World Applied Sciences Journal 12 (4): 568-575, 2011
ISSN 1818-4952
© IDOSI Publications, 2011

Feasibility Study of the Thermal Performance Improvement of Natural Draft Dry Cooling Towers Due to Flue Gas Injection

[1]*Ali Jahangiri* and [2]*Aliakbar Golneshan*

[1]Faculty of Mechanical Engineering, Semnan University, Semnan, Iran
[2]Faculty of Mechanical Engineering, Shiraz University, Shiraz, Iran

Abstract: In steam power plants, any improvement in the cooling towers thermal performance will cause the condenser's temperature to decrease; which in turn enhances the power plant's efficiency. Due to water shortage, many power plants all around the world rely on natural draft dry cooling towers (Heller type). In these cooling towers, condenser's exit hot water is pumped to vertical or horizontal finned radiators which are arranged all around of the tower. The tower should then be able to cool down the circulating water to the condenser's temperature. Conversely ambient air is sucked naturally through these radiators due to heat gain from the hot water. The heated air is then discharged through the tower's top exit to the ambient. Environmental conditions strictly affect Heller cooling tower's performance. Temperature, density and the mass flow rate of sucked air into the cooling tower will completely influence on its operation. To improve the cooling tower's performance it is proposed to inject the steam generator's flue gas into the tower for better air suction. In this study a natural draft dry cooling tower incorporating the flue gas duct is modeled to study the effect of flue gas injection on the sucked air flow rate and the radiator heat transfer. Considering the buoyancy effect term, for a 3-D incompressible flow, the navier stoke's equations as well as the energy equation are solved by computational fluid dynamic method (CFD) to obtain the air flow pattern in and around the tower as well as the heat transfer from the radiators. Results show that the flue gas injection will help to improve the performance of the cooling tower; however for this case (natural draft cooling) this efficiency enhancement is not significant. The maximum extra heat exchanged in the radiators due to flue gas injection is about 1.5 MW that can attain only a 0.07°C water temperature decrease. However, this amount is to be relied low to rely upon.

Key words: Natural Draft Dry Cooling Tower · Flue Gas Injection · Thermal Performance · Computational Fluid Dynamic

INTRODUCTION

The condenser's circulating water system is one of the most important parts of a power plant. Its task is to extract heat from the power cycle and release it to the environment, thereby ensuring improved efficiency of the power plant [1]. A natural draft dry cooling tower (NDDCT) is an energy-efficient and water-saving cooling equipment in power plants, widely used in the regions with lack of water. In these towers the air movement is dependent upon the difference in density between the entering cold air and the internal warm air. As the heat of hot water is transferred to the colder air passing through the radiators, this warm air then rises and draws fresh air in at the base of the tower. At natural condition, a uniform flow of the air is induced across the radiators installed vertically in the lower part of the tower, therefore the temperature decrease in the cooling water is uniform all around the tower. However environmental conditions strictly affect heller cooling tower's performance. in turn, the temperature, density and the mass flow rate of sucked air passing through the cooling tower will all influence on the tower's performance.

Some studies have been performed in this area. Baer *et al*. [2] investigated the thermal performance of a NDDCT under crosswind condition. They found that the thermal performance of the NDDCT is affected by the ambient air temperature as well as the crosswind velocity. Su *et al*. [3] studied the thermal performance of a dry-cooling tower under cross-wind conditions by

Corresponding Author: Ali Jahangiri, Faculty of Mechanical Engineering, Semnan University, Semnan, Iran.
Tel: +989122932632, E-mail: a_jahangiri@sun.semnan.ac.ir.

using computational fluid dynamics (CFD) method. Eldredge *et al*. [4] used numerical simulation to investigate some effects of flue gas injection on the wet cooling tower performance. The flue gas injection was found to have the most significant effect on tower performance (cold water temperature), because it strongly affects the buoyancy within the tower. Cooper *et al*. [5] presented an assessment of heat injection as a means of improving natural draft wet cooling tower performance. The enhancement of the airflow through the cooling tower resulted in more evaporation, causing the circulating water temperature to decrease.

In this study a natural draft dry cooling tower incorporating the flue gas duct is modeled. Considering the buoyancy effect term, for a 3-D and incompressible flow, the navier stoke's equations as well as the energy equation are solved by computational fluid dynamic method (CFD), to obtain the air flow pattern in and around the tower as well as heat transfer rate from the radiators.

Governing Equations: Assuming that adiabatic conditions exist for the tower's shell while neglecting the radiation effect, the heat exchanged between the hot water inside the heat exchanger and the cooling air crossing the NDDCT is expressed as:

$$q = h\left(T_{he} - T_{ai}\right) \tag{2.1}$$

Here q is the heat flux of the heat exchanger, h is the overall heat transfer coefficient, T_{he} is the mean temperature of hot water (arithmetic mean of the water entering and leaving the heat exchanger) and T_{ai} is the ambient air temperature.

The continuity, momentum and energy equations that describing mass, momentum and heat transfer can be written as follows [9, 10]:

$$\nabla . V = 0 \tag{2.2}$$

$$(V.\nabla)V = -\frac{1}{\rho}\nabla P + \nabla .\left(\frac{s}{\rho}\right) - \beta(T - T_a)g + S \tag{2.3}$$

$$\rho(V.\nabla)T = -\nabla .\left[(G + G_t)\nabla T\right] + Q \tag{2.4}$$

The turbulence equations describing turbulent kinetic energy (k) and turbulent dissipation rate (ε) are as follows:

$$(V.\nabla)k = \nabla .\left[(v + v_t / s_k)\nabla k\right] + P + G - e \tag{2.5}$$

$$(V.\nabla)e = \nabla .\left[(v + v_t / s_e)\nabla e\right] + c_{1e}\frac{e}{k}(P + G) - c_{2e}\frac{e^2}{k} \tag{2.6}$$

Here P is the generated kinetic energy of turbulence and G is the generated kinetic energy of buoyancy [10].

The thermal effectiveness of the cooling tower ε_{th} is defined as the ratio of the amount of ejected heat at any condition, to the maximum amount of ejected heat that occurs at no flue gas injection condition:

$$e_{th} = \frac{q_{conv}^{FGI}}{q_{conv}^{no\ FGI}} = \frac{(T_{wi} - T_{wo})_{FGI}}{(T_{wi} - T_{wo})_{no\ FGI}} \tag{2.7}$$

Here T_{wi} and T_{wo} are the temperatures at the inlet and outlet water, respectively.

Numerical Simulation: The NDDCT under investigation is *100m* high with a base diameter of *79m* and a radiator's height of *15m*. Boundary conditions specifying the flow and thermal variables on the boundaries of the physical domain are as shown on the schematic computational domain of the NDDCT (Figure 1). The nominal conditions of the simulated NDDTC are: $Q=214.3\ MW$, $T_{ambient\text{-}air}=15°C$, $P_{ambient}=85000\ Pa$ [8].

Pressure outlet boundary condition is used to define the static pressure of air at the domain outlets as well as other scalar variables in case of a back-flow. This is the case for the top and side boundaries of the domain Wall. No-slip condition is used at the solid boundaries such as ground and NDDCT's shell [7]. A lumped-parameter model for a heat exchange element is used as the radiator's boundary condition. Therefore, the radiator is considered infinitely thin, while the pressure drop through the radiator is assumed proportional to the dynamic head of air. Therefore, the pressure drop (Äp), varies with the normal component of velocity through the radiator (v), as follows:

$$\Delta P = 0.5\rho K_L v^2 \tag{3.1}$$

Here ρ is the air density and K_L is the non-dimensional loss coefficient. By specifying the definition of $K_L=f_1(v)$ the radiator pressure drop characteristics is defined, according to equation (3.2). On the other hand, by specifying the heat transfer coefficient $h=f_2(v)$ as well as the radiator temperature's T_{he}, according to equation (2.1), the radiator heat transfer characteristic is defined. At nominal conditions, T_{he} is kept constant at the design value of *313K* for all conditions. The air pressure drop through the radiators has the following relationship with the airflow rate [6]:

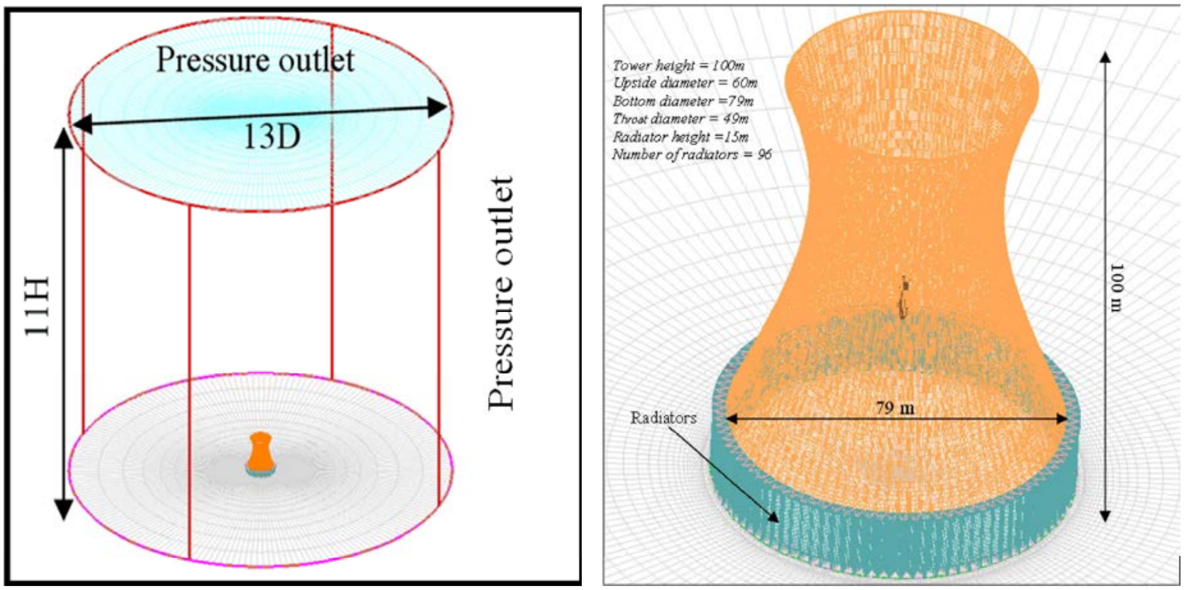


Fig. 1: Schematic computational domain of the NDDCT

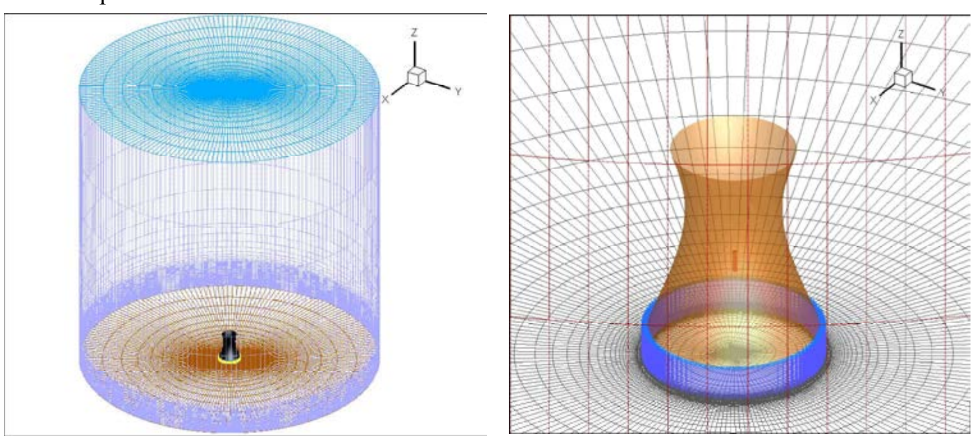


Fig. 2: Structured grids in use

$$\Delta P = 2.1\dot{m}^{1.76} + 0.06\dot{m}^{2} \tag{3.2}$$

Where, $\dot{m}$ is air mass flow rate through the radiators per unit frontal area [6].
The heat transfer coefficient h of the radiators is [6]:

$$h = 1374\dot{m}^{0.515} \tag{3.3}$$

It is assumed that air is a perfect gas and essentially dry with Prandtl number equal to *0.71*. Also air is allowed to travel perpendicular to the frontal area when it passes through the radiators.

Grid Generation: Structured grid scheme is used for grid generation (Figure 2). For grid study of the present simulation, four sets of grids were used. Considering the results with a maximum difference of one percent in the heat transfer rate, the case with *2874860* nodes was selected for the final simulation (Table 1).

Table 1: Grid study data

Case	Number of cells	Radiator heat transfer (MW)
1	2220220	213.45
2	2516929	213.68
3	2874860	213.81
4	3017660	213.86

Numerical Method: The finite-volume scheme was used for discretizing the flow field equations. A segregated solver and an implicit technique were used to solve the algebraic equations formed from the discretization of the closed set of equations (by expanding equations 2.2 to 2.6). SIMPLE algorithm was used for the calculation of the pressure and thus the velocity field, which were needed for the solution of the energy equation. For modeling of fully turbulent flow, standard k-ε model was used.

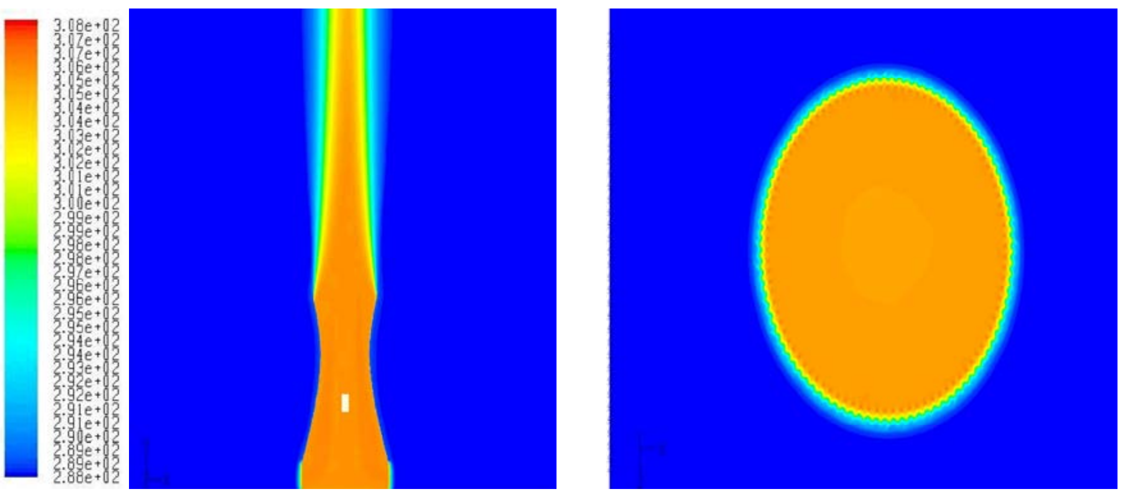
RESULTS

Model Validation: To verify the creditability of numerical results, the numerically predicted data are compared with measured values. Table 2 presents the numerically predicted total heat exchange rate and the water outlet temperature at the same operating conditions. The calculated results show about *0.23* percent discrepancy with the design values.

Table 2: Simulated results compared with the design and measured values at no injection condition

Case	T_{wi} (K)	T_{wo} (K)	Q (MW)
Design value	43.58	33.15	214.30
Num. Simulation	43.58	33.30	213.81
Measured value	43.58	33.70	202.80

Flue Gas Injection Velocity: There is no crosswind in natural convection operation; therefore the flow and temperature fields are axially symmetric (Figure 3). The distribution of temperature is uniform in the circumferential direction. This is due to uniform air distribution as it flows through the heat exchanger all around the tower, causing a uniform heat exchange to take place.



(a) (b)

Fig. 3: Temperature contours in no injection case (Ta=288.16 K), (a) z=0, (b) y=5

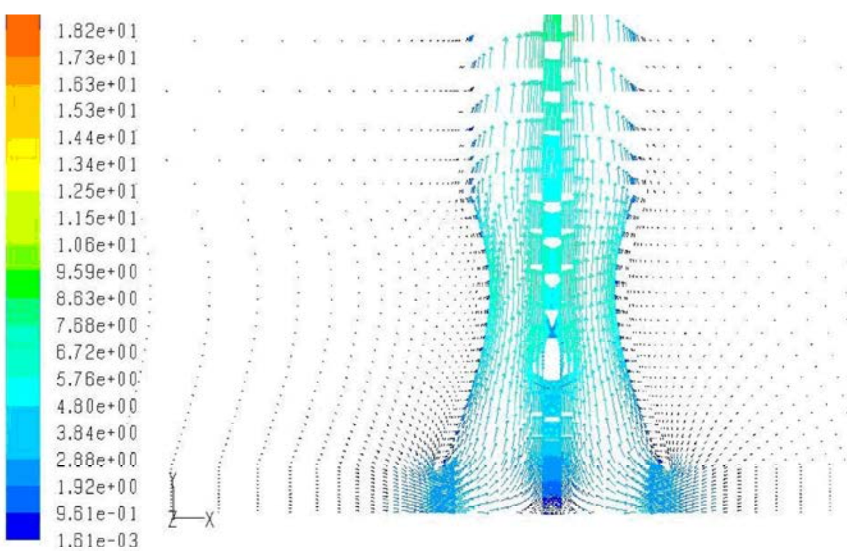


Fig. 4: Velocity vectors in no injection case (Ta=288.16 K), z=0

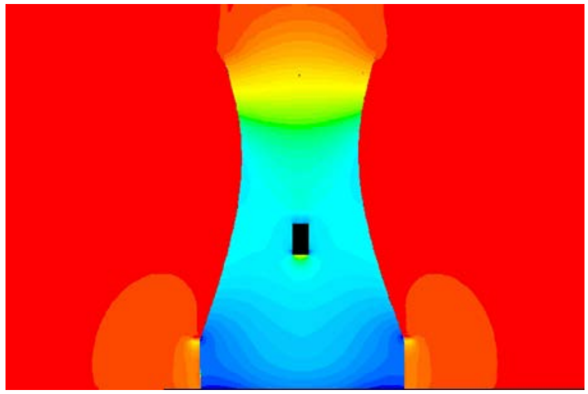


Fig. 5: Contours of pressure (natural draft with no injection)

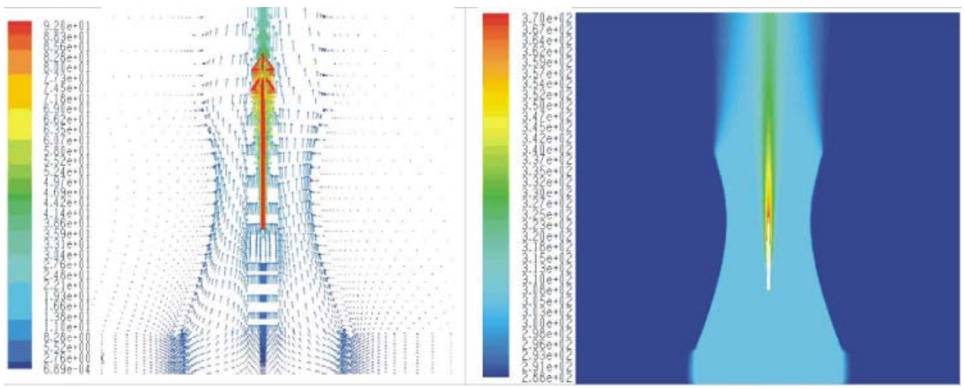


Fig. 6: Velocity vectors and contour of temperature in flue gas injection case

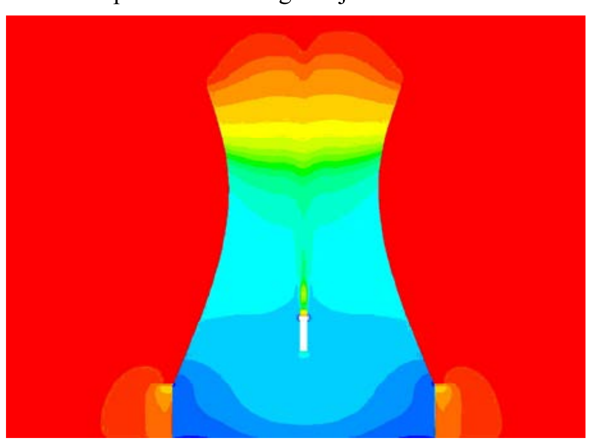


Fig. 7: Contours of pressure (natural draft with injection)

At different flue gas injection velocities corresponding to duct diameters of *2*, *4* and *6*m, the effect of injected flue gas on the thermal performance of the NDDCT is also investigated.

In all cases (NDDCT with and without flue gas injection) the flow, pressure and temperature fields are all shown to be axially symmetric (Figures 3 to 7).

Figure 8 shows the variations of mass flow rate of the sucked air into the tower with respect to the injection height for different flue gas duct diameters (gas velocities). The no-injection case is shown as reference. The best position occurs at a height of 35m for the least duct diameter (most flue gas velocity). Increasing the height from this value will cause the air mass flow rate to decrease sharply. However no gain will be achieved for higher duct diameters as shown in figure 9. This is due to the blockage effect of injected gas at low velocities. Similar variations are seen for the tower's thermal effectiveness (Figure 10).

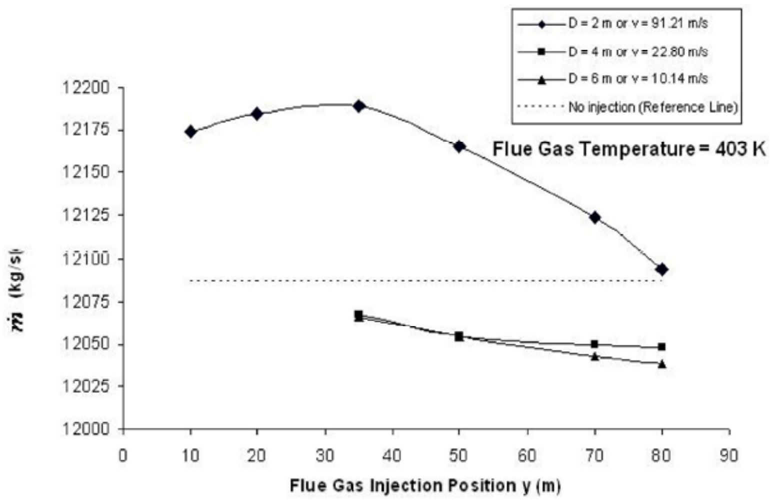


Fig. 8: Effect of variations in injection height on the inlet air mass flow rate for different injection velocities

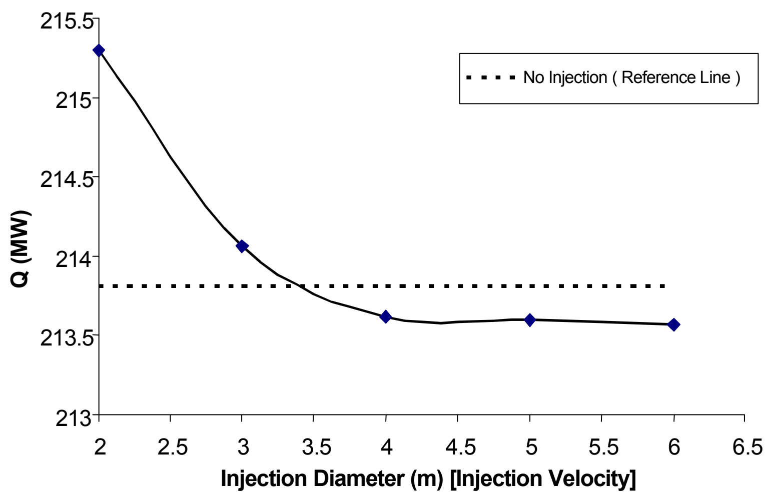


Fig. 9: Effect of variations on the flue gas injection diameter at y=35m

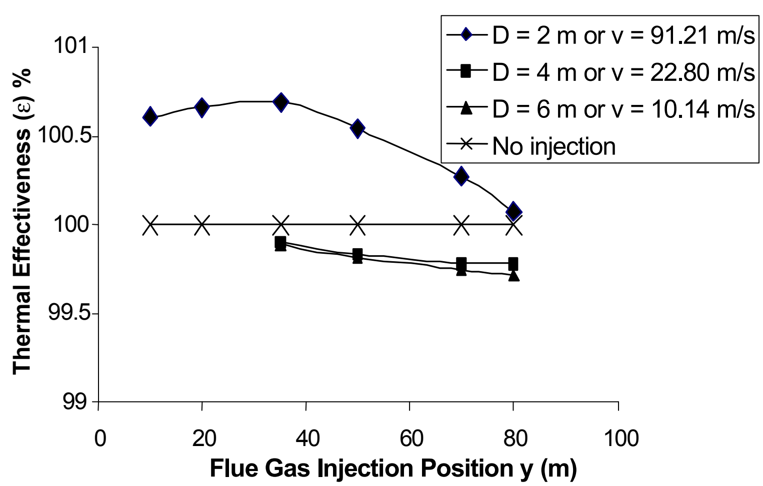


Fig. 10: Effect of variations in injection height on the thermal effectiveness for different injection velocities

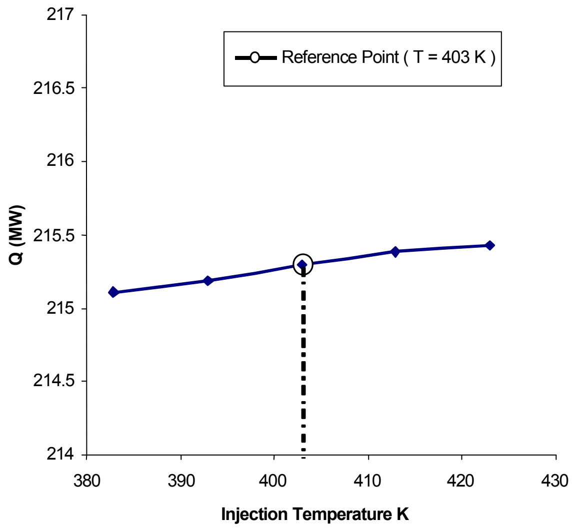


Fig. 11: Effect of variations in injection Temperature for D=2m & y=35m

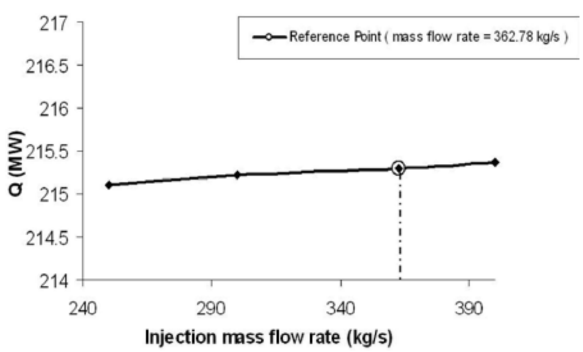


Fig. 12: Effect of variations in flue gas injection mass flow rate for D=2m & y=35

Also figures 8 and 10 show that at lower injection velocities (higher diameters) the total heat transfer and the thermal effectiveness (ε_{th}) are decreased, compared with no injection, for all injection heights. However increasing the injection velocity (D=2m) causes total heat transfer and the thermal effectiveness to be improved, with most improvement at lower injection heights.

Increasing the flue gas temperature will increase the rejected heat from the circulating water, however these variations are not significant for small variations in flue gas temperature (figure 11). The reference point shows the flue gas temperature for normal power station operation.

Figure 12 shows the amount of rejected heat versus injected mass flow rate. The reference point is representative of the mass flow rate corresponding to a single heat recovery steam generator flue gas. As shown no significant improvement is seen for a -20% to +10% variations in flue gas injection rate.

CONCLUSIONS

The most thermal effectiveness improvement (0.7%) corresponds to gas injection rate of *91.2 m/s* (D=2m) for a height of *35m*. The cooling water temperature decrease corresponding to this thermal improvement is *0.07°C* which may make it hard to economically prove the usefulness of flue gas injection when the tower is operating under natural draft condition. However at condition where crosswind exists, the flue gas injection may cause to reduce the tower's thermal inefficiency. This is aimed to work on by the authors in near future.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the office of gifted students of Semnan University for financial support.

Nomenclature

FGI	*Flue gas injection*
h	*Overall heat transfer coefficient*
K_L	*Non-dimensional loss coefficient*
$\dot{m}$	*Air mass flow rate through the radiator's per unit frontal area*
noFGI	*without flue gas injection*
q	*Heat flux of the heat exchanger*
T_{ai}	*Temperature of the air at the heat exchanger inlet*
T_{he}	*circulating water Mean temperature*
T_{wi}	*Temperature of the water at the inlet of the radiator*
T_{wo}	*Temperature of the water at the outlet of the radiator*
ΔP	*Pressure drop*

Greek symbols

ε_{th}	*Thermal effectiveness*
ρ	*Air density*

REFERENCES

1. El-Wakil, M.M., 1985. Power Plant Technology, *McGraw-Hill Book Company*, New York, St. Louis, San Francisco.
2. Baer, E., G. Ernst and D. Wurz, 1980. Effect of wind on the flow within the shell of a natural draught cooling tower, Vgb Kraftwerstechnik, 60: 11.
3. Su, M., Tang, G. and Fu, S, 1999. “Numerical simulation of fluid flow and thermal performance of a dry-cooling tower under cross wind condition”, *Journal of Wind Eng. and Ind. Aerodynamics*, Vol. 79, No. 3, pp.289-306.

4. Eldredge, T.V., D.J. Benton and J.W. Hodgson, 1997. An Investigation of The Effects of Flue Gas Injection on Natural Draft Cooling Tower Performance, J Engineering For Gas Turbine and Power, 119: 478-484.
5. Cooper, J., E. Grindle and R Lawson, 2002. Improving Natural Draft Cooling Tower Performance With Heat Injection, International Joint Power Generation Conference, Phoenix, AZ, USA.
6. Zhai, Z. and V. Fu, 2006. Improving Cooling Efficiency of Dry-Cooling Towers Under Crosswind Conditions by Using Wind-Break Methods, Applied Thermal Engineering, 26: 1008-1017.
7. Al-Waked, R. and Behnia, 2004. The Performance of Natural Draft Dry Cooling Tower Under Crosswind, CFD study, Int. J. Energy Re, Vol. 28, pp. 147-161.
8. Ghaffari, S., A.A. Golneshan and R. Mokhtarpoor, 2007. "3D Numerical Analysis of The Performance of An NDDCT with Windbreak Walls under Various Crosswind Velocities and Directions", *Conference on Applications and Design in Mechanical Engineering*, Kangar, Perlis, Malaysia.
9. Taeibi-Rahni, M., M. Ramezanzadeh. A. Davaran. Ganji. D.D.S. Soleimani. E. Ghasemi and H. Bararnia, 2010. Large-Eddy Simulations of Three Dimensional Turbulent Jet in a cross Flow Using A Dynamic Subgrid-Scale Eddy Viscosity Model with a Global Model Coefficient, World Appl. Sci. J., 9(10): 1191-1200.
10. Yousuf Ali, M., M. Masood and S.N. Mehdi, 2010. A CFD Combustion Analysis of a Hydrogen-Biodiesel Dual Fuel System, World Appl. Sci. J., 9(2): 144-150.