Numerical simulation of a wet cooling tower

J. van der Merwe[1] C.G. du Toit[2]

(First received October 2001; Final version May 2002)

A numerical model to predict the flow field in and around a counter flow induced draft wet cooling tower was developed and tested. The numerical model consisted of the Thiart fan model, a Lagrangian droplet model and a packing model. The droplet model exposed the need to model the effect of the packing on the droplets and a simplified model was implemented to prolong the droplet retention time in the packing. The full cooling tower model predicted trends, at design conditions and with a wind present, which compared well with results reported in the literature.

Nomenclature

a	Coefficient for a pressure drop across a given packing [kg/m^4]
A_s	Surface area of droplet [m^2]
b	Coefficient for a pressure drop across a given packing [kg/m^{3}s]
c_p	Specific heat [J/kg K]
C_d	Drag coefficient
C_L	Lift coefficient
F_m	Mass transfer rate per unit surface area [kg/s m^2]
$\overline{F}$	Force [N]
h_{fg}	Latent heat of phase change [J/kg]
K_g	Mass transfer coefficient [m/s]
m	Mass [kg]
p	Pressure [Pa]
q''	Surface heat transfer rate per unit surface area [W/m^2]
$\overline{Q}$	Volume flow rate [m^3/s]
$\overline{Q}_r$	Torque [N m]
r	Radius [m]
$S1$	Explicit momentum source term from linearised form [N/m^3]
$S2$	Implicit momentum source term from linearised form [Ns/m^4]
t	Thickness [m]
T	Temperature [°C]
$\overline{T}_r$	Thrust [N]
$\overline{u}$	Instantaneous velocity [m/s]

Greek

α	Fluid fraction
β	Angle between relative velocity and plane of rotation [rad]
ρ	Density [kg/m^3]
σ	Solidity of rotor
Δ	Axial thickness of fan rotor [m]

Subscripts

am	Virtual mass
b	General body
d	Droplet
dr	Drag
fr	Frontal
j	$j = 1, 2, 3$
p	Pressure
r	Relative
s	Vapour at droplet surface
t	Air
v	Partial
∞	Vapour in droplet surroundings

Introduction

In highly competitive markets like the chemical, petrochemical and power industries the production plants are constantly being optimized and new industrial plants designed to be more economical and to deliver products at competitive prices to the market. A great deal of research is devoted to the production process in such plants and unfortunately the support systems, e.g. the water-cooling systems, are often neglected. Therefore, a great need exists in this specific study field for good and reliable information.

The process heat of industrial applications is removed by continuously recirculating the process water through cooling towers: either wet or dry cooling towers or a combination of both are found in the industry. In a wet cooling tower, the hot process water is in direct contact with the cooler air, whilst in a dry cooling tower the water and air are separated. A wet cooling tower is more effective than a dry tower, because it relies mainly on the wet-bulb temperature of the air and not on the generally higher dry-bulb temperature, as in the case of a dry-cooling tower.

The induced draft counter flow cooling tower that was simulated falls into the wet cooling tower category. The heat transfer in this kind of tower is achieved partly by evaporation of some of the recirculating water and partly by the transfer of sensible heat. As seen in Fig. 1 the induced draft counter flow cooling tower consists basically of the inlet region, rain zone, splash packs or fill, nozzles and spray zone, drift eliminators, the fan and then the

1. School of Mechanical and Materials Engineering, Potchefstroom University for CHE, Private Bag X6001, Potchefstroom, 2520 South Africa

2. School of Mechanical and Materials Engineering, Potchefstroom University for CHE

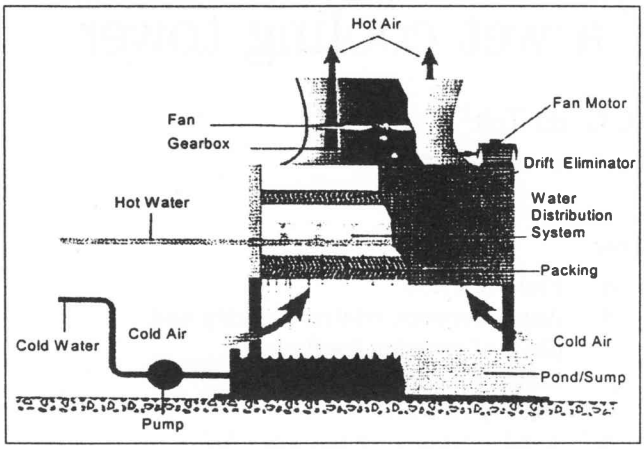


Fig. 1. Schematic representation of a counter flow induced draft wet cooling tower (courtesy of GEA)

outlet region where the plume (hot moist air exiting the tower) is formed.

Previously either an analytical or an experimental approach was used to solve engineering problems. With the arrival of high-speed digital computers and development of more reliable numerical algorithms, a third approach for solving problems has become available, in the form of numerical modelling.

In this paper, we aim to help address the need referred to above by contributing to a better understanding of the influence of atmospheric conditions on the performance of cooling towers. The main objective is the development of a numerical model to predict the flow field in and around a counter flow induced draft wet cooling tower. A typical counter flow wet cooling tower is shown in Fig. 1. Both windless conditions and winds perpendicular to the cooling tower were investigated. A better understanding of the performance and physics of a cooling tower can lead to a more effective design of the cooling tower. This can prove to be advantageous for the entire plant.

In the next section the relevant literature is reviewed. This is followed by a description of the numerical model and the applicable governing equations. The numerical model consists of the multiphase flow of the water and air that is modelled by a Euler-Lagrange method, the fan modelled by the Thiart model, and the packing or fill modelled by a porous medium model. In the Results section the numerical results are presented: first for the preliminary validation tests of the individual models and then for the full cooling tower model for windless and windy conditions. The paper ends with some remarks.

Literature review

There is a collection of one-, two- and three-dimensional numerical and mathematical models of cooling towers and their sub-components in the literature, e.g. Hoffmann & Kröger,[1] Webb & Villacres,[2] De Villiers & Kröger,[3] Weinacht & Buchlin,[4] Majumdar *et al.*,[5] Becker *et al.*,[6] Benocci *et al.*,[7] Dreyer *et al.*,[8] and Van Staden[9] to name but a few. The classification of two- and even three-dimensional models may be misleading since most of them only model the motion of the droplets as one-dimensional. Most of these models also model the droplets as spheres and there is still no consensus on a representative diameter used to model the droplets, due to its dependence on the packing. Rosendahl & Condra[10] addressed the perfect spherical homogeneous particles with a more general shape namely the super-ellipsoid, but at the date of publication they were still in the process of verifying their results.

The packing is generally modelled as a moderate porous medium[9] where the packing is represented as a momentum sink in the full Navier-Stokes equation. Not much literature is available on the effect of the packing on the droplets if the detail of the packing is not modelled. It seems that most of the mathematical models handle the heat and mass transfer with transfer coefficients being calculated for each zone,[3] which implicitly take the effect of the packing on the droplets into account.

Basically two fan models are discussed in the literature. The model of Van Staden[9] is suited for the one-dimensional modelling of many fans in one computational grid. This lacks the multi-dimensional effect of a fan blade on the air. The Thiart[11] model entails a full three-dimensional implicit modelling of the fan and will be discussed in more detail later. Other methods do exist, such as the direct modelling and the implicit method. Both these methods require extremely fine grids and the actual blade profile. This can result in a large number of computational cells and may require enormous computational resources, unwanted in a simulation where not only the fan, but also other components needs to be modelled.

Du Toit & Van Wyk[12] summarized the three basic methods for the numerical solution of two-phase flows. The first method is the Euler method or pseudo one-phase method and only requires solving concentration or transport equations. There is no relative movement between the air and the droplets, which results in no momentum transfer. The second method is the Euler-Euler method. Here both air and droplets are considered to be continuums that are intermingled. The size of the droplets can only be modelled by a single droplet size and as in the first case the interaction between the droplets and droplets and the droplets and walls cannot be modelled explicitly. The first two methods are more suited to modelling fine "mists" or clouds of particles rather than relatively large, heavy particles. The third method is the Euler-Lagrange method in which the air is considered to be a continuum and the water droplets discrete particles. The gas phase (air) is still modelled by the Euler method and the particles, droplets, are modelled using the Lagrange method. This implies that a droplet is followed on its path through the flow field. The transfer of mass, momentum and energy is modelled by using the appropriate sink and source terms in the airflow. In this model the change in droplet size and the interaction between the droplets and droplets can be modelled. The third method is the most accurate but also the most com-

plex and expensive, in terms of computing time, compared to the previous two methods.

NUMERICAL MODELLING

The separate models were chosen to be capable of handling three-dimensional flow with reasonable accuracy and also not to be too expensive with regard to computational time and storage (memory) required.

The commercial CFD software code, STAR-CD, was used for solving the Navier-Stokes equations, using a finite volume discretisation method. A structured co-located grid was used. The convection terms are discretised using the hybrid upwind differencing scheme. Equations for the values of each dependent variable in each grid cell are obtained in a linearised form by the integration of the differential equations over the cells with suitable interpolation assumptions. The equations are then solved implicitly in an iterative manner. In this paper the SIMPLE pressure correction algorithm, as incorporated in STAR-CD, was used. The standard $k-\varepsilon$ model was used to model the turbulent nature of the flow.

Multiphase flow (Euler-Lagrangian model)

The droplets are considered to be the dispersed phase and the air the continuum phase in the Euler-Lagrangian model. The total population of the droplet sizes and properties are represented by a finite number of computational parcels or samples. The number of samples is not arbitrary and must be large enough to accurately represent the whole population. One hundred samples were found to be sufficient for this specific simulation of a tower that has a surface area normal to the flow direction of the air of 192.4 m^2.

In the continuum phase a few modifications were necessary to account for the displacement of the air by the droplets. The fluid fraction α, which is the fractional volume occupied by the air, was multiplied by the density and the effective viscosity on the left side and the volume integrated sources on the right side of the Navier-Stokes conservation equations.

For the dispersed phase (droplets) the conservation equations from the STAR-CD methodology manual[13] in a Lagrangian framework are as follows:

The momentum equation for a droplet of mass m_d is given as:

$$m_d \frac{d\overline{u}_d}{dt} = \overline{F}_{dr} + \overline{F}_p + \overline{F}_{am} + \overline{F}_b \qquad (1)$$

where $\frac{d\overline{u}_d}{dt}$ is the change of the instantaneous droplet velocity. $\overline{F}_{dr}$ is the drag force, $\overline{F}_p$ is the pressure force, $\overline{F}_{am}$ is the 'virtual mass' force or the force required to accelerate the carrier fluid entrained by the droplet and $\overline{F}_b$ is a general body force term consisting of the gravitational and Coriolis forces. The contribution of the pressure force and the 'virtual mass' force were found to be negligible compared to the drag and gravity forces.

The droplet mass rate of change can be defined as follows:

$$\frac{dm_d}{dt} = -A_s F_m \qquad (2)$$

where A_s is the surface area of the droplet and F_m is the mass transfer rate per unit surface area, which Bird *et al.*[14] defined as:

$$F_m = K_g p_t \ln \frac{(p_t - p_{v\infty})}{(p_t - p_{vs})} \qquad (3)$$

where K_g is the mass transfer coefficient, p_t is the pressure of the air, $p_{v\infty}$ is the partial pressure of the vapour in the droplet surroundings and p_{vs} is the partial pressure of vapour at the droplet surface. The pressure, p_{vs}, is assumed to be equal to the saturation pressure at the droplet temperature T_d. The saturation vapour pressure can be calculated from correlations available in Kroger.[15]

The energy balance of the droplet takes the surface heat transfer rate and the loss or gain due to phase changes into account.

$$m_d \frac{d(c_{pd} T_d)}{dt} = -A_s q''_d + h_{fg} \frac{dm_d}{dt} \qquad (4)$$

where c_{pd} . is the specific heat of the droplet, h_{fg} is the latent heat of phase change and q'' is the surface heat transfer rate per unit surface area of the droplet.

There is still great uncertainty regarding the effect of the droplets on the turbulence of the continuous phase and it is therefore not taken into account. A droplet in turbulent flow experiences a randomly varying velocity field to which it responds according to its inertia. This process is determined with a stochastic approach.[13] The coefficients for drag, heat and mass transfer are applicable to single component liquid droplets in a gaseous carrier. The drag, heat transfer and mass transfer coefficients were taken from the STAR-CD manual,[13] El Wakil *et al.*[16] and the Ranz *et al.*[17] correlation, respectively.

Thiart fan model

The model of Thiart[11] is a three-dimensional model and is capable of simulating the fan without requiring a large number of cells. The blade element theory usually used for aircraft and ship propeller design calculations is used to model the body forces induced by the fan blades. The body forces, which appear in the Navier-Stokes equations, must have the units of Newton per unit volume. The final expressions for the thrust and torque body forces, respectively, are:[11]

$$\frac{\partial \overline{T}_r}{\partial V} = \frac{1}{2}\rho \overline{u}_r^2 \frac{\sigma}{\Delta}\left(C_L \cos(\beta) - C_D \sin(\beta)\right) \qquad (5)$$

$$\frac{\partial \overline{Q}_r}{\partial V} = \frac{1}{2}\rho \overline{u}_r^2 \frac{\sigma}{\Delta} r\left(C_L \sin(\beta) - C_D \cos(\beta)\right) \qquad (6)$$

If it is assumed that the direction and magnitude of the relative velocity $\overline{u}_r$ is known, it only remains to specify the lift and drag coefficients to complete the model. These coefficients normally depend on the geometry of the blade

element, the angle of attack, and the blade Reynolds number. The four parameters necessary for the empirical correlations are the blade angle, the chord, camber ratio, and the thickness ratio. The empirical correlations of Hoerner[18] and Hoerner & Borst[19] were used, as did Thiart.[11]

Porous media flow

The packing usually consists of some complex geometry such as grids that are suspended in the air by connecting rods. To model this section of the cooling tower in detail would require a large number of computing cells and large computational resources. For this reason the packing was modelled as a moderate porous medium. A moderate porous medium basically predicts a pressure drop across a certain distance for a specific volume flow. For the purposes of this discussion, it is worth recalling the full Navier-Stokes momentum equation. The momentum source is linearised in the following way:

$$\text{source in } x_j - \text{direction} = S1u_j - S2u_j * u_j \qquad (7)$$

$S1u_j$ is the explicit term that defines the momentum source as described in the multiphase flow section and $S2u_j$ is the implicit term that defines the momentum sink, which was used to simulate the packing. Van Staden[9] stated that the pressure loss across the fan could be converted to a force per unit volume. The same principle could be implemented for the packing. Since the volume flow rate Q, frontal area of packing A_{fr}, pressure drop p across and the thickness t of the packing are known, the momentum sink is defined as follows:

$$S2u_j = \frac{p}{t} = a\left(\frac{Q}{A_{fr}}\right)^2 + b\left(\frac{Q}{A_{fr}}\right) \qquad (8)$$

where a and b are the coefficients for a pressure drop across a given packing. The limitations with the modelling of the packing as a momentum sink are that it does not account for the turbulence effects caused by the packing and it does not account for the effects of the flow entering and leaving the packing. The effect of the packing on the droplets is also not accounted for. Introducing the porosity of the packing can account for the effect of the packing on the flow entering and leaving the packing. Introducing an extra body force, in the momentum calculations discussed in the multiphase section, on the droplets in the packing area can solve the effect of the packing on the droplets.[20] This body force basically prolongs the droplet retention time in the packing by slowing the droplet down with an increase in "packing resistance" on the droplet. This extra body force was determined with numerical experimentation to obtain the same retention time and outlet water temperature specified by the manufacturer under design conditions for the specific tower simulated. The implementation of the Thiart fan, packing and droplet sub-models in the CFD code STAR-CD and the results and interpretation thereof are discussed next.

Results

The fan model in conjunction with the porous medium was implemented first to validate these two models. Seven operating points on and around the design point on the fan characteristic curve were tested with the numerical model. The resistance of the porous medium was adjusted to induce the different operating points.

The numerical results compared well with the design conditions. The maximum difference between the design and predicted numerical pressure and volume flow rate was less than -8% and -2.4%, respectively. This was the worst-case scenario where the fan had to work at conditions of more than 15 Pa (10.91%) and 40 m^3/s (8.37%) off the design conditions. At design conditions the difference between the design and predicted numerical pressure and volume flow rate was found to be -2.35% and -1.20%, respectively. This is excellent considering the limited data available on the fan profile and the assumptions that the chord length, maximum thickness and chamber of the blade element are constant.

The droplet model was then implemented in conjunction with the porous medium to investigate the functioning of the droplet model. The droplet model was already part of the standard setup of STAR-CD, but the partial pressure of the vapour and the droplet momentum transfer had to be specified in the user coding. No quantitative validation was attempted, since there were not enough data available on the size and distribution of the droplets in the cooling tower. From the results obtained, the specified cooling range of the cooling tower could be obtained with either a droplet diameter of 2 mm or higher packing resistance, to prolong the retention time of the droplets in the packing, or with a combination of both. The droplets in this section were not modelled as polydispersed droplets with different diameters. This option was not used in this study but it is expected to have a significant effect on the contact surface area.

After the component sub-models were tested and validated to satisfaction, they were put together in a full computational test grid. The mesh consisted of approximately 240 000 mesh cells and the mesh was refined in the region of the cooling tower. Symmetry plane boundary conditions were used to reduce the number of cells required. The model was first tested under windless conditions and compared with the design conditions and results available in the literature.

The predicted numerical outlet temperature of the water was found to be 27.5°C and the design water outlet temperature was 27°C. This was with a droplet diameter of 3.5 mm, which was the same diameter used in a previous study.[3] The results obtained at design conditions were compared qualitatively with the results of Majumdar *et al.*,[5] as can be seen in Fig. 2 to Fig. 5. Fig. 2 and Fig. 3 show the similarities of the pressure contours between the present results, Fig. 2, and those of Majumdar *et al.*,[5] (Fig. 3). Pressure line 1 of the VERA2-D results does not show the influence of the hub since the hub was not modelled.

The results depicted in Fig. 4 and Fig. 5 show that the

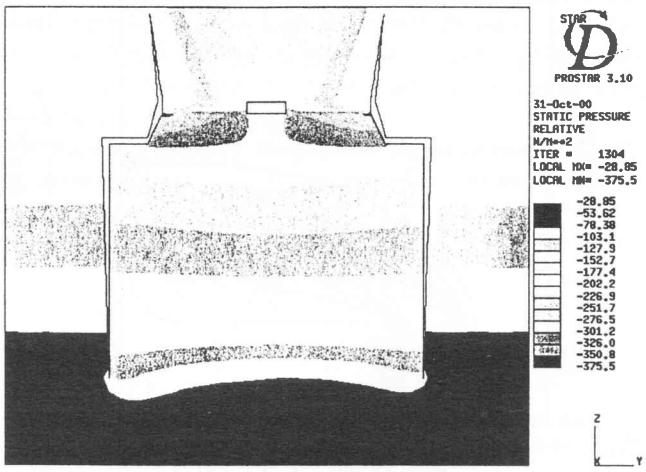


Fig. 2. Numerical relative pressure field results of the cooling tower model

highest vapour concentration appears in the side region of the tower. The higher vapour concentrations on the sides may be due to the air that moves slower in these regions and thus has longer contact time with the droplets. The recirculation in the corners may also contribute to this phenomenon. It could also be seen that the vapour concentration changed more rapidly in the packing zone compared to the rain and spray zones. This can be attributed to the prolonged contact time between the droplets and the air, as well as the smaller droplets present in the packing resulting in a larger contact area. From Fig. 2 and Fig. 3, it is also clear that the pressure drop across the packing is the most significant and that the rain and spray zones produce a relatively small pressure drop compared to the packing.

Subsequently, the effect of the wind on the model was tested and the trends were validated. A power law wind speed profile was used at the inlet boundary. The wind speed was measured at a height of 11.4 m above ground level. Becker *et al.*[6] investigated the recirculation at different wind speeds and stated that the recirculation is controlled by two opposing mechanisms. The first mechanism tends to increase recirculation as the wind speed increases. This is applicable when the wind speed is zero or almost zero. The plume then rises almost vertically into the air. In this case the recirculation is almost zero. If the wind speed increases, the plume is bent over and entrapped in the wake behind the tower. The tower then draws some of the hot plume in. The opposing second mechanism comes into play at higher wind speeds. As the wind speed increases the plume is increasingly diluted right on top and just behind the tower. This decreases the concentration of the hot moist plume before it is entrapped in the wake behind the tower. This mechanism thus tends to decrease the recirculation as the wind speed increases.

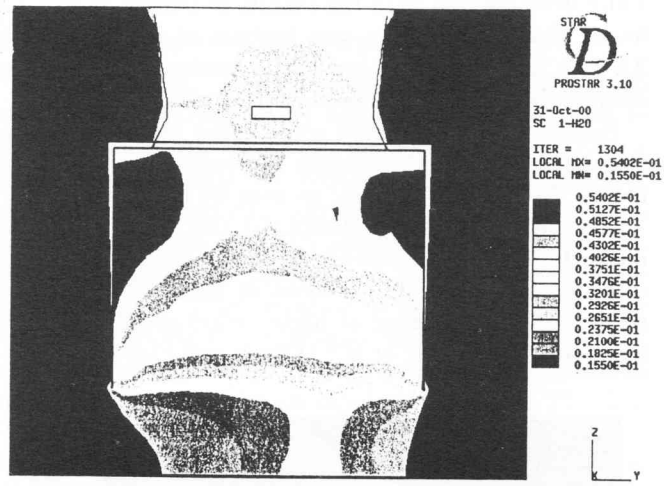


Fig. 4. Numerical vapour fraction results of the cooling tower model

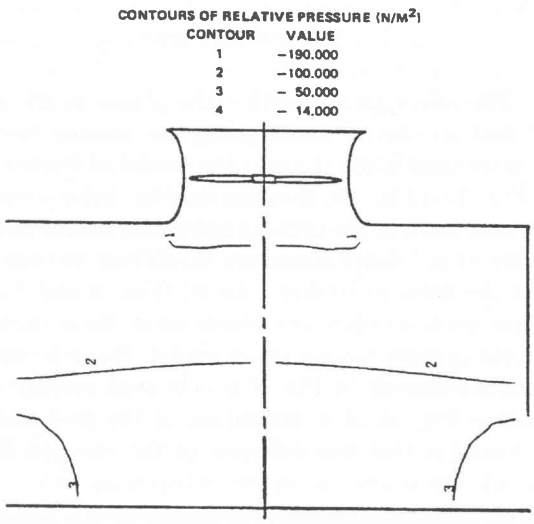


Fig. 3. Numerical relative pressure field results of Majumdar *et al.*[5]

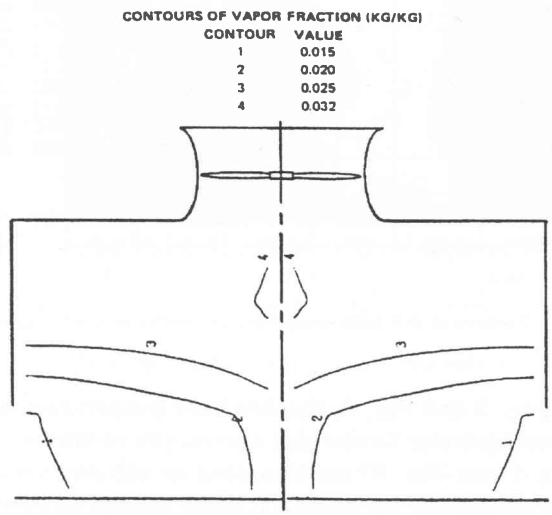


Fig. 5. Numerical vapour fraction results of Majumdar *et al.*[5]

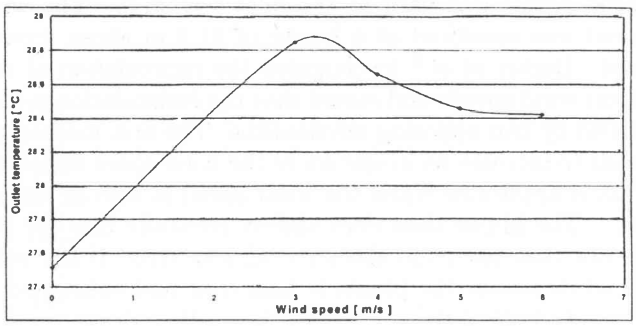


Fig. 6. Temperature curve showing the influence of the inlet velocity on the water outlet temperature

These opposing mechanisms may be the reason why the predicted outlet water temperature increased from where there was no wind up to approximately 3 m/s and then decreased as the wind speed increased (Fig. 6). This result correlates well with the results of Becker *et al.*[6] They found that recirculation was highest at a wind speed of 3.6 m/s and their field test data resulted in a maximum recirculation at a wind speed of 3.24 m/s.

Becker *et al.*[6] predicted some other results and found that as the air speed increases the temperature plume becomes thinner and shorter, as seen by comparison of Fig. 7 and Fig. 10. They attributed this to the wind that convects the hot air downwind faster than the turbulent diffusion can cause it to spread or mix. The plume also seems cooler on average at higher wind speeds, which was also found by Becker *et al.*[6] (Fig. 8 and Fig. 10).

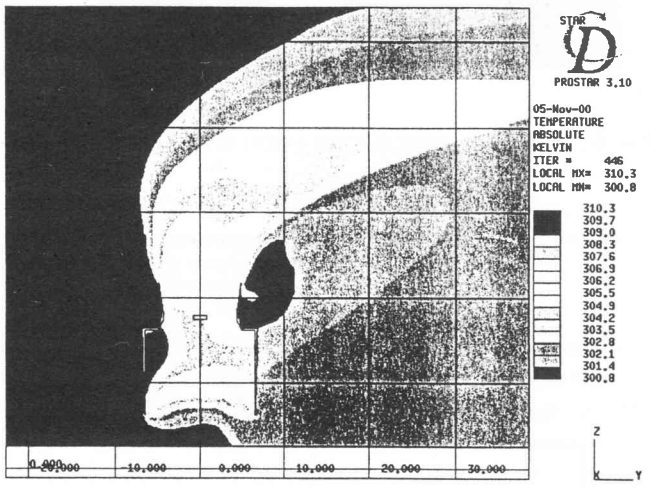


Fig. 7. Numerical dry bulb temperature results at a wind speed of 4 m/s

In Fig. 7 and Fig. 9, the dry bulb temperature field of the cooling tower model and the results of Becker *et al.*[6] in Fig. 8 and Fig. 10 are compared at different velocities. When comparing the results, it must be kept in mind that the height of the cooling tower used in this study was 21 m, whilst the tower used by Becker *et al.*[6] was approximately

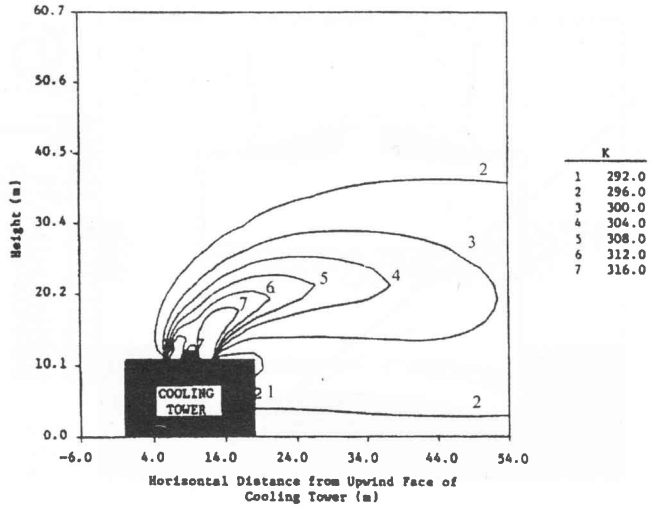


Fig. 8. Numerical dry bulb temperature results predicted by Becker *et al.*[6] at a wind speed of 4 m/s

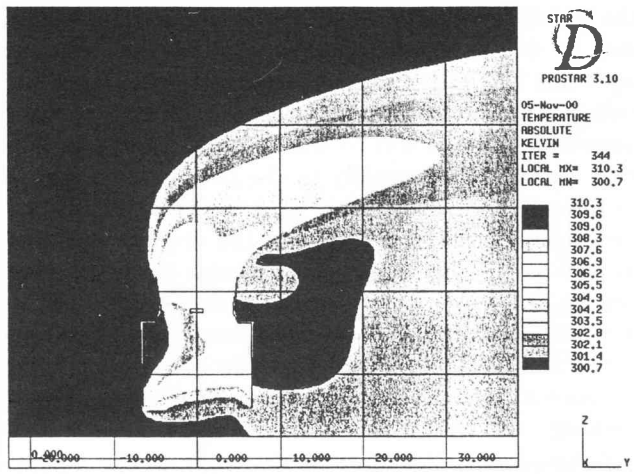


Fig. 9. Numerical dry bulb temperature results at a wind speed of 6 m/s

11 m. The effect therefore that the plume in the present model had on the air surrounding the cooling tower was much more significant than in the model of Becker *et al.*[6] From Fig. 7 to Fig. 10, the recirculation behind the tower can be seen both in the present numerical results and those of Becker *et al.*[6] Since there was no diffuser present in the path of the wind in Becker's model (Fig. 8 and Fig. 10), the wind tends to blow the plume away more than is the case in the present cooling tower model. It can be seen that the recirculating air in Fig. 7 is hotter on average than is the case in Fig. 9. The advantage of the present cooling tower model is that the influence of the wind on the flow field inside the tower can be investigated as well.

Fig. 11 shows the simulation results of the influence of the wind speed on the performance of the fan. The wind definitely decreases the volume flow of air through the fan.

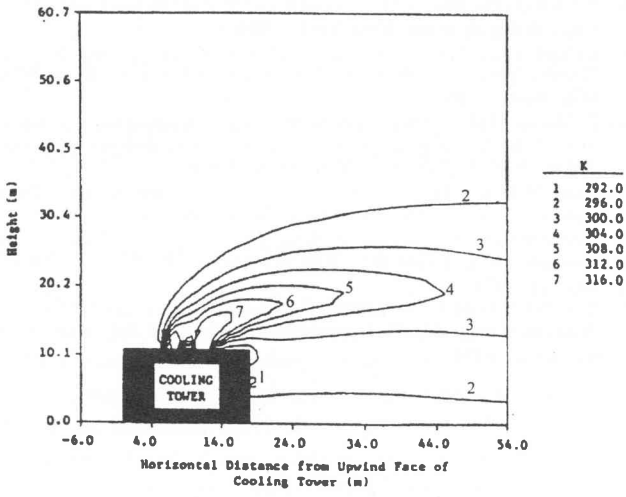


Fig. 10. Numerical dry bulb temperature results predicted by Becker *et al.*[6] at a wind speed of 6 m/s

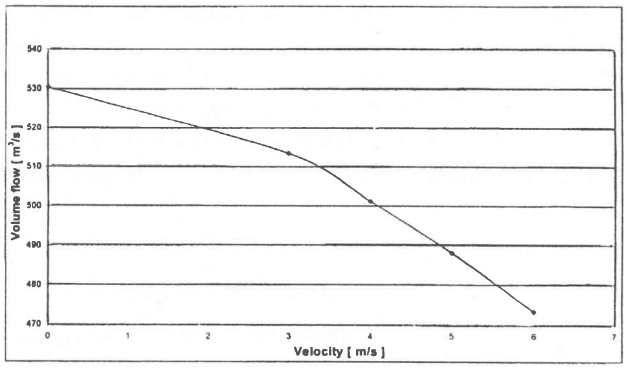


Fig. 11. The influence of wind on the volume flow produced by the fan

This is, however, not much as a wind speed of 6 m/s perpendicular to the tower only reduces the volume flow by approximately 10%. Less airflow through the tower to absorb the heat from the process water would seem to increase the outlet water temperature, but the opposite is seen in Fig. 6. The outlet water temperature only increases for wind speeds up to about 3 m/s and then starts to decrease. It can therefore be assumed that what is probably seen is the effect of recirculation and flow combined, with recirculation the dominant effect. According to Becker *et al.*[6] two opposing mechanisms control recirculation. At low wind speeds the hot moist plume is drawn in and the volume flow is decreased and therefore the expected increase in outlet water temperature. But when the second mechanism, (wind diluting the concentration of the plume) starts to play a larger role it seems that the reduced volume flow through the tower does not have the expected result on the outlet water temperature.

Conclusions

In this day and age of high-speed digital computers, the use of numerical modelling, to solve engineering problems, is becoming a more common occurrence. In this paper a numerical model to predict the flow field in and around an induced draft wet cooling tower was developed. The trends, at design conditions and with a wind present, predicted from the full cooling tower model compared well with results available in the literature. The full cooling tower model incorporates all the aspects concerning the cooling tower, which had previously been mostly modelled separately .

Even though the results in this study are only qualitative, they reveal a graphical insight into the physics involved in cooling towers and emphasize the power of computational fluid dynamics. A detailed verification of the model with experimental results is recommended to determine a more certain quantitative nature of the model. Preliminary verification proved to be satisfactory.

For optimal design and use of cooling towers in industry, performance of the cooling system and the influence of environmental conditions on these systems should be well understood. The influence of environmental conditions (wind, buildings, etc.) on cooling towers cannot be investigated theoretically due to the complex interaction between the systems and the uniqueness of each cooling system. This study makes an important contribution to addressing this problem by helping to develop better methods of understanding the aerodynamic behaviour of cooling towers. This paper also showed that commercially available codes could be used to simulate complex flow in and around a wet cooling tower effectively. This leads to cheaper and more effective research methods and improved design methods, which will speed up the development of the study field.

References

[1] Hoffmann JE & Kröger DG. Analysis of heat, mass and momentum transfer in the rain zone of a natural draft counter flow cooling tower. *Proceedings of the Ninth International Heat Transfer Conference*, Jerusalem, Israel, 1990.

[2] Webb RL & Villacres A. Performance simulation of evaporative heat exchangers (cooling towers, fluid coolers and condensers). *Heat transfer engineering,* **6**, 31–38, 1985.

[3] De Villiers E & Kröger DG. Analysis of heat, mass and momentum transfer in the rain zone of counter flow cooling tower. IJPGC. *Proceedings of the 1997 IJPGC*, Denver, November, **2**, 141–149, 1997.

[4] Weinacht P & Buchlin JM. A numerical model for gas- droplet flow application to liquid spray and cooling towers. *Technical Note 144: Von Karman Institute for fluid dynamics*. 1982.

[5] Majumdar AK, Singhal AK, Reilly HE & Bartz JA. VERA2D: Program for 2-D analysis of flow, heat and mass transfer in evaporative cooling towers — Vol. 1. California: *EPRI CS-2923*, 1983.

[6] Becker BR, Steward Jr WE, Walter TM & Becker CS. A numerical model of cooling tower plume recirculation. *Mathematical computational modelling*, **12**, 799–819,1989.

[7] Benocci C, Buchlin JM & Weinacht P. Prediction of the air droplet interaction in the inlet section of a natural draft cooling tower. *Proceedings of the Fifth IAHR Cooling tower workshop*, Monterey, California, 1986.

[8] Dreyer AA, Oosthuizen PC & Van Staden MP. The effect of wind on the performance of counterflow natural draught evaporative cooling towers. *Eleventh IAHR cooling tower symposium*, Cottbus, Germany, 1998.

[9] Van Staden MP. An integrated approach to transient simulation of large air cooled condensers using computational fluid dynamics. Johannesburg: Rand Afrikaans University, PhD Thesis, 2000.

[10] Rosendahl L & Condra T. A new basis for the formulation of the dispersed phase in gas-particle systems. *Nineteenth ICTAM*, Kyoto, Japan, 1996.

[11] Thiart GD. A Numerical procedure for predicting the effects of distorted inflow conditions on the Performance of axial flow fans. Stellenbosch: University of Stellenbosch, PhD Thesis, 1990.

[12] Du Toit CG & Van Wyk JS. Verbrandingsmodellering met druppel model. *BMI report 89/217*, University of Stellenbosch, 1990.

[13] STAR-CD version 3.10 Methodology Manual. Dispersed multiphase flow (Lagrangian model). *Computational Dynamics Limited*: 12.1–12.28, 1999.

[14] Bird RB, Stewart EW & Lightfoot EN. *Transport phenomena*. John Wiley & Sons, New York, 1966.

[15] Kröger DG. *Air-cooled heat exchangers and cooling towers — thermal-flow performance, evaluation and design*. Begel House, New York, 1998.

[16] El Wakil MM, Ueyhara 0A & Myers FS. A theoretical investigation of the heating-up period of injected fuel droplets vaporizing in air. *NACA Technical Note 3179*, 1954.

[17] Ranz WE & Marshall WR. Evaporation from drops - Parts I and II. *Chemical Engineering progress*, **48**, 1952.

[18] Hoerner SF. Fluid-dynamic drag. *Hoerner fluid dynamics*, 1965.

[19] Hoerner SF & Borst HV. Fluid-dynamic lift. *Hoerner fluid dynamics*, 1975.

[20] Van der Merwe J. Numerical simulation of an evaporative cooler. Potchefstroom: Potchefstroom University for CHE, Masters' dissertation, 2000.