Simulation of the flow field of water in an Olympic swimming pool.

R López, M Vacca, H Terres, A Lizardi, S Chávez and E Meza.

Universidad Autónoma Metropolitana -Azcapotzalco
Av. San Pablo No. 180, Col. Reynosa-Tamaulipas Del. Azcapotzalco, México, 02200, D.F. México,
E-mail: rlc@correo.azc.uam.mx

Abstract. This paper presents the modelling of hot-water flow at the inlet of a swimming pool in the absence of swimmers, in order to find the option that allowed the best distribution of the fluid with less temperature loss. Two different input velocities, 3 and 4 m/s and two injection-nozzles inclinations, 60° and 30°, were used in the simulation. COMSOL Multiphysics software, with appropriate boundary conditions that were obtained by direct measurements in the pool, was used. The best condition was observed for an input velocity of 4 m/s and an injection-nozzle inclination of 30°, since it presented the best distribution of the fluid at the temperature of 29.86 °C in most of the considered site.

1 Introduction

The pattern of the flow in swimming pools depends on the number and the position of the inputs and outputs of water, the geometry of the pool and feed flow rate. Currently, there are no universal regulations that govern the design of a pool, the International Swimming Federation (FINA, Fédération Internationale de Natation), has only imposed geometries of 25 m x 15 m x 1.6 m for regional competitions and 50 m x 21 m x 1.8 m in national and international competitions [1], without specifying any function related to the number of inputs and outputs of water. When a competition is going to take place organizers demand that water does not have any movement and that temperature is uniform across the pool. Pools are used for different purposes: learning and recreational swimming practice, training and competition, or for therapeutic purposes, all of which have different needs regarding water temperature. According to the American Society of Heating, Refrigeration and Air conditioning Engineers (ASHRAE), the average temperature for comfortable swimming is 27 °C, although it may vary by 5 °C [2].

Some studies have been conducted to obtain a reliable methodology for the prediction of energy losses from the pool and the property where it is located. Li et al., [3] investigated the relationship existing between the evaporation of water and the movement of air, thus determining the evaporation mass flow rate to achieve the appropriate sizing of ventilation systems for enclosures intended for swimming. Models of evaporation of water on the surface of the pool have been developed through computational fluid dynamics in two and three dimensions.

Mančić et al. [4] developed a mathematical model of a swimming pool, created with the TRNSYS software to determine the demand for energy within the enclosure of the pool, including losses by evaporation of the fluid, which were analyzed for different temperatures of the water in the pool and the surrounding air. The simulation showed that the pool water heating accounted for about 22% of the heat demand, while heating and ventilation consumed around 60% of energy within the enclosure, evaporation loss was in the range of 46% to 54% of the total losses from the pool.

Also, Cloteaux et al. [5] using experimental and computational fluid dynamics found the effect of the design of the pool on the hydraulic behavior, considering the pool as a chemical reactor with its
hydraulic characteristics. They were able to develop a mathematical model based on the principle of an agitated reactor, which could be used as a first approximation to describe the hydraulic behavior of regular swimming pools, being a suitable example for the study of physical and chemical phenomena with long characteristic times, including the prediction of the concentrations of chemical species, as well as the optimization of the chlorination and the daily renewal of water. The present paper examines the way in which the flow field is affected when varying the angle of the injection of fluid into 30° and 60° (120° and 150° from horizontal plane) in the supply of hot water, in order to find the best temperature distribution.

2 Theoretical analyses

The hypotheses about the behavior of density with changes of pressure and/or temperature are relevant in the study of flows. Multiphysics interface of non-isothermal flow [6] is used to simulate the coupling between modules of heat transfer in fluids [7] and k-ε turbulent flow [6]. Non-isothermal flow multi-physics interface contains the formulation of the equations of continuity and momentum, 

$$\frac{\partial \textbf{u}}{\partial t} + \nabla \cdot (\rho \textbf{u}) = 0$$

$$\rho \frac{\partial \textbf{u}}{\partial t} + \rho \textbf{u} \cdot \nabla \textbf{u} = -\nabla p + \nabla \left( \mu (\nabla \textbf{u} + (\nabla \textbf{u})^T) - \frac{2}{3} \mu (\nabla \cdot \textbf{u}) \right) + \textbf{F}$$

where $\rho$ is the density of the working fluid, $\textbf{u}$ is the velocity vector, $p$ is the pressure, $\mu$ is the dynamic viscosity of the fluid, and $\textbf{F}$ is the vector of body strength. The basic law that governs all heat transfer is the first law of thermodynamics, commonly known as the conservation of energy principle. For a fluid, the resulting equation is:

$$\rho C_p \left( \frac{\partial T}{\partial t} + (\textbf{u} \cdot \nabla)T \right) = -\nabla \cdot \textbf{q} + \tau : \textbf{S} - \frac{T}{\rho} \frac{\partial \rho}{\partial T} \left( \frac{\partial \rho}{\partial t} + (\textbf{u} \cdot \nabla) \rho \right) + \mathcal{Q}$$

where, $C_p$ describes the amount of heat energy required to produce a change in temperature per mass unit, $T$ is the absolute temperature, $q$ is the flow of heat by conduction, $\tau$ is the tensor of viscous efforts and $S$ is the tensor of deformation velocity. Heat transfer in fluids interface uses the following version of the heat equation to model heat transfer in fluids, either by conduction, convection or radiation:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \textbf{u} \cdot \nabla T = -(k \nabla T) + \mathcal{Q}$$

The thermal conductivity of the fluid $k$ is a scalar or tensor and the heat source (or sink) is represented by $\mathcal{Q}$. In steady state, where temperature does not change with time and velocity is set to zero, the equation that governs the conductive heat transfer is:

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = \mathcal{Q}$$

This expression describes the local evolution of temperature with respect to time ($\partial T/\partial t$) motivated by a heat conduction due to a difference in temperature between the point of study and its immediate surroundings $\nabla(-k \nabla T)$ and local heat $\mathcal{Q}$ generation. Turbulence is a property of the flow field and is characterized by a wide range of scales of flow, with a non-stationary and three-dimensional nature. The Navier-Stokes equations completely describe fluid flow. $k - \varepsilon$ turbulent flow interface, is one of turbulence models most commonly used for industrial applications. This module introduces two
equations of transport and two dependent variables: the turbulent kinetic energy, $k$ and turbulence dissipation rate, $\varepsilon$. The turbulent viscosity is modeled as

$$\mu_T = \rho C_\mu \frac{k^2}{\varepsilon}$$

where $\mu_T$ is the Eddy viscosity, also known as viscosity of turbulence and $C_\mu$ is a constant of the model.

3 Geometrical descriptions
The geometry of the public pool at the Aquatic Center of Azcapotzalco is Olympic-type, and its actual dimensions are: 50 m-length, 25 m-width and 1.3 m-depth. This pool operates in a closed circuit, i.e., the water that comes out of it is fed again, after adding the amount of lost energy. The water is collected 10 cm below the free surface through two overflow channels (spillways), located on the North and South outer walls. Fourteen circular-section entries are aimed to supply hot fluid to pool, seven of which are housed in each of the longer sides. An experimental period for the pool water recirculation was not considered, since a steady-state study was interpreted.

The numerical simulation to the pool was done cautiously modifying the model, this allowed us to work with only a portion of it, gradually assuming that in some of the adjacent entities, the geometry is symmetrical.

In this model, the geometry was developed from a work cell, representing approximately half of the longitudinal section of the complete pool. Therefore, the domain was established by a main rectangle and two secondary rectangles, one for each water outlet (weir), as well as four tiny cylinders, representing the hot water supply line. Such a procedure has made it possible to reduce the modeling space and therefore, achieve a refinement in the discretization mesh with the minimum amount of operable items. The model dimensions are: 12.5 m-length, 6.25 m-width, and 1.3 m-depth. Weirs are 5 cm height by 5.25 m long, while the nominal diameter of the pipe is only 0.05 m. The three-dimensional model is shown in Figure 1.

4 Boundary conditions
Boundary conditions used in the simulation were obtained from measurements made in the Azcapotzalco Aquatic Center pool, they are: temperatures of water in different parts of the swimming pool and the velocity of flow in the supply line, just at the entrance of the swimming pool, the values are 3 and 4 m/s. The simulation was done with COMSOL Multiphysics software using k-ε turbulence model. The objective was to obtain the temperature distribution in the bulk of the fluid considering the velocity of injection to the pool. Temperatures are: 30 °C at the entrance, the operation one of 28 °C and the third condition is on the free surface of the fluid in the pool, 29 °C. The obtained mesh had 45,662 elements, Fig. 2.
5 Results analysis and discussion

The results obtained when modelling the fluid entry with four injection nozzles at 60° are presented in Figure 3. The top-view distribution of the flow at a velocity of injection of 3 m/s in the x-y plane is observed as a partially radial behavior, caused by the tilt angle of the nozzles. Since most of the flow is directed towards the surroundings, regions of low velocity originate in the middle part of the geometry, with a section plane right to the center of the injection nozzle; this is 0.5 m below the free surface of the fluid. Central vortex is located at low velocities, with values between 0.02 m/s and 0.1 m/s; this could almost be considered as a region of stagnation, since the fluid bumped into the middle part of the walls (the vectors are preserved with a rectilinear motion), it did not adhere completely on the edges of the pool. With respect to the distribution of the temperature in the same x-y plane, isotherm curves for the same velocity of 3 m/s are presented in Fig. 4. The temperature in the central region has a vast field in which the value remains close to 29.70 °C and in the periphery it is 29.73 °C; only in the inlet region the observed value is 29.80 °C but it is a very small region.

The results for an injection velocity of 4 m/s are presented in Fig. 5. They show a parabolic behavior on the periphery of the cell, where the greatest amount of flow is grouped. The main vortex is observed in the middle part of the model, as well as small regions with geometric similarity at the corners of the swimming pool, which are considered areas of stagnation. Lowest velocities range between 0.03 m/s and 0.1 m/s, and in the input section it reaches 0.7 m/s. A more uniform distribution of temperature is observed throughout the region, with 29.80 °C in the center and 29.82 °C towards the edges of the pool; in very few places it reaches 29.83 °C. In the inlet section the fluid presents a 29.85 °C maximum temperature, the geometry of the corresponding isotherms is elliptical, in the central region, however they are slightly distorted towards the corners, Fig. 6.
When the angle of the nozzle at the inlet was modified to 30°, the fluid tended to move very close to the walls of the pool and towards the center, its value decreased considerably. For a velocity of 3 m/s the distribution of velocities had a significant change with respect to the values obtained previously, then the flow presented a lower velocity, reaching values close to 0.1 m/s in a wide section of the enclosure and towards the center this value was close to 0.01 m/s, near the corners the fluid it remained almost static with a value of 0.001 m/s, Fig. 7. With regard to the temperature distribution a slight increase with respect to the original status when the fluid entered at 60°, was observed, the value in a broad cross section of the enclosure was 29.74 °C and towards the corners, 29.77 °C, Fig. 8.0.
For an entry velocity of 4 m/s, the flow pattern remained elliptical and presented a slight modification increasing its value on the walls of the pool to 0.12 m/s; but towards the center it decreased again, down to 0.08 m/s in a considerable section of the enclosure, Fig. 9. The distribution of temperature reached the highest values between 29.86 and 29.88°C for the quasi-totality of the fluid. From the results obtained for the four conditions imposed, two of velocity variation and two angles of entry of the fluid, it can be concluded that the best option for the injection of the hot water to the pool would be a velocity of 4 m/s and nozzles inclination of 30°, since this produces a distribution of the fluid with less temperature loss.

6 Conclusions
This paper presents the modelling of the flow of hot water at the entry of a pool in the absence of swimmers, seeking the option that allows the best distribution of the fluid with less temperature loss. Two different input velocities were simulated: 3 and 4 m/s and two nozzle - inclinations: 60° and 30°. COMSOL Multiphysics software, with appropriate boundary conditions that were obtained by direct measurements in the Aquatic Center of Azcapotzalco, was used. The best condition was an entry
velocity of 4 m/s and an injection nozzle inclination of 30 °, since the best distribution of the fluid at 29.86 °C in most of the considered site was obtained.

7 References
[1] FINA. (2009). http://www.fina.org/
[2] ANSI, “ANSI/NSPI-1 (2003) Standards for Public Swimming Pools,” 2003. www.webtore ANSI.org
[3] Li Z and Heiselberg K 2005 CFD Simulations for Water Evaporation and Airflow Movement in Swimming Baths. Institutet for Bygningsteknik: Aalborg Universitet.
[4] Mančić V, Živković S, Milosavljević M and Todorović N 2014 Mathematical Modelling and Simulation of the Thermal Performance of a Solar Heated Indoor Swimming Pool. Thermal Science, 18(3), 999-1010.
[5] Cloteaux A, Gérardin F and Midoux N (2013). Influence of Swimming Pool Design on Hydraulic Behavior: A Numerical and Experimental Study Eng. 5(5), 511-524.
[6] COMSOL Multiphysics. (2014). CFD Module User's Guide. Versión 5.0. COMSOL AB.
[7] COMSOL Multiphysics. (2014). Heat Transfer Module User's Guide. Versión 5.0. COMSOL AB.
[8] Wilcox D C (1998). Turbulence Modeling for CFD. DCW Industries.