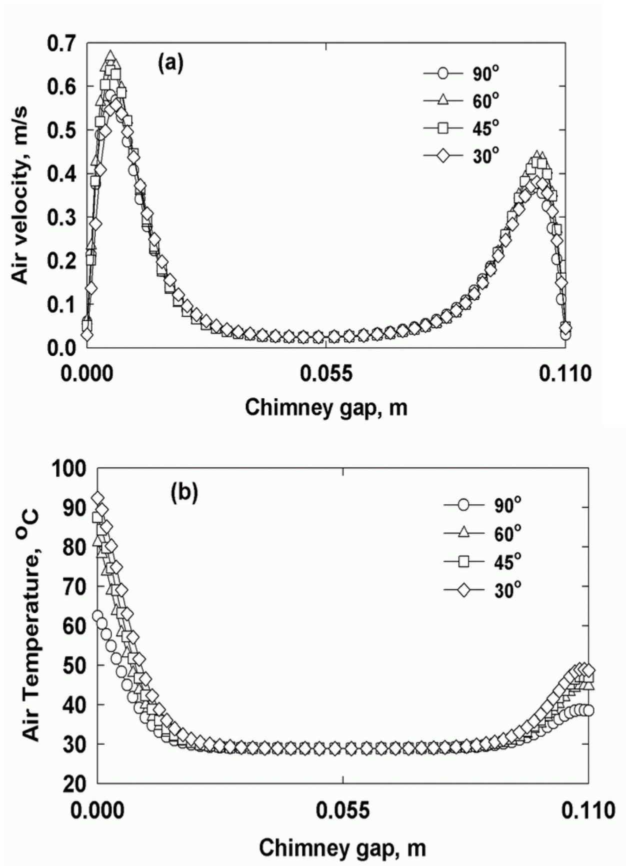

Fig. 6. Air velocity profile (a) and air temperature profile (b), across the chimney gap for different tilt angles for a chimney of 1m length.

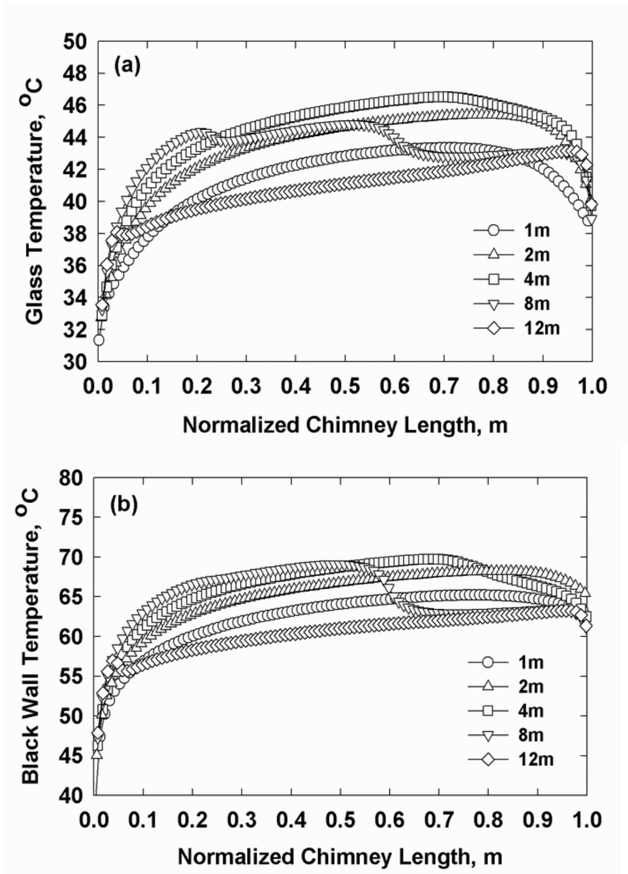

Fig. 7. Temperature of glazing (a) and absorber wall (b) along the chimney length for different chimney lengths at the vertical position.

The influence of chimney length on (a) the glazing temperature and (b) the absorber temperature is shown in Figure 7 for an inclination of 90°. Normalization in the length scale is performed by division with the total chimney length. Both walls are heated up significantly within a very short distance from the inlet of the chimney. Near the top of the chimney, radiation heat losses reduce the temperature of the walls. While for chimneys 1 and 2 m length the temperature of the walls increases monotonously with length, for higher chimneys there is a point where the temperature of the wall starts to decrease. This is attributed to onset of turbulence. For very high chimneys, e.g., 12 m, the wall temperatures increase almost linearly with height indicating a pretty constant turbulent field.

Next, the CFD results are compared against predictions from the engineering model and experimental measurements of Sakonidou et al. [7]. Figure 8 compares the glazing and absorber temperatures for different inclinations of the chimney. Error bars denote the standard deviation of measurements. The agreement between predictions and measurements is good.

4 Conclusion

The developed CFD model predicts the velocity and temperature distribution of air inside the chimney and the temperatures of the glazing and the black painted absorber, as a function of tilt and height using as input only the solar energy absorbed by the solar chimney walls. It is found that the tilt for the maximum air temperature is not the same with the tilt for the maximum air flow rate.

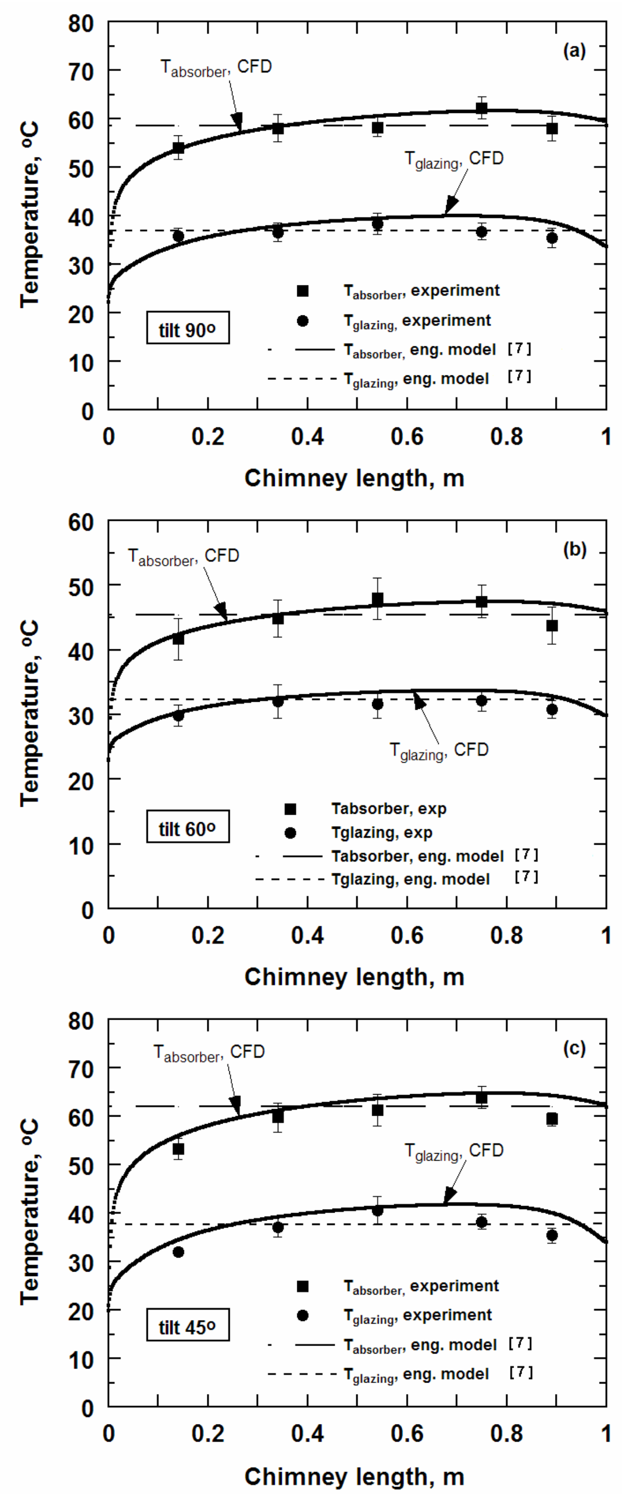

Fig. 8. Comparison between CFD results against the engineering model predictions and experimental results of [7] as regards the temperatures of the glazing and the absorber along the chimney length (a) for 90º, (b) 60º and (c) 45º angle of tilt. Plot (a): day=305, H=9.70 MJ/m^2, T_{amb}=21.7°C. Plot (b): day=305, H=9.05 MJ/m^2, T_{amb}=19.1°C. Plot (c): day=306, H=9.90 MJ/m^2, T_{amb}=23.1°C.

Moreover, it is shown that for chimneys taller than ~3 m, heat transfer from the walls to the chimney air is drastically enhanced, indicating turbulent flow conditions. The reasonable agreement between CFD results and the engineering model predictions and experimental results of [7] encourages the use of the engineering model as a tool for evaluating design parameters and for comparative studies.

Acknowledgments:
The project is co-funded by the European Social Fund and National Resources - (EPEAEK-II) ARHIMIDIS.

References:

[1] Vlachos, N.A., Karapantsios, T.D., Balouktsis, A.I., Chassapis, D., Design and testing of a new solar tray dryer, *Drying technology*, Vol. 20, 2002, pp. 1243-1271.

[2] Bansal, N.K., Mathur, J., Mathur, S., Jain, M., Modeling of window-sized solar chimneys for ventilation, *Building and Environment*, Vol. 40, 2005, pp.1302-1308.

[3] Heras, M.R., Jiménez, M.J., San Isidro, M.J., Zarzalejo, L.F., Pérez, M., Energetic analysis of a passive solar design, incorporated in a courtyard after refurbishment, using an innovative cover component based in a 2005, pp. 85-96.

[4] Andersen, K.T., Theoretical considerations on natural ventilation by thermal buoyancy, *ASHRAE Transactions*, Vol 101, No. 2, 1995, pp. 1103–1117.

[5] Gan, G., Riffat, S.B., A numerical study of solar chimney for natural ventilation of buildings with heat recovery, *Applied Thermal Engineering*, Vol. 18, 1998, pp. 1171-1187.

[6] Moshfegh, B., Sandberg, M., Flow and heat transfer in the air gap behind photovoltaic panels, International *Journal of Renewable & Sustainable Energy Reviews*, Vol. 2, 1999, pp. 287-301.

[7] E. P. Sakonidou, T. D. Karapantsios, A. I. Balouktsis and D. Chassapis, "A Model for Estimating the Optimum Tilt of a Solar Chimney for Maximum Air Flow", *Solar Energy.* in press.

[8] Fluent user's guide, Fluent Inc., 2003.

[9] ELOT, Greek Bureau of Standards No. 1291, 1991