Optimization of Air Conditioning Diffusers Location in Large Agricultural Warehouses Using CFD Techniques

D. Valero Huerta and R. García-Bartual
Instituto de Ingeniería del Agua y del Medio Ambiente (IIAMA)
Universidad Politécnica de Valencia. Camino de Vera s/n 46022 Valencia, Spain.

Keywords: numeric simulation, turbulence model, climate control, air conditioning, FLOW-3D

Abstract
The implementation of air conditioning devices for climate control in large agricultural warehouses is required to keep adequate atmospheres for several plant species and industrial or human activities. Factors as temperature, air velocity or humidity are fundamental on plant growing or product conservation. Diffusers geometry and their placement strongly affect temperature and velocity resulting fields. In the case study presented, a metallic structure warehouse in the south of Spain is considered, assuming that very warm outside conditions (temperature of 38°C) are reached outside. An evaporative cooling system of low energy consumption is used to cool the warehouse, reducing the extreme temperature outside to allow human activity inside the warehouse in acceptable conditions. The cooling system uses only a fan and a small water pump, generating humid air breeze across wet pads.

Diffusers separation is investigated for an optimal cooling of the warehouse using CFD techniques. The complexity involved due to convective and diffusive terms, buoyancy forces and nonlinearities of the flow governing equations, make it necessary in practice to develop individual numerical simulation for each diffuser geometry, climate scenario, boundary conditions and objective climate control conditions.

INTRODUCTION
Air conditioning in a large warehouse is particularly important under very warm climatic conditions, as those occurring in many parts of Spain during summer season. When comfort conditions required are not excessively demanding in large warehouses, efficient low consumption cooling systems are required. On the other hand, CFD techniques can be helpful in simulating the resulting temperature and velocity fields, allowing performance evaluation of different locations and spatial distributions of the diffusers. This can yield to more efficient cooled air flows inside the warehouse, resulting in improved overall conditions with equivalent energy consumption.

Numerical Flow Analysis
The flow equations solved numerically are the URANS (Unsteady Reynolds Averaged Navier-Stokes) with variable density: (Blazek, 2005)

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \mathbf{u}_i)}{\partial x_i} &= 0 \\
\frac{\partial \mathbf{u}_i}{\partial t} + \frac{\partial (\mathbf{u}_i \mathbf{u}_j)}{\partial x_j} &= -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 \mathbf{u}_i}{\partial x_j \partial x_j} - \frac{\partial \mathbf{u}_j \mathbf{u}_j}{\partial x_j}
\end{align*}
\]

(Eq. 1)

(Eq. 2)
Where \( \bar{p}(x,y,z,t) \) is the mean pressure, \( \bar{u}_i(x,y,z,t) \) the mean velocity in \( i \) direction and \( \bar{\rho}(x,y,z,t) \) is the Reynolds averaged density, function of both temperature and pressure. \( \bar{u}_i'\bar{u}_j' \) is the closure term due to covariance of velocity components.

Thus,

\[
\frac{d\rho}{dt} = \frac{\partial \rho}{\partial t} + \frac{\partial \rho \bar{T}}{\partial T}
\]

(Eq. 3)

The density dependence with respect both variables (temperature and pressure) is modeled linearly (Hirsch, 2007)

\[
\frac{\partial \rho}{\partial T} = -\rho_0 \beta
\]

(Eq. 4)

Where \( \beta \) is thermal expansion coefficient.

\[
\frac{\partial \rho}{\partial \rho} = c^2
\]

(Eq. 5)

Being \( c \) the speed of sound in the fluid. Combining eq.4 and eq. 5, we obtain,

\[
\frac{1}{\rho_o c^2} \frac{dp}{dt} + \nabla \cdot \bar{u} = \beta \frac{dT}{dt}
\]

(Eq. 6)

Eq. 6 is used in numerical schemes instead of eq. 1, as it incorporates density changes from thermal expansion effects as well as from fluid compressibility.

The finite volume method (FVM) is used to discretize the flow equations in over a number of control volumes or cells, ensuring the flux mass preservation in every control volume.

The evaluation of the closure term \( \bar{u}_i'\bar{u}_j' \) requires a turbulence model. The k- \( \varepsilon \) renormalization group (RNG) model has been used in this research (Yakhot et. Al, 1991). This turbulence model has been extensively used for many CFD applications ( Tota 2009, Bombardelli et al. 2010, Väyrynen et al. 2008).

It is based on Kolmogorov cascade theory tested experimentally predicting turbulent energy amounts \( k \) and dissipation rates \( \varepsilon \) by means of two transport equations.

\[
k = \frac{1}{2} \bar{u}_i'\bar{u}_j'
\]

(Eq. 7)

\[
\varepsilon = \nu \left\{ \left( \frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i} \right) \left( \frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i} \right) \right\}
\]

(Eq. 8)

The eddy viscosity used is \( \nu_T = C_\mu \frac{k^2}{\varepsilon} \), with \( C_\mu = 0.085 \).

The advective terms in equations 2 and 6 and turbulence model are approximated with a second order space scheme with monotonicity preserving which ensures gradient conservation (Hirsch, 2007).

**Application**

The warehouse under consideration has dimensions 41 m x 41 m, being 10 meters height.

Outside temperatures are assumed to be stationary, with values of 32°C on the floor and 38°C on the ceiling, representing typical adverse conditions occurring in a very
warm summer day in Southern part of Spain. The non-slipping condition is imposed on the contour.

Diffusers are located 9 meters above the floor, and homogeneously distributed over the warehouse separated a distance \( d \) (fig. 1). Numerical flow simulations are developed to evaluate cooling performance and comfort conditions with different “\( d \)” values.

These diffusers are hexagonal with 6 vertical faces, expelling air at 21.5°C of temperature with 30° angle (fig 2.)

Computational mesh used is an orthogonal structured mesh, with 1.5 million hexahedral cells. This density of the mesh resulted from a previous sensitivity analysis, which helped to find a satisfactory compromise between accuracy of results and assumable computational cost of the simulations. In this computational mesh, cell size varies gradually as we move away from contours and diffusers, being very fine near them, and getting coarser in central regions further away from obstacles and contours.

Different values of distance \( d \) have been considered, ranging from 7 meters to 15 meters.

For each case, the velocity, pressure and temperature fields have been estimated with the numerical procedure previously described. The software flow-3d was used for the numerical simulations. Such simulations provide exhaustive definition of space-time evolution of the flow since the initiation of the cooling process, describing the transient flow during an initial period, until a stationary situation of temperature and velocities is finally reached. During the cooling process, the evolution of iso-thermal surfaces for different reference-temperatures is also analyzed. Fig. 4 shows 32.5°C stabilized iso-thermal surface after one of the numerical simulations. In practice, this surface can be considered the spatial frontier dividing two regions: the lower region being effectively cooled with relevant flow recirculation and the upper region (closer to the ceiling) affected by ceiling warming and remaining with significantly high temperatures.

In this overall numerical solution, particular attention has been placed to temperature distribution at height \( z = 1.5 \) meters, which is affecting the workers comfort conditions.

Fig. 4 shows temperature distribution in plane \( z= 1.5 \) m for \( d \) equal to 9 meters.

RESULTS AND DISCUSSION

Fig. 3a and 3b show estimated temperature along \( x \) and \( y \) axes, for different separation between diffusers. Results should be expected to be same along both directions for the case of symmetrical diffuser geometry (i.e., a radial cone). But the diffuser under consideration is hexagonal (fig. 2), yielding to direct fluxes perpendicular to its sides (y direction), and recirculation fluxes in the x-direction. An optimal value of \( d=7.5 \) meters is derived after this analysis, as it can be observed in figs. 3a and 3b.

CONCLUSIONS

A CFD technique has been successfully applied to analyze temperature and velocity distributions generated inside a warehouse, with different location and distribution of diffusers. An optimal distance between them for best performance of the cooling devices is recommended after this numerical analysis for the presented case study.

The technique has shown to be a useful tool for sensitivity analysis of the solution with respect relevant design parameters of the cooling infrastructure. Moreover, it can be helpful to find optimal configuration for an improved efficiency of the system. The
drawback of this methodology is that any small change of geometrical characteristics in
the warehouse or diffusers requires a complete new numerical simulation, with significant
cost involved.

Finer analysis should be done incorporating velocity values, as they affect to the
comfort in two different ways: higher velocities yield to lower temperature-feeling while
excessive velocities over a threshold do not help to workers comfort improvement.

ACKNOWLEDGMENTS

The authors are gratefully acknowledged to Termigo Microclimas, who provided
the data for the case study and supported the research.

REFERENCES
Elsevier.
Fabian A. Bombardelli, Inês Meireles, Jorge Matos, 2010. Laboratory measurements and
multi-block numerical simulations of mean flow and turbulence in the non-aerated
Petri J. Väyrynen, Sami K. Vapalahti, Seppo J. Louhenkilpi, 2008. On Validation of
Mathematical Fluid Flow Models for Simulation of Tundish Water Models and
Industrial Examples.
Prasad V. Tota, 2009. Turbulent Flow Over a Backward-Facing Step Using the RNG k- ε
Model. Flow Science, Inc.
V. Yakhot, S. Thangam, TB Gatski, SA Orszag, CG Speziale, 1991. Development of
Turbulence Models for Shear Flows by a Double Expansion Technique. ICASE No.
91-64, NASA.

Figures

Fig. 1. Warehouse geometry and diffusers placement. A, B points are 1.5 meters height.
Fig. 2. Diffusers geometry.

Fig. 3a. Estimated temperature for A point after different d values.

Fig. 3b. Estimated temperature for B point after different d values.
Fig. 3. Streamline colored by temperature for $d=9$ m.

Fig. 4. Temperature and velocity for horizontal plane $z=1.5$ meters height, $d=9$ m.

Fig. 5. 32.5°C iso-thermal surface for $d=9$ m.