Alok singh, S P S Rajput / International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com
Vol. 2, Issue 1,Jan-Feb 2012, pp.1050-1056

# Application of CFD in Natural Draft Wet Cooling Tower Flow

**Alok singh [1] S P S Rajput [2]**

[1] Assistant professor [2] Associate professor, Department of Mechanical Engineering, Maulana Azad National Institute Of Technology, Bhopal (MP) India

**A 2D CFD model of a natural draft wet-cooling tower is developed. A commercial code Fluent has been used to simulate the transport phenomena inside the 130 m height, 98 m base and 68 m top diameter of tower. There is a large scale difference in the phenomena occurring in the tower. The geometrical scales of the environment are of the order of hundreds meters whilst the heat and mass in the fill zone is transferred within a few centimeters. The developed model is therefore supplied with a low dimensional representation of heat and mass transfer in the fill, whose purpose is to determine the heat and mass rejected from the cooling water to the air. The Euler multiphase model was used to simulate the flow, heat and mass transfer in the rain zone. The RNG k-ε model with the dispersed option is used for the turbulence modeling of multiphase flow. Due to rapid mixing and buckling of the plume the draft is reduced in the 2D model showing lower average velocities and higher cooled water temperature.**

Key wards:- cooling tower, CFD

## Introduction

Natural draft cooling towers are generally used in thermal power plants to remove heat absorbed in circulating cooling water. The performance of cooling tower can be affected by effect of crosswind condition. The knowledge of three dimensional flow structures can help us to design internal part of cooling tower. This is important especially in the case where flue gas goes directly to the cooling tower. Natural draft cooling tower can operate in different working regimes. Spreading of flue gas inside the cooling tower is very important from the construction service life point of view. Current design procedure is based on two dimensional analytical model for heat transfer coefficient and mass transfer coefficient. The work proposed two dimensional model based on the system of ordinary differential equations. Designed model allows to solve time evolution of the state in the natural draft cooling tower. CFD model developed in the work solves multi-phase steady state flow in two dimensional domain using commercial code" Fluent 12". Air flow is solved as continuous phase using Eulerian approach whereas droplet trajectories are solved as dispersed phase using the Lagrangian approach. The influence of crosswind condition to the thermal performance of natural draft wet cooling tower was also investigated in this work.

## Natural draft wet cooling tower

This study is concerned with natural draft wet cooling towers (NDWCT) in counter-flow configuration. These structures are most commonly found in power generation plants.
In a NDWCT in counter flow configuration.

There are three heat and mass transfer zones,

1. spray zone
2. fill zone
3. rain zone

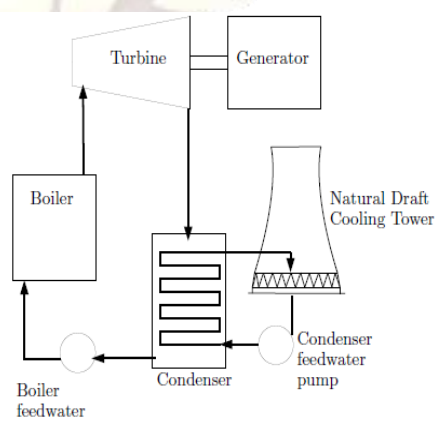



Figure 1. Power plant cycle with cooling tower

The water is introduced into the tower through spray nozzles Approximately 8-10m above the basin. The primary function of the spray zone is simply to distribute the water evenly across the tower. The water passes through a small spray zone as small fast moving droplets before entering the fill. There are a range of fill types. Generally they tend to be either a splash bar fill type or film fill type.The splash bar type acts to break up water flow into smaller droplets with splash bars or other means. A film fill is a more modern design which forces the water to flow in film over closely packed parallel plates. This significantly increases the surface area for heat and mass transfer.

## Modeling of cooling tower

Computation Fluid Dynamic Modeling: In order to analysis different conception of cooling tower behavior in wind first the computational fluid dynamic modeling of cooling tower is developed. The CFD (Anysis) code" FLUENT 12" is used for modeling. This package has been employed in this study to develop a two dimensional steady state simulation of NDWCT.

## Geometry

In the first step geometry is created in 2 D using reference data (4) providing different parts of cooling tower considering important details. The structure of whole model imagined in advance, because the possibilities in the subsequent steps depended on the composition of different geometrical shapes .Assumptions were made to take into account the main features of real construction.

- 2-D symmetry model is developed, fix the fill corresponding to real arrangement.
- Inlet and outlet space is created at bottom and top of the tower
- Cooling tower shell is considered as a wall with zero thickness and its profile is formed by curve by three point including throat.
- Assuming symmetrical thermal and flow field in the model, only one half of the cooling tower is modeled with a symmetry boundary condition.

| | |
|---|---|
| Tower height | 130 m |
| Air inlet height | 9 m |
| Fill depth | 1 m |
| Tower basin diameter | 98 m |
| Fill base diameter | 94 m |
| Tower top diameter | 68 m |
| Spray zone height | 12 m |
| Water flow rate | 15000 kg/s |
| Water inlet temperature | 318 K |
| Ambient air temperature | 298 K |
| Ambient air humidity | 55 % |
| Ambient pressure | 101 kPa |
| Inlet turbulence intensity | 1 % |

Design parameters for reference tower

## Mesh

After geometry mesh is generated. During mesh generation much attention to be paid with mesh quality requirement recommendation in FLUENT . In order to have an appropriate resolution of the flow field inside the cooling tower the computational domain is discretised into a large number of finite volume cells.

- Different parts is meshed with different element sizing .
- Fill zone must be fine meshed.
- By using mapped face meshing mesh the model with appropriate element sizing.
- After mesh generation create name of different parts of cooling tower.
- The inner and outer surface of the wall inside the model have identical shapes but are disconnected, so the mesh sizes on the two sides of the walls can be different.

## Cell zone condition

In cell zone surface body is considered as fluid. The operating pressure is 101325 Pa in upstream from the centerline of the cooling tower. The gravitational acceleration is 9.81 $m/s^2$. Operating temperature is 288.16 K and operating density is 1.22 $kg/m^3$ entered.

## Boundary conditions

Velocity inlet boundary condition is used to define the inlet velocity and other properties of air. Velocity magnitude of air takes normal to the boundary of inlet. Turbulence is taken as intensity and length scale. Thermal condition and species in mole fraction is defined.

Pressure out let is defined at out let of air .Other zone also define likewise.

## Governing equations

The governing equations for incompressible steady fluid flow can be written in general form as:

$$\nabla \cdot (\rho u \varphi - T\varphi \nabla \varphi) = S\varphi$$

where ρ is the air density (kg/m3), u is the fluid velocity (m/s), φ is the flow variable (u, v,w, k, ε, T, ω) and $T_\varphi$ is the diffusion coefficient for φ and $S_\varphi$ the source term. These equations can be expanded into the individual momentum and transport equations which, together with the continuity equation give the Navier-Stokes Equations. These equations can be solved numerically enabling fluid flow to be simulated forming the basis for CFD.
For all flows, FLUENT solves conservation equation for mass and momentum. For flows involving heat transfer and compressibility an additional equation energy conservation is solved. For flows involving species mixin ,a species conservation equation is solved.

The continuity equation for conservation of mass in Cartesian coordinates for transient flow can be given as,

$$\partial \rho / \partial t + \nabla \cdot (\rho \vec{v}) = S_m$$

where $S_m$ is the mass source term. The steady equation is obtained by simply neglecting the transient terms, ∂/ ∂t , from the left hand side.

The equation for conservation of momentum can be written as,

$$\partial / \partial t\, (\rho u_i) + \partial / \partial x_j\, (\rho u_i u_j) = - \partial p / \partial x_i + \partial / \partial x_j\, [\, \mu\, ( \partial u_i / \partial x_j + \partial u_j / \partial x_i)] + S$$

where S is now a source term for momentum. The source term for buoyancy can be written as,

$$S_b = (\rho - \rho_{ref})g$$

The transport equation for a scalar φ can be written as:

$$\partial/\partial t(\rho\varphi) + \partial/\partial x_j(\rho\varphi u_j) = \partial/\partial x_j\, [\rho T\, (\partial\varphi/\partial x_j)] + S_\varphi$$

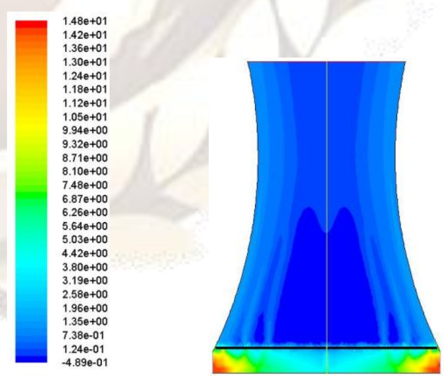
Navier-Stokes equations represent all the scales of fluid motion. Many flows in engineering are highly turbulent and so resolving all the scales explicitly using direct numerical simulation is too computationally intensive, requiring very fine discretisation of the above equations. Turbulence models are employed to reduce the computational work load by introducing simplifying assumptions and representing some of the scales of motion with additional equations.
The transport equations for the turbulence kinetic energy, k, and the rate of dissipation, ε, are given as

$$\partial/\partial t(\rho k) + \partial/\partial x_i\, (\rho k u_i) = \partial/\partial x_j\, [(\alpha_k \mu_{eff})\, \partial k/\partial x_j\, ] + G_k + G_b - \rho_\varepsilon + S_k$$

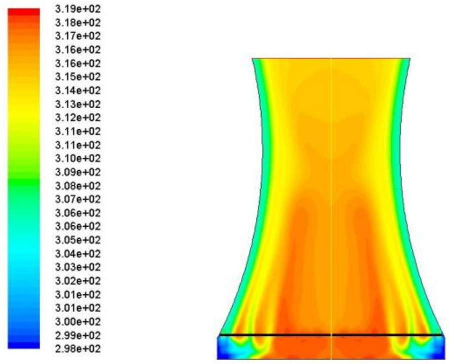
$$\partial/\partial t\, (\rho\varepsilon) + \partial/\partial x_i\, (\rho\varepsilon u_i) = \partial/\partial x_j\, [(\alpha_\varepsilon\, \mu_{eff})\partial_\varepsilon/\partial x_j] + C_{1\varepsilon}.\varepsilon/k\, (G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\, \varepsilon^2/k\ \ -R\varepsilon + S_\varepsilon$$

The air entering in to cooling tower suddenly changes temperature due to contact of hot water coming out from nozzle. Highest temperature zone is near the axis of cooling tower. As the air flows up the temperature goes down. Near wall of cooling tower the air temperature is comparatively low.
At inlet the pressure of air is maximum and gradually decreases as the air flows up. Above the spray zone and around the axis up to some extent the pressure having very less some, time it is negative also. Some distance from wall the value of pressure is like constant through- out the wall. Below the fill it is always higher than above the fill. Dynamic pressure is having higher value near the wall and lesser near the axis thorough-out the cooling tower height. Static pressure decreases up to fill and than decreases up to air outlet. Value of pressure coefficient is high at inlet and low at outlet.

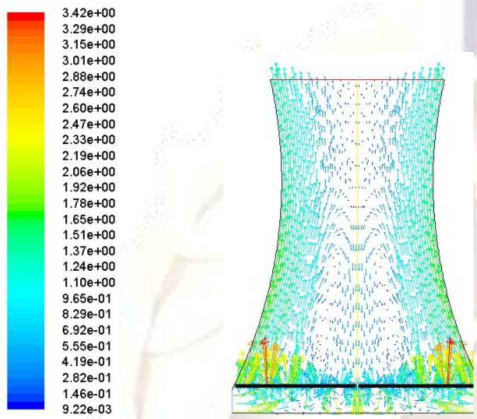
## Results and discussions



Contours of Total Pressure (pascal)



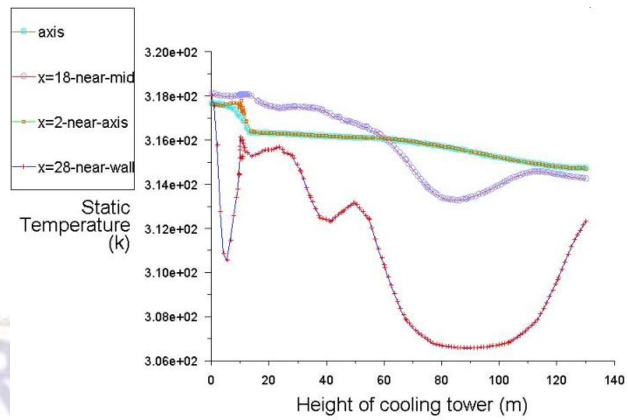
Contours of Static Temperature (k)



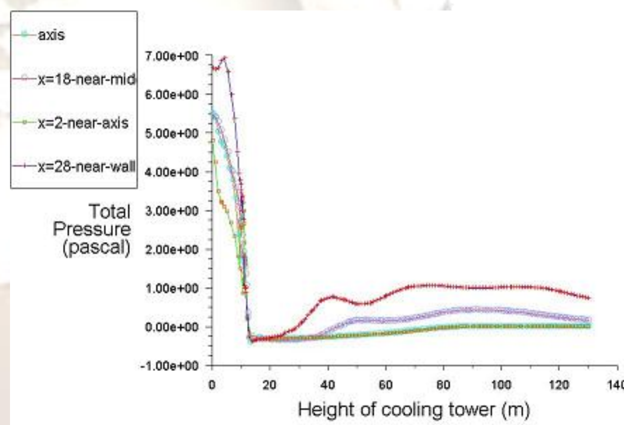
Velocity Vectors Colored By Velocity Magnitude (m/s)

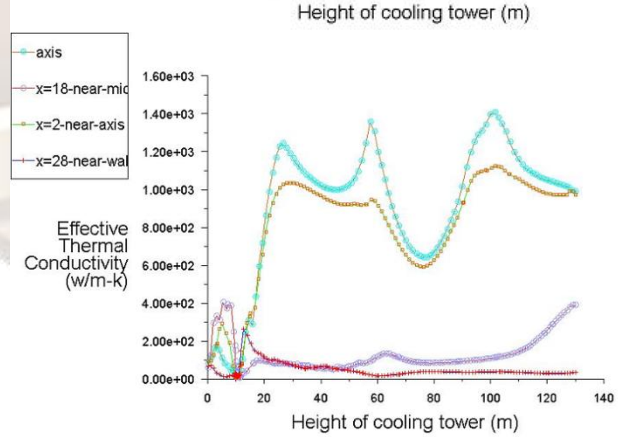
To find out the various properites of cooling tower there must be some line horizontzl with refrence to axis and ground level.

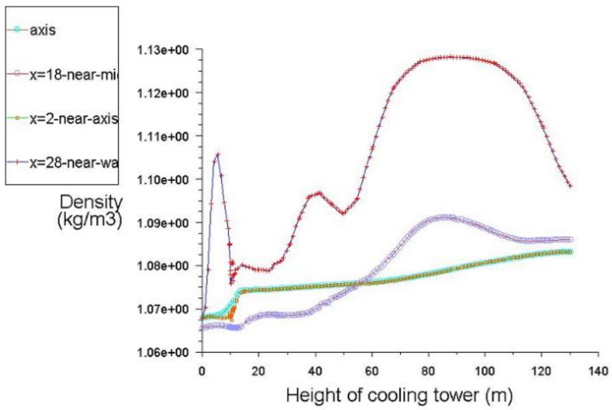
| S. NO. | Wind speed | Vertical line coordinate (Height of cooling tower) | | Line name |
|---|---|---|---|---|
| | | $(X_1, Y_1)$ | $(X_2, Y_2)$ | |
| 1 | 2 m/s | (2, 0 ) | (2, 130) | X=2, near axis |
| 2 | 2 m/s | (18, 0) | (18, 130) | X=18, at middle of axis and wall |
| 3 | 2 m/s | (28, 0) | (28, 130) | X=28, near wall |




To investigate thermal performance of tower with increase in height three line have been drawn as near axis ,near wall and between axis and wall. Temperature is having its high value in middle line and lower near the wall. The graph shows at very near to wall its value is very low around 307 k and out let axis having highest value of temperature.

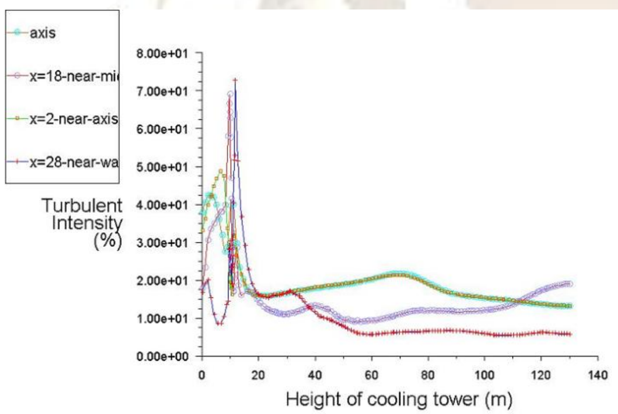


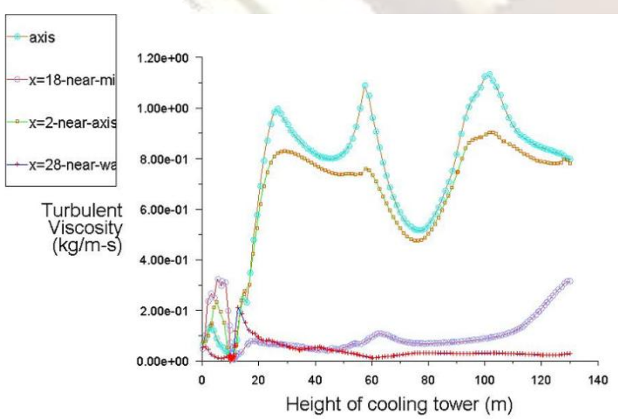



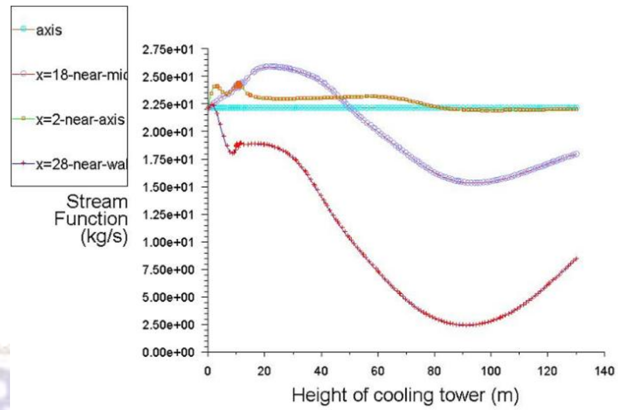


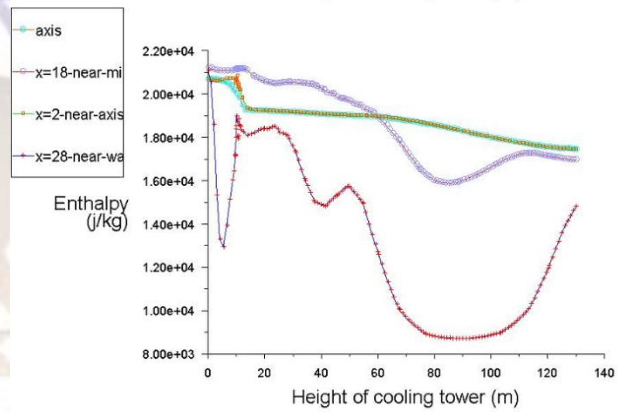

Total pressure suddenly falls to fill area at 12 m than slight increases according to height.

The thermal conductivity is very high at and near the axis because of high temperature and low density and very poor near wall. Density is high near wall and low near axis. Turbulence intensity changes very randomly up to the 12m that is fill zone and then takes its smooth values.




Turbulence viscosity is zero at fill zone and takes its high values suddenly for line axis, and nearly constant near wall. Stream function is linearly constant for axis and decreases according to height for middle line and line near wall. Enthalpy decreases after rain zone and its value is very low near wall area.

## Conclusion

By conducting 2 dimensional CFD simulation for investigating the thermal performance of wet cooling tower the overall changes take place in three zone. Every thermodynamics characteristics changes after rain zone either increases or decreases. Temperature is having its high value in middle line and lower near the wall. Pressure decreases to the value from 7 pa to zero at fill zone area than increase slightly according to height. Highest value of thermal conductivity is near axis. Density having reciprocal values of temperature. Enthalpy is low at near wall. Turbulent intensity increases up to rain zone than decreases, turbulent viscosity decreases to rain fill zone than increases. Stream function is linearly constant for axis and decreases according to height for middle line and line near wall. Enthalpy decreases after rain zone and its value is very low near wall area.

1. Al-Waked, R. and Behnia, M. The performance of natural draft dry cooling towers under cross-wind: CFD study. International Journal of Energy Research, 28:147–161, 2004.

2. J.R. Khan, B.A. Qureshi, S.M. Zubair, A comprehensive design and performance evaluation study of counter flow wet cooling towers, International Journal of Refrigeration 27 (2004) 914–923.

3. Al-Waked, R. and Behnia, M. The effect of windbreak walls effect on thermal performance of natural draft dry cooling towers. Heat Transfer Engineering, 26(8):50–62, 2005.

4. Kloppers, J.C. and Kroger, D.G. A critical investigation into the heat and mass transfer analysis of counterflow wet-cooling towers. International Journal of Heat and Mass Transfer, 48:765–777, 2005.

5. M. Lemouari, M. Boumaza, An experimental investigation of thermal characteristics of a mechanical draft wet cooling tower, in: Proceedings 13th IAHR, Poitiers, France, 2005.

6. Kloppers, J.C. and Kr¨oger, D.G. Refinement of the transfer coefficient correlation of wet cooling tower fills. Heat transfer engineering, 26:35–41, 2005.

7. Al-Waked, R. Development of Performance-Improving Structures for Power Station Cooling Towers. PhD thesis, University of New South Wales, Sydney, Australia, 2005.

8. Kloppers, J.C. and Kr¨oger, D.G. The Lewis factor and its influence on the performance prediction of wet-cooling towers. International Journal of Thermal Sciences, 44:879–884, 2005.

9. Kloppers, J.C. and Kr¨oger, D.G. Influence of temperature inversions on wet-cooling tower performance. Applied Thermal Engineering, 25:1325–1336, 2005.

10. Sarker, M. M. A., Kim, E. P., Moon, C. G. and Yoon, J. I.; "Thermal Performance Characteristics of Closed-Wet Cooling Tower", J. Korean Society for Power System Engineering, Vol. 9, No. 2 (2005), pp. 88-92.

11. Al-Waked, R. and Behnia, M. CFD simulation of wet cooling towers. Applied Thermal Engineering, 26(4):382–395, 2006.

12. Rossouw, A., Optimazation Of Natural Draught Cooling Tower Performance: Minimizing Towr Inlet Loss and INcreasing Effective Diameter, B.Eng final year project, Department of Mechanical Engineering, University Of Stellenbosch, Stellenbosch, South Africa, 2006.

13. Viljoen, D., Evaluation and Performance Prediction of Colling Tower Spray Zones, MSc.Eng Thesis, Department of Mechanical Engineering, Stellenbosch, University of Stellenbosch, South Africa, 2006.

14. Zhai, Z. and Fu, S. Improving cooling efficiency of dry-cooling towers under cross-wind conditions by using wind-break methods. Applied Thermal Engineering, 26:1008–1017, 2006.

15. Mokhtarzadeh-Dehghan, M.R., Konig, C.S. and Robins, A.G. A numerical study of single and two interacting turbulent plumes in atmospheric cross flow. Atmospheric Environment, 40:3909–3923, 2006.

16. Sirok, B., Blagojevic, B., Novak, M., Hochevar, M. and Jere, F. Improving the efficiency of natural draft cooling towers. Energy Conversion and Management, 47:1086–1100, 2006.

17. Qureshi, B.A. and Zubair, S.M. A complete model of wet cooling towers with fouling in fills. Applied Thermal Engineering, 26(16):1982–1989, 2006.

18. Pierce, D.J., Evaluation and Performance Prediction of Colling Tower Rain Zone, MSc.Ing Thesis, University of Stellenbosch, Stellenbosch, South Africa, 2007.

19. Qureshi, B.A. and Zubair, S.M., Second-law-based performance evaluation of cooling tower and evaporative heat exchangers, International Journal of Thermal Sciences, Vol. 46, pp. 188-198, 2007.

20. Forhad, G., Reza, H. and Shohreh, F.; "Experimental Study on the Performance of Mechanical Cooling Tower With Two Types of Film Packing", Energy Conversion and

Management, Vol. 48, No.1, pp.277-280 (2007).

21. 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.

22. A. Klimanek, R.A. Białecki, Numerical Model of a Natural Draught Wet-cooling Tower. Archives of Thermodynamics, 29 (2008), No. 4, pp. 63-72.

23. Williamson, N., Behnia, M. and Armfield, S. Comparison of a 2D axisymmetric CFD model of a natural draft wet cooling tower and a 1D model, International Journal of Heat and Mass Transfer 51 (2008) 2227-2236

24. Lucas, M., Martinez, P. J., and Viedma, A., (2009). Experimental Study on The Thermal Performance of A Mechanical Cooling Tower with Different Drift Eliminators. Energy Conversion and Management 50: 490-497.

25. A. Klimanek, R.A. Białecki, Solution of heat and mass transfer in counterflow wet-cooling tower fills. International Communications in Heat and Mass Transfer, 36 (2009), No. 6, pp. 547-553.

26. Cooper, J., E. Grindle and R Lawson, Improving Natural Draft Cooling Tower Performance With Heat Injection, International Joint Power Generation Conference, Phoenix, AZ, USA. 2010

27. 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.