Proceedings of the International Conference on Heat Transfer and Fluid Flow
Prague, Czech Republic, August 11-12, 2014
Paper No. 173

Heat Balance of a Simulated Natural Convection Vertical Cylinder with Inlet Constriction and Flow Reversal at Chimney Exit

Chi-Ming Chu*, Md. Mizanur Rahman, Sivakumar Kumaresan and Hooi-Kim Phang
Thermal and Environment Research Group, Material and Mineral Research Unit,
Faculty of Engineering, Universiti Malaysia Sabah
chrischu@ums.edu.my; mizanur@ums.edu.my; shiva@ums.edu.my; phang83@gmail.com

Abstract - In a series of experiments temperature and pressure drop data were obtained from an air-cooled heat exchanger model with cross-sectional flow areas of 0.56 m^2, 1.00 m^2 and 2.25 m^2 operating under natural convection that indicated significant cold inflow, resulting in the reduction of effective chimney height. Further experiments were carried out in seven configurations of a smaller scale natural draft air-cooled heat exchanger model at two different supplied heat loads of approximately 0.75 kW and1.25 kW, to determine the effects of wire mesh screen positioning on the chimney performance. The two series of experiments appeared to corroborate each other satisfactorily, where smoke was employed for flow visualization in the second series of experiments. A simulation of the effect of wire mesh on cold inflow is required to understand the mechanism. Simulation by computational fluid dynamics using Phoenics was performed for a vertical cylinder of diameter 100m and height of 200m, at opening ground clearances of 10m and 14.68m, with heated air temperature of 50°C and ambient air temperature of 30°C in a still surrounding. Results confirmed that cold inflow could be simulated for cylinders of these arrangements and that wire mesh was found to mitigate the flow reversal occurring at chimney exit, in agreement with earlier experimental observations. By mitigating cold inflow or flow reversal the updraft was improved by a significant 14 per cent and the heat discharge in the operation improved from -75 percent to virtually no difference from the input.

***Keywords*:** Flow Reversal, Chimney, Vertical Cylinder, Natural Convection.

Nomenclature

a, b	coefficients of pressure drop equation
k	phase number, 1 or 2 in Phoenics.
Pr_t, Pr_l	the turbulent and laminar Prandtl numbers.
S_ϕ	the source term in CFD calculation
t	time
U	vector velocity in CFD calculation
u, v, w	x-, y- and z- components of velocity vector
w_s	superficial vertical velocity
x_k	distance in k-phase
Δp	pressure drop
Γ_ϕ	the diffusive exchange coefficient for ϕ
ϕ	the variable in question in CFD calculation
ρ	density
ν_t, ν_l	the turbulent and laminar viscosities

1. Introduction

Chimneys have been around since time immemorial. Whether the field is cooling tower or solar chimney or vertical channel cooling, a commonly acknowledged problem of chimneys is that they experience unstable flow with breezes leading to undesirable downdraft (Costola and Etheridge, 2008;

Kröger, 2004; Bouchair, 1994; Baer et al, 1980). While no recent industrial data have been reported that cold inflow is occurring in solar chimneys or cooling towers (Zhou et al 2010; Goodarzi and Kaimanesh, 2013), there is increasing evidence under laboratory and pilot scale tests (Chu et al, 2012a & b) that this is a factor that needs to be considered for mitigation during the design stage.

Under laboratory conditions, the first systematic study of cold inflow were undertaken by Jörg and Scorer (1967) and followed by others, in ascertaining the critical Froude and Reynolds numbers for the onset of cold inflow (Modi and Torrance, 1987; Fisher and Torrance, 1999).

Andreozzi et al (2009) were able to simulate numerically the experimental results of Auletta et al (2001), and confirmed by the cold inflow experimental study reported in Chu et al (2012a). They contended that the geometry studied in their numerical investigation was important different technical fields, such as in electronic cooling and in building and houses' solar components. Chu et al (2012b) carried out further experiments by examining seven configurations of wire mesh positions in a small chimney model. Smoke flow visualizations showed that the chimneys were prone to cold inflow and it could be alleviated by placing a wire mesh on the chimney exit. The results provided some basic understanding of how cold inflow had occurred at the chimney exits, and the conditions under which the phenomenon could be repeated were noted. However, it remains to be investigated by CFD as to the conditions under which the phenomenon of cold inflow occurs and whether wire mesh does indeed restore the loss of draft, and if so, to what extent.

CFD simulations on cooling towers have been modelled as vertical cylinders by Lu et al (2012) for geothermal energy applications. The same concept is being adopted here for the present study.

2. Basis of Simulation

To demonstrate the phenomenon using CFD, the set up was designed to be as simple as possible to ensure that the effect sought to be demonstrated is traceable to the manipulated variable. A vertical straight cylindrical tube of 100-m diameter and 200-m height, located at an opening height above ground, will be simulated for a heating source of 5m height at 50°C and an ambient air temperature of 30°C, under atmospheric pressure in a still surrounding (Figure 1). The opening inlet height of the chimney had two settings: firstly 14.68m as it is the height of the highest natural draft cooling tower (Busch et al, 2002) and secondly 10m to show the effect of reducing the gap. The ground of the chimney (Solid Base) was set at 120 m diameter, where its magnitude was not regarded to be critical. The heating source was located at 0.5 m into the chimney inlet and it was assumed that air was receiving thermal energy from the heater instantaneously upon contact in rising through the chimney. Normal wall friction loss behaviour was allowed for the fluid flowing past the wall.

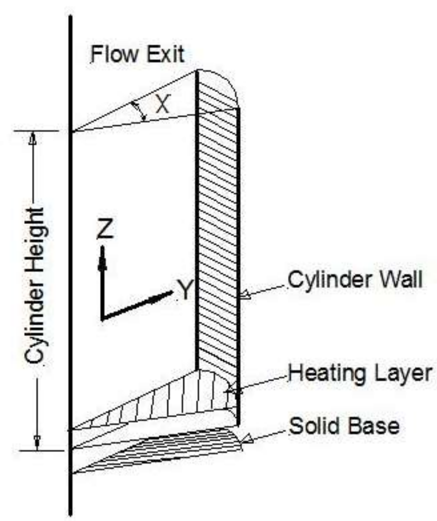

Fig. 1. Flow Domain of the Vertical Cylinder.

The CFD software employed was the Phoenics 2012, developed by CHAM, London. The cylinder was simulated at steady state in cylindrical-polar mode using a 0.20-radian(X-) slice of the total flow area, with the domain set at 200m for horizontal (Y-) direction and 300m for vertical (Z-) direction. The number of mesh cells in the flow domain's X-direction is 12, Y-direction is 60 and 36 in the Z-direction, which shows that increasing the number of mesh cells from 10 × 50 × 30 does not alter the draft rate significantly. The density is as shown in Figure 2.

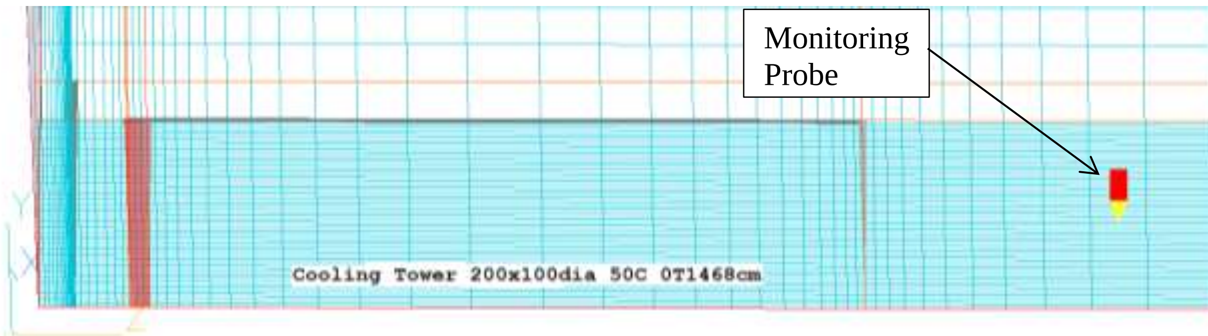

Fig. 2: Numerical mesh grid of the vertical cylinder.

The single-phase governing equations in the generalised form is (CHAM):

$$\frac{\partial(\rho\phi)}{\partial t}+\frac{\partial}{\partial x_k}\left(\rho U\phi-\Gamma_\phi\frac{\partial\phi}{\partial x_k}\right)=S_\phi \quad (1)$$

where ϕ = the variable in question

ρ = density

U = vector velocity

Γ_ϕ = the diffusive exchange coefficient for ϕ

S_ϕ= the source term

Particular forms are:

Momentum $\phi = u, v, w$ (2)

$$\Gamma_\phi = \rho(\nu_t+\nu_l) \quad (3)$$

$$S_\phi = -\frac{\partial p}{\partial x_k}+\text{gravity}+\text{friction}+.... \quad (4)$$

Enthalpy ϕ= h (5)

$$\Gamma_\phi = \rho\left(\frac{\nu_t}{\Pr_t}+\frac{\nu_l}{\Pr_l}\right) \quad (6)$$

$$S_\phi = -\frac{Dp}{Dt}+\text{heat sources}+.... \quad (7)$$

Continuity ϕ = 1 (8)

Γ_ϕ= 0 (9)

S_ϕ= 0+ boundary sources (10)

where ν_t, ν_l are the turbulent and laminar viscosities, and Pr_t, Pr_l are the turbulent and laminar Prandtl numbers.

The default and recommended turbulence model is the Chen-Kim's (1987) modified κ-ε model with the following settings: σ_κ, coefficient for turbulent kinetic energy κ = 0.75; σ_ε, coefficient for its dissipation rate ε = 1.15; model constants C_1 = 1.15, C_2 = 1.9 and C_3 = 0.25. The wire mesh was modeled as a porous plate with a pressure drop formula of

$$\Delta p = a w_S^b \tag{11}$$

where Δp is the pressure drop for a single layer of wire mesh,
w_S is the on-coming superficial velocity, and
a = 4.42, b = 1.361,.

Initial values at the monitoring probe were ambient pressure and temperature, 1.0 x 10^{-10} for v(radial velocity), w (vertical velocity) and turbulent parameters. The equations (1) to (10) were represented in finite volume discretised form and solved in steady state, using a variant of the SIMPLE numerical scheme such as SIMPLEST as the programme selects, with a global convergence criterion of 1 per cent for pressure, velocity and temperature at the monitoring probe. The natural convection process requires the use of elliptic-staggered formulation instead of parabolic, or the marching-forward step calculations. At both the inlet and outlet openings, the flow is set to be at fixed pressure and not at fixed velocity. All the simulations were required to be primed by performing transient simulations for up to 900 seconds, followed by steady state. The maximum number of sweeps at steady state was set at 6000. The process took less than 3 hours per run.

3. Results and Discussion of Simulation

Since the main objective is to demonstrate flow reversal, it is found that temperature contour plot is able to highlight mixing and reversal more succinctly than velocity contour plots. However, vector arrows showing the flow are also superimposed on the temperature contours. For heat balance, the averaged chimney inlet and outlet temperatures are used together with the mass flowrate through the chimney exit area, where the air specific heat capacity is taken to be 1000 $Jkg^{-1}K^{-1}$.

3.1 Without Wire Mesh for 14.68m Opening Height (Mode 0)

In Figure 3, 14.68 m is taken from the inlet opening height of the cooling tower reported by Burch et al (2002). It may be observed that in the exit zone, the temperature near the wall is mixed with the ambient air temperature and lowers it from 50°C.

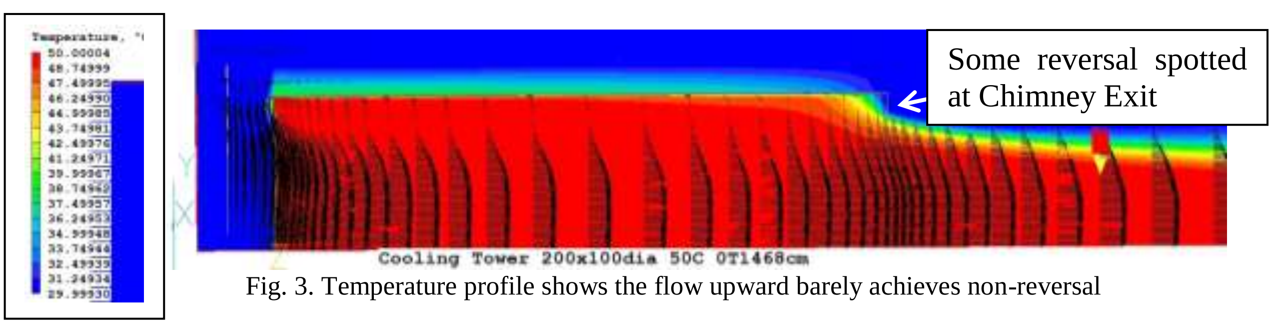

Fig. 3. Temperature profile shows the flow upward barely achieves non-reversal

3.2 With Wire Mesh for 14.68m Opening Height (Mode 1)

In the Mode 1 arrangement (Figure 4), the contour plot shows the temperature at the chimney exit zone appearing to maintain a more gradual change of radial temperature and the velocity vectors do not show similar reversal of the no-mesh situation (Mode 0) (Figure 3). The wire mesh acts as a re-distributor of velocity profile in this mode. It is worth mentioning that the pressure contour plot shows the pressure builds up at the upstream side of the wire mesh, and is positive relative to the ambient, unlike in the Mode 0 case, which show negative pressure profiles inside the chimney. The plot is not shown here due to lack of space.

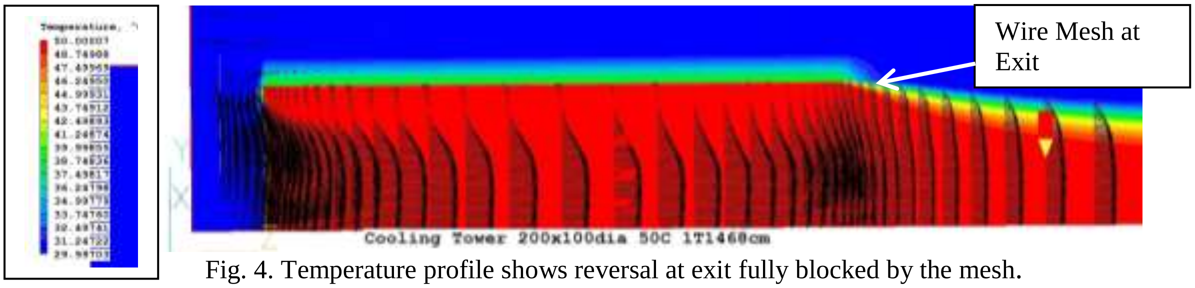

Fig. 4. Temperature profile shows reversal at exit fully blocked by the mesh.

3.3 Without Wire Mesh for 10m Opening Height (Mode 0)

As can be seen in Figure 5, in Mode 0, the cylinder with inlet opening of 10-m clearly suffered from flow reversal that penetrated down to the inlet. This has resulted in temperature contamination, and the heat balance cannot be matched between the input and output based on temperature readings and measured updraft. Velocity vectors also reverse direction near the wall.

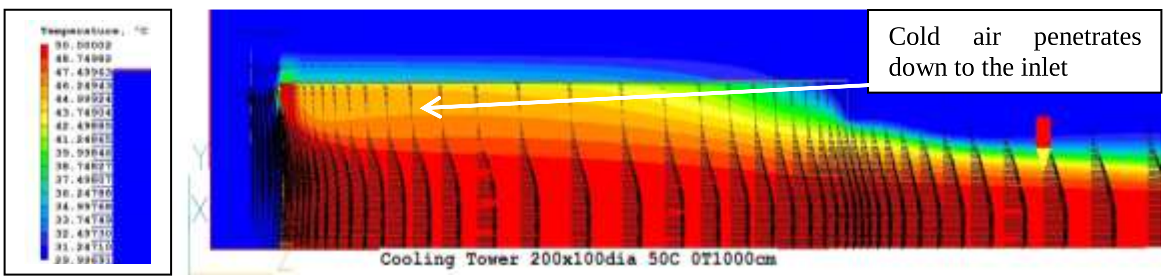

Fig. 5. Flow reversal in Mode 0 of 10-m inlet opening as indicated by temperature profile and the vectors

3.4 With Wire Mesh for 10m Opening Height (Mode 1)

Figure 6 shows that in the Mode 1 arrangement, the velocity at the chimney exit appears to maintain a sufficient temperature level across the diameter for a uniform flow profile upwards without penetration unlike the no-mesh situation (Mode 0). The temperature of the entire inside chimney can be seen to be almost the same as that of the heater's.

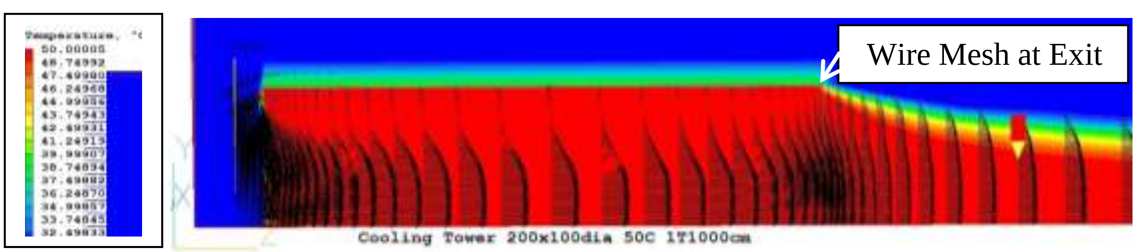

Fig. 6: The installation of a wire mesh succeeds in blocking flow reversal at chimney exit for the 10-m inlet opening height

Table 1. Comparison of Vertical Cylinder Natural Convection Performance between the two Inlet Opening Heights for both Modes (Diff. = Mode 0 – Mode 1 (T-only) and [M0-M1]/M1).

Inlet opening height (m)	14.68			10.00		
Average Values	**Mode 0**	**Mode 1**	**Diff.**	**Mode 0**	**Mode 1**	**Diff.**
Mass flowrate × 10^{-3} (kg/sec)	49.18	49.14	+0.08%	31.62	36.92	-14.35%
Velocity (ms^{-1})	5.39	5.38	+0.19%	3.46	4.04	-14.35%
Inlet Temperature (°C)	31.7	31.6	+0.1 K	33.3	32.7	+0.6 K
Outlet Temperature (°C)	43.6	50.00	-6.4 K	39.0	50.00	-11.0 K
Heat load (MW)	585.2	969.7	-39.7%	180.2	724.1	-75.1%

In Table 1, the performance parameters of the vertical cylinder are shown to compare what the CFD predicts about the two inlet opening heights in both modes. The average values of updraft mass flowrate, i.e. flow in the Z- direction only, are extracted from the simulations.

The simulation has replicated previous experimental observations of the occurrence of flow reversals at the exit of a chimney operating under natural convection (Chu and Rahman 2009; Chu et al 2012a,b), by setting clearances of the vertical cylindrical chimney from ground at heights typical of natural draft cooling towers, namely 14.68m and 10m. The effect of wire mesh installed at the exit of the chimney has also been replicated. Wire mesh is confirmed by the simulation to re-distribute the flow to the extent that flow reversal at the exit is minimized. The mechanism of how it distributes the flow warrants further study. What can be said here is that in the pressure contour plot not shown here, the pressure inside the chimney with wire mesh installed (Mode 1) is positive with respect to the ambient, unlike the Mode 0 case. The results show that, in agreement with Andreozzi et al's (2009) simulation, that even in a still ambient environment, cold inflow to a chimney occurs provided the right condition exists.

The use of a cylinder instead of a square duct was found not to affect the results of the simulation. Cylinder is of the shape that causes less distortion of flow than a square duct, having no sharp corners unlike the latter. In fact, the result enhances the hypothesis that the flow reversal, or cold inflow, occurs under a still ambient surrounding, without awkward geometrical shapes to cause the reversal, but due to the flow constriction at inlet, similar to the Bottom-only configuration in Chu et al (2002b).

The simulation also replicated the enhancement of flow by placing wire mesh on the chimney exit. The mass flow rate enhancement in Mode 1 over Mode 0 for the case of 10-m inlet opening height is 14 per cent, which is close to the range of 45 to 90 per cent that was obtained in the research facility described in Chu and Rahman (2009), but because of different aspect ratios, 100m dia. × 200 m high, against 0.75m wide × 0.75m long × 0.30m high, the enhancement would be different.

In the 14.68m-inlet opening height case, the addition of wire mesh resulted in a reduction in draft and heat load, which from normal observations is expected of the effect of installing wire mesh in the path of a flow. This serves as a convenient indicator that flow reversal had not taken place with this inlet opening, though it appeared to be on the verge of happening.

Examination of the updraft for without mesh top (Mode 0) and with mesh top (Mode 1) demonstrates that cold inflow is detrimental to the updraft of the cooling tower, and that by mitigating it the updraft can be improved by up to 14 per cent and the heat balance becomes representative of the operation. The improvement is counter to normal observation as mentioned above.

4. Conclusion

- Flow reversal at the exit of chimneys operating under natural convection has been replicated buy simulation of a vertical cylinder of 100m diameter and 200m height.
- Flow reversal appears to depend on the inlet constriction.
- Wire mesh at the exit does not enhance the draft if there is no flow reversal.
- The simulation results concur with the experiments of Chu et al (2012a,b) and the simulation of Andreozzi et al's (2009).
- By mitigating cold inflow / flow reversal the updraft was improved by up to a significant 14 per cent and the heat balance became representative of the operation when it was improved by up to 75 per cent.

Acknowledgements

The authors would like to express their sincere thanks to the Ministry of Higher Education, Malaysia, for providing a research funding under grant No. FRG0197-SG-1/2010 and FRG0022-TK-1/2006.

References

Andreozzi, A., Buonomo B. and Manca O., (2009), Thermal management of a symmetrically heated channel–chimney system. International Journal of Thermal Sciences, **48** (3): 475-487.

Auletta, A., Manca, O., Morrone, B. and Nasoe, V., (2001), Heat transfer enhancement by the chimney effect in a vertical isoflux channel, International Journal of Heat and Mass Transfer, Elsevier, 44, pp. 4345-4357.

Baer, E., Ernst, G. and Wurz, D., (1980), Effect of wind on the flow in the chimney of a Natural Draft Cooling Tower, VGB Kraftwerkstechnik,Vol. 60, No. 11, pp. 875-881.

Bouchair, A., (1994), Solar chimney for promoting cooling ventilation in southern Algeria, Building Service Engineering, Research and Technology, 15, (2), pp. 81–93.

Busch, D., Harte, R., Krätzig, W.B., Montag, U., (2002), New natural draft cooling tower of 200 m of height, Engineering Structures, 24 1509–1521, Elsevier.

Chen, Y.S. and Kim, S.W., (1987), Computation of turbulent flows using an extended k-e turbulence closure model, NASA CR-179204.

Chu, C. M., Rahman, M.M. and Kumaresan, S., (2012a), Effect Of Cold Inflow On Chimney Height of Natural Draft Cooling Towers, Vol. 249, August, pp. 125– 131, Nuclear Engineering and Design, Elsevier.

Chu, C.C.M., Chu, R.K.H. and Rahman, M.M., (2012b), Experimental Study of Cold Inflow and its Effect on Draft of a Chimney, pp.73-79, Advanced Computational Methods and Experiments in Heat Transfer XII, WIT Press.

Chu, C.M. and Md. Mizanur Rahman, (2009), A Method to Achieve Robust Aerodynamics and Enhancement of Updraft in Natural Draft Dry Cooling Towers, HT2009-88289, 19-23 July, "ASME Summer Heat Transfer Conference", San Francisco, Calif., U.S.A.

Costola, D. and Etheridge, D.W., (2008), Unsteady natural ventilation at model scale – flow reversal and discharge coefficients of a short stack and an orifice, Building and Environment. **43**: 1491 – 1506.

Fisher, T.S. and Torrance, K.E., (1999), Experiments on chimney-enhanced free convection, Journal of Heat Transfer, ASME, 121, pp. 603-609.

Goodarzi, M. and Kaimanesh, R., (2013), Heat rejection enhancement in natural draft cooling tower using radiator-type windbreakers, Energy Conversion and Management, 71, pp. 120-125, Elsevier.

Jörg, O. and Scorer, R.S., (1967), An experimental study of cold inflow into chimneys, Atmospheric Environment", Vol. 1, No 6, pp 645-646, Elsevier Ltd.

Kröger, D.G., (2004), Air-cooled heat exchangers and cooling towers, PennWell Books, Tulsa Oklahoma.

Lu, Y., Guan, Z. and Gurgenci, H., (2012), CFD simulations on small natural draft dry cooling towers, "18th Australasian Fluid Mechanics Conference", Launceston, Australia. 3-7 December.

Manca, O., Musto, M. and Naso, V., (2002), Flow visualization and air temperature measurements in symmetrically heated vertical channels with adiabatic extensions, American Society of Mechanical Engineers, Heat Transfer Division, (Publication) HTD**,** Volume 372, Issue 1, Pages 123-134, ASME.

Modi, V. and Torrance, K.E., (1987), Experimental and Numerical Studies of Cold Inflow at the Exit of Buoyant Channel Flows, Journal of Heat Transfer, ASME, N.Y., Vol. 109, Issue 2, pp. 392-399.

Zhou, X.P., Wang, F. and Ochieng, R.M., (2010), A review of solar chimney power technology, Renewable and Sustainable Energy Reviews, 14, pp. 2315-2338, Elsevier.

Web sites:

CHAM, http://www.cham.co.uk/phoenics/d_polis/d_lecs/general/maths.htm, Concentration, Heat and Momentum Limited, Bakery House, 40 High Street, Wimbledon Village, London, SW19 5AU, Consulted 30 Apr 2014.