Plume simulation of natural draught cooling towers

Karsten GRASEMANN, Falk MOHASSEB, Norbert WIENERS,
and Gabriel REYMANN

GEA Energietechnik GmbH, Herne, Germany

Keywords: Plume simulation, CFD, Boundary-Setup, Weather Effects, Atmospheric physics.

ABSTRACT: This paper deals with the setup of numeric calculations to estimate the plume of natural draught cooling towers. The incorporation of several physical effects and boundary conditions and possible ways of measuring or estimating them will be introduced also. Finally, a best practice guide and a checklist are offered to ease up future simulations for customers and their contractors.

1 PLUME SIMULATIONS

1.1 *Overview*

Plume simulations are more and more popular today. Plume simulations shall result in the area/volume of a produced plume to estimate its effect on sibling towers (recirculation), air traffic, farms or habitats. Very often, these results are necessary for governmental clearing processes.

The problems of missing, or very complicated models for describing these kinds of calculation seems to be solved by the technology of numerical simulations on computers (CFD). Computer hardware nowadays is relative cheap compared to the beginning ages of electronic calculators. Even specialized software for numerical simulations evolved and is available today for free of charge (e.g. OpenFOAM). Nevertheless, modern computers and modern software packages are only as good as the data they will be fed with. A flaw setup of a numerical simulation or a too simplified mathematical model will lead into very unreliable data. Another problem in providing “good” data to a numerical software system is the effective measurement of real world data.

A possible setup, the physical effects and their models within such kind of simulations and some ideas to measure these quantities will be introduced within this paper.

1.2 *Definitions of terms*

A numerical simulation of a plume produced by a natural draught cooling tower has to cover a huge area/volume around the cooling tower. This volume is named the “Domain” of the simulation and is very often of cubical shape. Other shapes are possible, but may lead into problems within the solving process. Within this domain, the emitting cooling tower resides. On the faces of the domain and the cooling tower, startup values of several physical quantities have to be defined. These values are called “Boundary Conditions”. Different faces of the domain and the tower can be grouped together to define a specific boundary condition onto them. These groups of faces are called “Patches”. The arrangement of possible patches and their names are shown in Fig. 1 on this page.

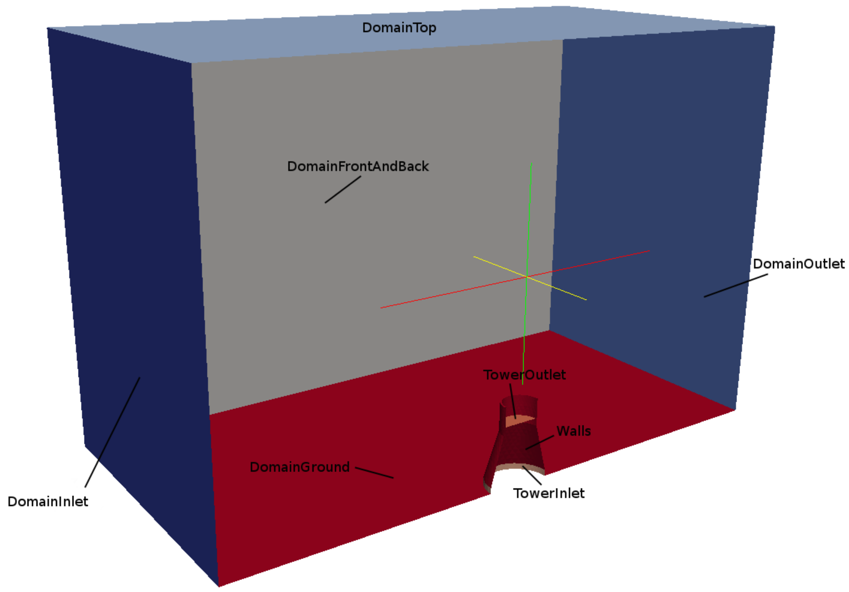

Figure 1 – definition of a simulation domain

The inside of the domain has to be discretized into several finite volumes. The borders of these discretization leads to several points and edges in space. They build up the “Grid” of a simulation. The topology and the geometry of the grid highly influence the results of a calculation, because results are averaged between corner or center points of the grid. Thus, a grid space with a corner length of e.g. 5 m cannot reflect turbulences with sizes of 10 cm.
On the other hand, a very fine grid (a grid with billions of points) will lead into a very long lasting calculation, needs lots of computer memory and will rise the costs of a simulation. A good balance between accuracy and effort has to be found by the CFD specialist, regarding the needs of the customer.

1.3 *Usual provided design data*

It is sufficient for the thermodynamical design of a cooling tower to define “Design data”. A set of design values regarding the ambient air conditions consists of:

- Ambient air velocity (e.g. 3 m/s),
- Ambient relative air humidity (e.g. 71 %),
- Ambient air temperature (e.g. 21 °C) and
- Ambient air pressure (1013.5 mbar)

These values are measured at one specific point in a specific height (e.g. 10 m). Unfortunately, these values are insufficient for a plume calculation

2 PHYSICAL EFFECTS AND BOUNDARY CONDITIONS

As mentioned before, the standard set of design data is insufficient for a reliable plume simulation. The atmosphere consists of different layers with changing physical quantities (like temperature, density and pressure) within height. Plume calculations cover a large volume with height up to 800 m and therefore, functions depending on the height for all ambient quantities have to be defined. Unfortunately, earth's atmosphere is a highly dynamical system and within physical correctness, such functions should be time dependent too. But such formulations will lead into very complex models and need data that can only be measured with a very high amount of costs. Fortunately, several reliable formulations describing the "standard atmosphere" like ISO 2533:1975, the International Civil Aviation Organization (ICAO) Atmosphere and the U.S. standard atmosphere exist and should be incorporated into the simulation.

2.1 *Ambient air temperature*

The atmosphere consists of a non-constant temperature coefficient cf. Eq. (1). It is given by ICAO within the standard atmosphere up to 11 km from sea level to

$$k_{adiabatic} = -0.0065\ {}^{K}/_{m} \tag{1}$$

Therefore, the resulting formula for the temperature is

$$T(h) = k_{adiabatic} \cdot h + T_{design} \tag{2}$$

It should be kept in mind that other, more complex definitions are possible and must be incorporated into the simulation regarding the given situation.

2.2 *Ambient air pressure*

The pressure drop within the standard atmosphere can be described according to ICAO as

$$p(h) = 101325\,(1\ -\ 2.25577^{-5}\,h)^{5.25588} \tag{3}$$

Where p(h) is in [Pa] and h is given in [m].

2.3 *Ambient air velocity*

Because of the dynamical nature of the atmosphere, air velocity assumptions are one of the most difficult predictions. Air movements (wind) can vary in strength and direction over time and height very rapidly. Also some standard atmosphere models try to give formulations for this phenomena, the incorporated equations are quite complex and did not reflect local conditions very well. A simple model for the air velocity can be given by the equation of a plate boundary flow

$$v_{air}(h) = v_\infty \times \left(\frac{h}{\delta}\right)^{1/7} \tag{4}$$

With δ as the height where v_∞ occurs. The calculation of v_∞ can be done either by predicting the cloud layer height or giving a secondary velocity at a different height that the design data.

2.3.1 Predicting v_∞ via the cloud layer height

The cloud layer height is the altitude from where the ambient air lost its capability of saving steam. By rising up an air particle, it will cool down until no more humidity can be kept within and the air has a relative humidity of 100 %. This temperature is named the "dewpoint". The difference between a given temperature and the dewpoint is called "spread". With a rule of thumb used in avionics (valid for spreads < 10 °C), the spread can be estimated by

$$100 - spread * 5 = \varphi_{rel} \tag{5}$$

With the calculated spread, the cloud layer can be estimated by a rule of thumb used in avionics too.

$$h_{cloud} = spread \cdot 122 \tag{6}$$

Where h_{cloud} is given in [m] and spread is given in [°C].
Assuming the velocity v_∞ is reached at the cloud layer, v_∞ is given by

$$v_\infty = \frac{v_{air_{design}}}{\left(\frac{h_{design}}{h_{cloud}}\right)^{1/7}} \tag{7}$$

2.3.2 Predicting v_∞ via a secondary velocity

By knowing a second velocity, a system of two equations for the two unknowns v_∞ and δ can be solved. This approach should be favored instead of the "dewpoint" approach.

2.4 *Ambient air humidity*

The ambient absolute humidity is assumed constant throughout the atmosphere at the domain boundaries, because no sources or sinks of steam occurs here.

2.5 *Buoyancy effects*

To reflect the effect of different types of ground materials and its heating onto the ambient air, the surrounding areas of a cooling tower should be defined by proper surface temperatures. Therefore, a correct temperature distribution map shall be available to be modeled into the calculation. Especially when large surfaces of water lie beneath the power plant, the atmospheric equilibration processes have an intense effect on plume behavior.

3 THE SOLVER

The CFD-Solver for a plume distribution must be capable of calculating humid air up into the super-saturated-state. A good approach to model humid air is the description of humidity as an additional scalar. Depending on the underlying software system, some alterations to the solver have to be done. By describing the humidity as a scalar, formulations for non-constant heat capacities and thermal conductivities of humid and/or super-saturated air and their effects on pressure, velocity and density has to be modeled with additional equations and programmed into the solver. Furthermore, the solver has to incorporate the equations of the Buoyancy effects too.

By the dynamic nature of earth's atmosphere, its time varying turbulence fields resulting in the bumpy looking plume, a steady-state-solver is of improper use. The solver must be of transient type to give good results.

To increase the stability of the solver, a pre-calculation of the potential field of the ambient flow as a basis for the main calculation is advisable.

4 BOUNDARY SETUP

It is very important to incorporate the temperature and pressure gradients within the internal field of the domain. The *DomainInlet* shall incorporate the velocity gradient. A special focus must lie on the *TowerInlet* patch. This patch is an outlet for the domain. If this patch is modeled with a constant pressure value, the calculation will result in false values. The *TowerInlet* must be modeled with an averaged mass-flux condition to reflect circular varying pressure and velocity gradients of the ambient air flowing into the tower.

The *DomainOutlet* must be wave transmissive to allow a correct plume flow out of the domain. It shall contain the pressure gradient too.

The formulation of wall functions within the selected turbulence model may lead to some problems. Usually wall functions rely on a relative fine boundary layer (<0.5 cm). This will lead within the large dimensions of the simulation domain to a very fine mesh near the walls of the cooling tower and the *DomainGround*.

5 CONCLUSIONS

The numerical simulation of a plume produced by a natural draught cooling tower is a very challenging task. Several additional physical effects in comparison to a normal wind study with time and space varying quantities influences the results. Some numerical pre-calculations have to be done to stabilize the solver. The provision of sufficient ambient input data leads to a bigger effort than determine the thermal design data on the contractor side. The following points shall ease up this process as far as possible.

A problem not solved yet is the finesse of the grid around the plume. A very fine grid downwind the cooling tower will lead to a long calculation time and high memory consumption on the computer. An automatic mesh refinement algorithm could be helpful within this problem, but it is not yet programmed into the solver used by GEA Energietechnik GmbH. This work is still in progress.

5.1 Providing ambient data

A good rule of thumb is "The more, the better". Provide as much data as possible to the CFD personnel. A small ToDo-List for the contractor might look like this:

- Define a goal of the simulation
  Try to form a specific question to the numerical calculation like “Is the plume visible after 2 km downwind the tower under the given conditions?”, “Interferes the plume the marked flight path of the nearby airport?” or “Is the plume in conflict with the following governmental building restrictions?”
  The clear formulation of a question is one of the most helpful tools within simulations. A reasonable selection of solvers and possible extensions rely on these questions.

- Define an atmospheric model
  Select a standard atmospheric model described in chapter 2 or define a customized one for all atmospheric quantities.

- Weather conditions and maps
  Define a standard weather condition and measure quantities at a day under these conditions. Provide meteorological charts and weather maps from that day or from a longer time period to the CFD personnel. Provide maps that cover the surrounding of the cooling tower within a minimum area of 5 km. These maps should cover topology and orographic information. Estimations on turbulence fields and adoptions to the standard atmospheric models can be derived from that data.

- Measure air velocities, temperatures and pressure
  Measure these quantities synchronous in a minimum of three different heights (e.g. 1.5 m, 5 m and 10 m) continually within 10 minutes. Mark the measurement point within the provided maps.

- Define ground surface areas and measure their temperatures
  Mark special ground surface types (wood, grass, field, concrete or water) on the maps and measure their temperatures under the predefined weather conditions to incorporate Buoyancy effects and atmospheric equilibrations.

5.2 Checking the calculation results

After a successful simulation run, the contractor shall answer the following questions with the data provided by the CFD personnel.

- Are the simulated results equal the measured values at the measurement points?
- Does the simulation maintain the atmospheric model (velocity, temperature and pressure gradient) in upwind direction of the cooling tower?
- Is the mass continuity equation fulfilled with the averaged boundary data?
- Does the solution (global errors) converge relative smoothly?
- Is the turbulence field time dependent?
- Is the domain covering all boundary definitions?
- Are turbulence model specific values (like Y+ on a RANS solver) within their maximum limits?
- Looks the velocity field even in the areas of strong Buoyancy effects explainable and sensual?

If all questions can be answered with a “Yes”, the simulation results seem to be reliable within the approximations and assumptions. If one or more questions must be answered with “No”, the results are highly discussable.