CFD numerical simulation of air natural convection over a heated cylindrical surface

M Flori and L Vîlceanu

1 Politehnica University of Timisoara, Engineering and Management Department, 5 Revolution Street, 331128 Hunedoara, Romania

E-mail: flori.mihaela@upt.ro

Abstract. In this study a CFD numerical simulation is used to describe the fluid flow and heat transfer in air surrounding a heated horizontal cylinder. The model is created in 2D space dimension involving a finite element solver of Navier-Stokes equations. As natural convection phenomenon is induced by a variable fluid density field with temperature rising, the Boussinesq approximation was coupled to the model.

1. Introduction

In industrial applications it is common that the density of a process fluid varies. These variations can have a number of different sources but the most common one is the presence of an inhomogeneous temperature field [1].

The representative problem is of a fluid external flow which is in contact with a warmer surface (in many cases, horizontal circular pipes). Initially, the heat is transmitted by conduction from the cylinder surface to surrounding fluid with formation of a thermal boundary layer around the cylinder. While the fluid begins to increase in temperature, its density decreases. The buoyant force causes the less dense fluid to rise, promoting fluid motion. With time the velocity around the cylinder increase and the heat is transmitted not only by conduction but also by convection [2].

According to Newton’s Law of Cooling the rate of heat transfer between a warmer cylindrical surface and surrounding fluid, depends on the heat transfer coefficient h[W/(m²K)], cylinder diameter d[m], surface and ambient temperatures (Ts[°C] and T∞[°C]) by equation [1]:

\[ q = h \cdot \pi \cdot d \cdot (T_s - T_\infty) [W/m] \] (1)

This study aims determining the temperature and velocity fields around a heated cylinder placed horizontally in air environment, due to natural convection. The cylinder surface has a constant temperature bigger than that of surrounding air. Therefore, a multiphysics model involving a finite element solver of Navier-Stokes equations was generated. So, both heat transfer and air flow phenomena around the horizontal cylinder are taken into account with the following assumptions: flow around the cylinder is laminar and driven in two-dimensions; initial, the fluid has a uniform temperature and is at rest, i.e. the velocity field is zero; all fluid physical properties, except density, have no significant changes with temperature.

Also, knowing the temperature field it can be determined the total normal heat flux coming from heated cylinder surface and further calculate the average heat transfer coefficient between heated horizontal cylinder and surrounding air.
2. Multiphysics model description

The CFD analysis was done by coupling the following physics involving finite element solvers of commercial Comsol Multphysics software (v. 4.2): Single-Phase Laminar Fluid Flow and Heat Transfer in Fluids. The analysis type was set as Stationary.

Concerning space dimension, in CFD analysis involving cylinders, by neglecting any effect from the geometry domain walls, the solution is constant in the longitudinal direction of cylinder [1]. So, the model was created in 2D enabling solution for half geometry along a symmetric vertical axis to reduce the modeling time.

Modeled geometry is a horizontal cylinder (having a radius of 0.06 m and constant temperature \( T_s \) of 323.15 K) placed in room-temperature air environment (\( T_\infty \)=293.15 K). The air domain is considered rectangular, having a width of 0.2 m and a height of 0.8 m. As study concern natural convection around the isothermal cylinder surface, it is represented as a semicircular cut in the rectangle left wall, its center being positioned in origin (Figure 1a). As may be seen in Figure 1a, model domain boundaries were numbered from 1 to 7 as following: boundaries 1 and 2 overlap symmetry vertical axis, boundaries 3, 4 and 5 delimit the air domain, while boundaries 6 and 7 correspond to cylinder half-surface.

![Figure 1. 2D Geometry (a) and mesh distribution (b) of model domain; dimension scale in [m].](image)

By employing a triangular mesh with extremely fine feature at domain left boundaries and fine feature for the rest of domain, the modeled geometry was discretized in 1503 elements (Figure 1b). The refined mesh along the symmetric axis and cylinder surface was added because at these boundaries the changes in fluid flow due to natural convection are expected to be more developed.

Further, a list of initial parameters were defined in the model, i.e. air thermo-physical properties evaluated at the mean fluid temperature \( (T_s + T_\infty)/2 \) of 308.15 K: density \( \rho_\infty =1.139 \text{ kg/m}^3 \), dynamic viscosity \( \mu =18.89 \text{e}^{-6} \text{ Pa} \cdot \text{s} \), thermal conductivity \( k =0.02684\text{W/(m} \cdot \text{K)} \) and heat capacity at constant pressure \( C_p =1007 \text{ J/(kg} \cdot \text{K)} \) [2].

Boussinesq approximation was also coupled to the model to represent air density changes induced by temperature field, and set according to the following equation [3-5]:

\[
\rho = \rho_\infty \left[ 1 - \beta (T_s - T_\infty) \right] \quad \text{[kg/m}^3]\tag{2}
\]

where \( \beta =1/T_\infty [\text{K}^{-1}] \) is the coefficient of thermal expansion.

The Boussinesq approximation assumes that variations in density have no effect on the flow field around horizontal cylinder, except that they give rise to buoyant forces [1].
2.1. Laminar Fluid Flow physics settings

Boundary conditions specified in the model were: Symmetry for boundaries 1 and 2, Open Boundary/Normal Stress for boundary 3 and Wall, No-Slip for boundaries 4 to 7.

The Symmetry condition means that an identical process takes place to the left, outside the domain. The Open Boundary condition describes boundaries that are open to large volumes of fluid which can both enter and leave the domain. The Wall feature includes a set of boundary conditions describing the fluid flow condition at a wall. The No Slip condition assumes that the velocity is zero (\( u = 0 \)), i.e. the fluid at the wall is not moving [1].

In order to express the gravity influence on the fluid flow around the cylinder, a volume force (buoyant force) was also defined on y-direction, with the following expression [5]:

\[
F_y = g \cdot (\rho_\infty - \rho) \ [N/m^3]
\]  

(3)

where \( g = 9.81 m/s^2 \) is gravity acceleration.

2.2. Heat Transfer physics settings

As stated above, in the model the cylinder surface (boundaries 6 and 7) is kept at constant temperature \( T_s = 323.15 K \), while the initial temperature of air domain is room temperature \( T_\infty = 293.15 K \). As air warms up, it rises upwards to upper boundary (no. 3) therefore it was set as Open Boundary/Normal Stress. Also, thermal radiation from cylinder surface to ambient air is negligible compared to convection, and so is neglected in the model.

2.3. Model governing equations

The Navier-Stokes equations (4 and 5) describing fluid dynamics and the heat transfer in the fluid equation (6) are coupled through the bidirectional multiphysics model, with the following expressions:

- conservation of momentum equation:

\[
\rho(u \cdot \nabla)u = \nabla \left[ -p I + \mu \left( \nabla u + (\nabla u)^T \right) - \frac{2}{3} \mu (\nabla : u) I \right] + F
\]

(4)

- continuity equation, which represents the conservation of mass:

\[
\nabla \cdot (\rho u) = 0
\]

(5)

- conservation of energy equation, formulated in terms of temperature, excepting the heat source terms:

\[
\rho C_p u \cdot \nabla T = \nabla \cdot (k \nabla T)
\]

(6)

where besides air physical properties, \( u [m/s] \) is velocity, \( p [Pa] \) is pressure, \( T[K] \) is absolute temperature and \( F[N/m^3] \) is volume force (body force).

The momentum and energy equations are inherently coupled since the temperature appears in the momentum equation and the velocity in the energy equation [1].

3. Results and discussion

CFD analysis results were obtained in terms of temperature (in [K]) and velocity (in [m/s]) distributions in the fluid around the heated horizontal cylinder due to natural convection (Figures 2 and 3).

Temperature field develops around the cylinder (see magnification in Figure 2) and also above it along the symmetry line which passes through the cylinder center. At small temperature differences between warm surface and surrounding fluid, a laminar flow is expected [6].

Also, the velocity field develops in lateral direction, but has bigger intensity above cylinder surface where maximum velocity values may be identified in vertical direction along the symmetry line (surface colored in red in Figure 3).

Moreover numerical values of temperature and velocity variations with y-coordinate of computed domain were obtained and presented in Figures 4.a) and b). In these plots, the y-coordinate axis (horizontal axis) corresponds to the vertical symmetry line of computed domain where x-coordinate is
0. So, in y-direction the computed domain was plotted from $y = -0.2 \text{ m}$ to $y = 0.6 \text{ m}$, resulting domain height of 0.8 m.

![Figure 2. Computed temperature field; color scale in [K].](image1)

![Figure 3. Computed velocity field; color scale in [m/s].](image2)

Analyzing variation from Figure 4.a one may observe that below the heated cylinder surface ($y < -0.06 \text{ m}$) the air warms up only on a small distance of about 0.02 m. Above the heated cylinder surface air warms up constantly, having maximum value near cylinder surface ($y = 0.06 \text{ m}$) and about 302 K at height $y = 0.6 \text{ m}$.

Analyzing air velocity distribution from Figure 4.b, one may observe that below the cylinder surface the velocity is zero, indicating that no convection currents are formed here. Above cylinder surface air velocity increase rapidly attending a maximum value of 0.54 m/s at height $y = 0.6 \text{ m}$.
Moreover, the computed temperature field generates a total normal heat flux coming from heated cylinder surface of $q = 57.242 \text{ W/m}$. With this value, average heat transfer coefficient between heated horizontal cylinder and surrounding air is calculated with relation (1) and obtained value is $h = 5.06 \text{ W/m}^2\text{K}$.

4. Conclusions
A finite element analysis of free convection around a heated horizontal surface was done to determine temperature and velocity distributions in air surrounding environment. Besides initial and boundary conditions, the influence of gravity on air flow around the cylinder was expressed in terms of volume force, while the Boussinesq approximation was used to represent the variable density field with temperature rising.

By computing the temperature field around horizontal cylinder due to natural convection, the total normal heat flux was determined and average heat transfer coefficient between heated horizontal cylinder and surrounding air was calculated.

References
[1] ***COMSOL Multiphysics User’s Guide, ©Copyright COMSOL AB., 2010
[2] Linenhard J H IV and Linenhard J H V 2008 A heat transfer textbook. Third edition, Phlogiston Press, Cambridge MA
[3] Sairamu M, Nirmalkar N and Chhabra R P 2013 Natural convection from a circular cylinder in confined Bingham plastic fluids, International Journal of Heat and Mass Transfer 60 567–581
[4] Chandra A and Chhabra R P 2012 Laminar free convection from a horizontal semi-circular cylinder to power-law fluids, International Journal of Heat and Mass Transfer 55(11–12) 2934-2944
[5] Pasut W and De Carli M 2012 Evaluation of various CFD modelling strategies in predicting airflow and temperature in a naturally ventilated double skin façade, Applied Thermal Engineering 37 267–274
[6] Atayilmaz Ş O and Teke I 2009 Experimental and numerical study of the natural convection from a heated horizontal cylinder, International Communications in Heat and Mass Transfer 36 731-738

Figure 4. Variation of air temperature (a) and velocity (b) with y-coordinate.