Case Studies in Chemical and Environmental Engineering 6 (2022) 100264

Contents lists available at ScienceDirect

Case Studies in Chemical and Environmental Engineering

journal homepage: www.sciencedirect.com/journal/case-studies-in-chemical-and-environmental-engineering

Case Report

Theoretical and experimental analysis of temperature decay along an industrial chimney using analytical and *k*-*ω* turbulence models

Mostafa El-Shafie

Institutes of Innovation for Future Society, Nagoya University, Furo-cho, Chikusa-ku, Nagoya, 464-8601, Japan

ARTICLE INFO

Keywords:
Chimney design
CFD
Natural convection
Heat transfer
Modified lumped model

ABSTRACT

Numerical simulations and analytical calculations were performed to predict the bulk and wall temperatures of hot gases undergoing turbulent flow inside industrial chimneys. The natural convection was numerically simulated using the commercial COMSOL computational fluid dynamics code with Wilcox's k-ω turbulence model on 2D cross-sections of a chimney. Two analytical models, the Cortés and modified lumped models, were used to determine the temperature distribution along the chimney height. The numerical and analytical models were validated with empirical data from a glass furnace chimney. Cross-sectional velocity and temperature profiles were simulated at different chimney heights, along with dimensionless parameters such as the Nusselt number. Good agreement was found between the analytical, numerical and empirical results. The modified lumped model performed particularly well – this model is especially detailed, taking into account the chimney roughness and fouling. It was concluded that the modified lumped model can be used as an analytical model, and the k-ω turbulence model as a numerical simulation model, by designers of industrial chimneys.

1. Introduction

The chimney is an important structure in the chemical industry, where it is used to disperse flue gases far from the ground level, including important air pollutants resulting from high-temperature combustion, such as nitrogen oxides, carbon monoxide and carbon dioxide. A chimney must be capable of dispersing flue gas without suffering deterioration, while some pollutants must be removed before the flue gas is emitted into the atmosphere. The most common methods for removing pollutants from the flue gas involve reducing the exit temperature below the dew point of the pollutants, which can be as high as 115–140 °C [1]. However, acid deposition on the chimney's inner wall should be avoided to prevent corrosion.

In factories and power plants, flue gases are always discharged through the chimney duct at high temperatures. Therefore, heat transfer through the chimney walls must be considered in chimney design. The heat loss by convection from the outer surface of the chimney wall depends on the wind velocity. The natural convection of hot gases depends on the flue gas temperature and the buoyancy force. The temperature distribution throughout the height of a chimney has been investigated using two analytical models [2]. Moreover, a 1D lumped model has been used to calculate the heat transfer and the temperature gradient along the chimney height [1]. The 1D lumped model has been applied using standard heat transfer correlations. Furthermore, calculation of the axial variation in mean bulk temperature has been carried out by iteratively evaluating a single exponential function. Theoretical and numerical analysis has been conducted for the prediction of the mean bulk temperature [3]. Also, the agreement between theoretical and numerical results has been investigated. A numerical simulation has been used to study the thermo-convective behaviour of a smoke conduit [4].

Computational fluid dynamics (CFD) is a branch of fluid mechanics in which problems involving fluid flows are solved using computer simulations. The relationship between the extent of cold inflow and the effective total chimney height can be obtained from the differential pressure drop data, collected over a natural draft air-cooled heat exchanger model [5]. Thermal radiation emitted by smoke heats the internal chimney cylinder wall, and different configurations have been investigated to understand the influence of cylinder spacing and cylinder height [6]. The natural convection from a heated plate placed inside a chimney has been investigated both experimentally and numerically [7]. Furthermore, the effect of chimney height on heat transfer distribution rates from the vertical plate and flow field inside the chimney has been studied. A semi-analytical approach has been developed for predicting the mean bulk and interface temperatures of hot gases moving inside thick-walled metallic tubes with isothermal conditions at the outer surface [8]. Three design rules have been studied and tested for 13 industrial chimneys [9]. Moreover, the cross-wind vibration of a new

E-mail address: mostafaelshafie81@gmail.com.

https://doi.org/10.1016/j.cscee.2022.100264
Received 28 June 2022; Received in revised form 7 October 2022; Accepted 8 October 2022
Available online 10 October 2022

Nomenclature

A_i	inner cross-sectional area of the duct
$c_{p,i}$	specific heat of combustion gases
D	chimney diameter
D_e	outside diameter
D_i	inside diameter
g	acceleration due to gravity
$\mathrm{Gr}_{e,L}$	external Grashof number based on L, $g\beta_e(T_{w,e} - T_\infty)L^3/\nu_e^2$
h_e	external convective coefficient
h_i	internal convective coefficient
k_e	air thermal conductivity
k_i	flue gas thermal conductivity
k_w	thermal conductivity of the chimney wall
L	chimney height
$\dot{m}$	flue gas mass flow rate
Nu_e	external Nusselt number, $h_e D_e/k_e$
$\overline{\mathrm{Nu}_e}$	streamwise mean of Nu_e
$\overline{\mathrm{Nu}_{e,L}}$	streamwise mean Nusselt number based on height L, $(L/D_e)\,\overline{\mathrm{Nu}_e}$
Nu_{eq}	equivalent Nusselt number, UD/k_i
$\overline{\mathrm{Nu}_{eq}}$	streamwise mean of Nu_{eq}
Nu_i	internal Nusselt number, $h_i D_i/k_i$
$\overline{\mathrm{Nu}_i}$	streamwise mean of Nu_i
Pr_e	Prandtl number of the ambient air
Pr_i	Prandtl number of the internal flue gas
q_r	radiative heat flux
R	chimney radius
R_i	inner radius
Re_e	external Reynolds number based on wind velocity, $u_\infty D_e/\nu_e$
Re_i	internal Reynolds number, $4\dot{m}/\mu_i \pi D_i$
$T(r,z)$	temperature of combustion gases
T_b	bulk temperature of combustion gases
T_o	flue gas inlet temperature
T_r	effective radiating temperature of the surroundings
T_w	wall temperature
$T_{w,e}$	outer surface temperature
$T_{w,i}$	inner surface temperature
T_∞	ambient air temperature
U	overall heat transfer coefficient
u	instantaneous velocity in vector notation
u_∞	wind velocity component normal to the chimney axis
z	axial coordinate

Greek symbols

α_e	air thermal diffusivity
α_i	thermal diffusivity of combustion gases
β_e	thermal expansion coefficient of air
β, β_0, β_0^*	closure coefficients
γ	correction factor for curvature effects
γ_i	dynamic viscosity of the combustion gases
ε_w	outside wall emissivity
k_e	thermal entrance factor
k_p	temperature profile factor
k_t	transition regime factor
μ	Molecular viscosity
μ_T	eddy viscosity
ν_e	air kinematic viscosity
ν_i	kinematic viscosity of the combustion gases
ρ_i	flue gas density
σ	Stephan–Boltzmann constant
σ_ω, σ_k^*	closure coefficients
ω	specific dissipation rate; vorticity vector magnitude

Subscripts

e	ambient air; outer side
i	combustion gases; inner side
L	based on length L
w	wall
free	natural convection
rad	thermal radiation

100 m steel chimney has been studied to measure the damping properties of the chimney [10]. Numerical analysis of transient natural convection in a vertical chimney that is symmetrically heated at a uniform heat flux has been investigated [11]. In some industries, such as glass melting, materials that are deposited on the chimney surface are an important parameter affecting heat transfer through the chimney height. These fouling materials in glass furnaces have been analyzed to determine their thermal conductivity [12]. Several approaches for designing optimum stack filters have been characterized in terms of costs [13]. A flue gas monitoring system has been designed based on gross gamma activity released from the stack in a nuclear reactor [14]. Different methods have been studied the calculation of the ultimate strength of circular sections of reinforced concrete chimneys at various wind loading [15]. High outlet temperature gases are generated from most of industrial applications, therefore, various studies have been discussed the waste heat recovery from exhaust gases. The design of a heat recovery unit for the optimum use of waste heat from flue gases of the industrial glass furnace has been investigated [16]. The waste heat content in exhaust gases was delivered to preheat the absorption chiller dilute solution (lithium bromide). The performance of thermoelectric generator used for waste heat recovery from the vertical chimney has been investigated [17]. The waste heat recovery has been reviewed for different thermal energy processes using thermoelectric generators [18]. It has been reported a review on the previous works of the waste heat recovery from the industrial chimneys and vehicles.

This study investigates the chimney design parameters using two analytical models, the Cortés and modified lumped models, and compares the results with those of a CFD *k-ω* turbulence model. Analysis is carried out to evaluate the ability of these models to predict the temperature decay along the chimney height. The analytical and numerical simulation results are validated by comparing them with the empirical data from an industrial glass furnace chimney. The temperature and velocity distribution along the chimney height are determined using Wilcox's *k-ω* turbulence model. The temperature distribution results from both the analytical and numerical models are compared with the empirical data. Moreover, convective and radiative heat transfer are considered for both the *k-ω* turbulence model and analytical models, and the internal Nusselt number results of the analytical and simulation models are compared.

2. The industrial chimney investigation models

The main function of a chimney, or 'stack', is to disperse flue gases from a boiler, stove, or industrial furnace. The outlet flue gases have a much higher temperature than the outside air, so they move up and out of the chimney through natural convection. The most important design parameter for industrial chimneys is the minimum height required to obtain an adequate dispersion of pollutants. The height and structure of the stack are used to determine the exit temperature of the flue gases and subsequently the available static draft and the required fan specifications. Condensation of water, which occurs at temperatures around 40 °C, should be prevented – condensed water can entrain dissolved sulfur compounds, leading to corrosion. A good chimney design should therefore have an insulation wall to avoid cold spots. Moreover, in the glass industry, materials deposited on the chimney surface can affect its thermal properties and should be taken into account.

Due to the high Reynolds number of the combustion process in industrial furnaces, there is often highly turbulent flow, resulting in random fluctuations in the flow velocity, temperature and species concentrations. The level of mixing of momentum, heat and mass is increased in turbulent flows compared to laminar flows. The chemical reactions in glass melting furnaces require high temperatures. The operation of a glass melting furnace is illustrated in supplementary materials Fig. S1, in which natural gas fuel is used to raise the temperature to 1450 °C. In this study we simulated the flue gas drafting process in the chimney of a glass melting furnace located at El-Araby Group Co., Quesna, Egypt. The natural convection of the industrial chimney was modelled using the COMSOL Multiphysics™ package. The simulation results are compared with the analytic Cortés and modified lumped models. The temperatures and velocities inside the chimney are determined. The chimney design is evaluated by comparing the empirical measurements, analytical calculations and simulation results. The calculation of the temperature profile can be accomplished by estimating the heat transfer rate from the flue gases to the surroundings through the chimney wall. Usually, the given data include gas flow rates and inlet temperatures, ambient conditions, and selected height and diameter. The chimney is insulated with aluminium silicate material. A gas analyzer was used to measure flue gas composition. At first, a dry gas analysis was performed to determine the concentration of gas components, flow rate, and thermodynamic properties.

2.1. Analytical models

The heat transfer processes for combustion gases inside a chimney involve internal forced convection and external heat transfer to the surroundings. There are various methods of simulating these processes and calculating the temperature decay and distribution along the chimney length. A 1D lumped model has been proposed by Cortés [1] to estimate the mean bulk temperature at the exit of the chimney. Campo [2] has proposed a 2D lumped model, consisting of a partial differential energy equation subjected to a non-linear convective boundary condition via a simple solution for a uniform. It has been suggested that a modified lumped model can more accurately predict the temperature distribution along the chimney [3].

2.1.1. Cortés model

In the Cortés model, the external and internal heat transfer coefficients are calculated using a dimensionless group. The internal Nusselt number is determined as a function of Reynolds number Re_i and Prandtl number $\mathrm{Pr_i}$ [1],

$$\overline{\mathrm{Nu_i}} = k_t k_p k_e \frac{0.023\mathrm{Pr_i^{0.9}}\ \mathrm{Re_i^{0.8}}}{\left(1+\frac{8\ \mathrm{Pr_i^{1.7}}}{\mathrm{Re_i^{0.3}}}\right)^{1/3}} \tag{1}$$

where k_t, k_p and k_e are the transition regime factor, the temperature profile factor and the thermal profile factor, respectively.

It is necessary to calculate the outside convective heat transfer, since the chimney's outside walls are exposed to the low-temperature ambient conditions. The most common approximation for the free convection is the one proposed by Churchill and Chu [19],

$$\overline{Nu_{e,L}} = \left[0.825 + \frac{0.387 Ra_{e,L}^{1.6}}{\varnothing(Pr_e)}\right]^2, \tag{2}$$

where

$$\varnothing(Pr_e) = \left[1 + (0.492/Pr_e)^{\frac{9}{16}}\right]^{8/27} \tag{3}$$

For an isothermal wall, $\mathrm{Ra}_{e,L}$ is the Rayleigh number based on the height L:

$$Ra_{e,L} = \frac{gB_e\left(T_{w,e} - T_{w,\infty}\right)L^3}{\nu_e \alpha_e} \tag{4}$$

The external Nusselt number for forced turbulent flow convection, using the classical correlations of Zukauskas et al. [20] for an isothermal cylinder in cross flow,

$$\overline{Nu_e} = 0.023 Re_e^{0.8} Pr_e^{0.4}, \tag{5}$$

It can be used in the range $2\times10^5 < Re_e < 2\times10^6$, where $Re_e = u_\infty D_e/\nu_e$ is the Reynolds number based on the outside diameter and u_∞ is the wind velocity. The wind velocity is assumed perpendicular to the chimney axis. The local equivalent Nusselt number Nu_{eq} is used to describe the mixed convection across the chimney as follows [1]:

$$\frac{1}{Nu_{eq}} = \frac{1}{Nu_i} + \frac{1}{k_e/k_i}\frac{1}{Nu_e} + \frac{1}{2}\frac{k_i}{k_w} ln\left(\frac{D_e}{D_i}\right) \tag{6}$$

2.1.2. Modified lumped model

In the modified lump model, the chimney is considered as a heat exchanger and the bulk temperature relation can be used, which depends on the overall heat transfer coefficient, the mass flow rate of flue gases, the heat capacity of flue gases and the area, but the variation in results is too high. For that by the comparison between the two correlations to determine the bulk temperature the 1D lumped model which used the non-dimensional form is more useful. The change of gas temperature can be expressed by the first order differential equation [2];

$$\frac{\mathrm{d}T_{\mathrm{bz}}}{\mathrm{d}z} = -\frac{U\pi D}{\dot m\ c_{p,\mathrm{i}}}\ (T_\mathrm{b} - T_\infty), \tag{7}$$

where the mass flow rate is $\dot m$, the velocity profile is v, the surrounding air temperature is T_∞, and the gas temperature is T_g. The heat transfer resistances can be expressed as a series thermal circuit, involving the inside and outside convection coefficients, h_i and h_e, respectively, and thermal conductivity resistance of the wall. The overall heat transfer coefficient U can then be defined as;

$$\frac{1}{U} = \frac{1}{h_\mathrm{i}} + \frac{1}{h_\mathrm{e}}, \tag{8}$$

where the bulk temperature at the chimney inlet at zero position ($T_\mathrm{b} = T_\mathrm{o}$)

$$T_{\mathrm{b}(z)} = \frac{\int \rho_\mathrm{i}\ \mathrm{V}\ c_{p,\mathrm{i}}\ \ T\ \mathrm{dA}}{\dot m c_{p,\mathrm{i}}} = \frac{\int V\ \ T\ \mathrm{d}A}{V\mathrm{dA}} \tag{9}$$

The modified lumped model assumes low velocity and incompressible flow, and it neglects the change in density and heat capacity due to the temperature decay along the chimney duct.

It has been shown that in the modified lumped model, the internal heat transfer coefficient can be determined using the Graetz correlation [21]. The buoyancy force has also been studied: it can be defined as the rate of change of gas density with respect to temperature along the chimney height. The buoyancy force depends on the roughness of pipe, which can be defined in terms of a friction factor f [22]. In the case of laminar flow ($Re_i < 2300$), the inner Nusselt number can be calculated by using the Graetz correlation [2];

$$\overline{Nu_i} = 3.657 + \frac{0.0668\ G_z^{1/3}}{0.04 + G_z^{-2/3}} \tag{10}$$

where the Graetz number can be written as

$$G_z = \frac{Re_i\, Pr_i D_i}{X_i} \tag{11}$$

The first step to calculate the heat transfer coefficient inside the duct is to calculate the Reynolds number based on the chimney inner diameter. Equation (10) is used to calculate the Nusselt number for laminar flow as a function of Graetz number. But in the case of turbulent flow, the Nusselt number is determined as a function of Prandtl number, Reynolds number and friction factor. The Reynolds number for turbulent flow ($2300 \le Re_i < 5 \times 10^6$) is [23]

$$\overline{Nu_i} = \frac{(f/8)(Re_i - 1000)Pr_i}{1 + 12.7\sqrt{f/8}.\left(Pr_i^{2/3} - 1\right)}. \tag{12}$$

The radiation heat transfer coefficient is of a comparable order of magnitude, depending on the effective radiating temperature of the surroundings and the wall emissivity. The radiation heat transfer analysis was determined according to equation (14). Moreover, the two-dimensional simulation model concluded heat transfer losses by radiation. The model physics considered the heat transfer in solids and fluids, the emissivity input value was 0.9. The chimney loses heat through thermal radiation due to the interaction with ambient radiation. Although the ambient air is essentially a nonparticipating gas, the coupling with convection is an interesting heat transfer problem. Then the total external Nusselt number can be formulated as [24].

$$\overline{Nu_e} = (\overline{Nu_e})_{free} + (\overline{Nu_e})_{rad} \tag{13}$$

The second term of radiation is

$$q_r = \varepsilon_w \sigma \left(T_{w,e}^4 - T_r^4\right) \tag{14}$$

where ε_w is the wall emissivity, σ is the Stephan–Boltzmann constant $5.67 \times 10^{-8}\ \mathrm{W/m^2\,K^4}$ and T_r is the effective radiating temperature of the surroundings and can be considered equal to the surrounding temperature ($T_r \approx T_\infty$).

The calculation which only considers natural convection leads to inaccurate temperature profiles. Adding thermal radiation to the external heat transfer coefficient yields profiles close to the real temperature distribution along the chimney walls. The modified analytical model has been described in detailed in our previous study [2]. The most significant difference between both analytical models is the internal *Nu* number calculation formula. In the modified model we used the Graetz equation to determine the Nusselt number as a function of the friction factor. In the present study, we compare both analytical models with the turbulence *k-ω* model, specifically focusing on the temperature distribution and the temperature decay along the chimney length.

These models are used in the design of chimneys to calculate the exit temperature to prevent flue gas condensation. The main difference between the Cortés model and the modified lumped model is that in the Cortés model, the internal Nusselt number is a function on the internal Reynolds number and the internal Prandtl number, while in the modified lumped model, the internal Nusselt number is a function of the internal Reynolds number, the internal Prandtl number and the friction factor. That is, in the modified model, losses due to friction are considered to improve predictions of the temperature decay along the chimney length. The modified lumped model also takes into account the impact of the fouling layer thickness on the conduction process.

For outside walls forced convection caused by the high-speed wind. The overall heat transfer coefficient is given by

$$U = \frac{1}{A_{inside}}\left[\frac{1}{(hA)_{\text{inside}}} + R_{twall} + \frac{1}{(hA)_{\text{outside}}}\right]^{-1} \tag{15}$$

Both analytical models can effectively predict the temperature decay along industrial chimneys.

2.2. Governing equations of the numerical CFD k-ω model

Wilcox's *k-ω* model is used for the numerical simulations in this study – it was chosen because of its popularity and high performance [25]. Generally, the two model equations have a turbulent kinetic energy equation term, *k*, and a destruction or specific dissipation term *ω*. The 2D-axisymmetric model was simulated using the *k-ω* model which can be expressed by the following two equations,

$$\rho(u.\nabla)u = \nabla.[-PI + K] + F + \rho g, \tag{16}$$

$$\rho\nabla.u = 0, \tag{17}$$

$$K = (\mu + \mu_T)\left(\nabla u + (\nabla u)^T\right), \tag{18}$$

$$\rho(u.\nabla)k = \nabla.\left[\left(\mu + \mu_T\sigma_k^*\right)\nabla k\right] + P_k - \beta_0^*\rho\omega k, \tag{19}$$

$$\rho(u.\nabla)\omega = \nabla.[(\mu + \mu_T\sigma_\omega)\nabla\omega] + \alpha\frac{\omega}{k}P_k - \beta_0\rho\omega^2. \tag{20}$$

where $\mu_T = \rho\frac{k}{\omega}$ and $P_k = \mu_T[\nabla u.(\nabla u + (\nabla u)^T)]$.

This numerical simulation study uses the following values for the Wilcox constants: $\beta_0 = \frac{9}{125}$, $\beta = 5.2$, $\beta_0^* = \frac{9}{100}$, $\alpha = \frac{13}{25}$, $\sigma_k^* = 1/2$, and $\sigma_\omega = 1/2$.

3. 2D geometry and boundary conditions

The natural convection of exhaust gas flow inside a glass furnace chimney was analytically modelled using both the Cortés model and the modified lumped model. The results from the analytical models were compared with the results from COMSOL CFD analysis, in which the temperature decay along the chimney duct was simulated using the *k-ω* model. Both of the analytical models and the *k-ω* turbulence model were used to evaluate and predict the gas temperature and velocity distributions. The analysis was carried out for the glass melting furnace chimney, for which a schematic diagram of the chimney parameters and the 2D geometry are shown in Fig. 1. The heat transfer calculation took into account the fouling layer deposited on the chimney internal wall. The deposit layer thickness was assumed to be fixed along the chimney wall at 1.5 cm. The glass melting furnace chimney parameters were used as input data for the analytical and simulated models.

The simulation results were compared with empirical measurements taken from the El-Araby glass factory in Quesna, Egypt, in which natural gas is used as a fuel to melt soda lime glass materials. The outlet flue gas temperature was registered at the chimney inlet. The flue gases are removed from the glass furnace by natural convection [16]. The outlet exhaust gas flow rate was determined based on a chemical composition analysis of the exhaust gas components (CO_2, CO, NO_x, SO_x) measured by a gas analyzer, which has previously been discussed in detail [3]. A 2D axisymmetric domain was used in the numerical turbulence modelling and the analytical calculations. The boundary conditions are presented in details in Table 1 . The 2D-axisymmetric model is simulated, the flow gradient is set to be zero in the normal direction to the axisymmetric boundary conditions. Furthermore, the velocity is settled to be zero on the chimney walls by means no slip wall [26].

The inlet exhaust gas temperature was 316 °C, which was measured using a K-type ceramic thermocouple with an accuracy of ±2.2 °C. The calculated feeding input gases value was 0.8642 kg/s, determined as a function of the properties of hot flue gas components (CO_x, NO_x, O_2, and H_2O).

3.1. 2D-geometry mesh parameters

A 2D grid model was chosen to examine the simulation parameters. . The 2D numerical simulation model can provide a fair match with the actual industrial measurement data. Therefore, the 2D numerical model

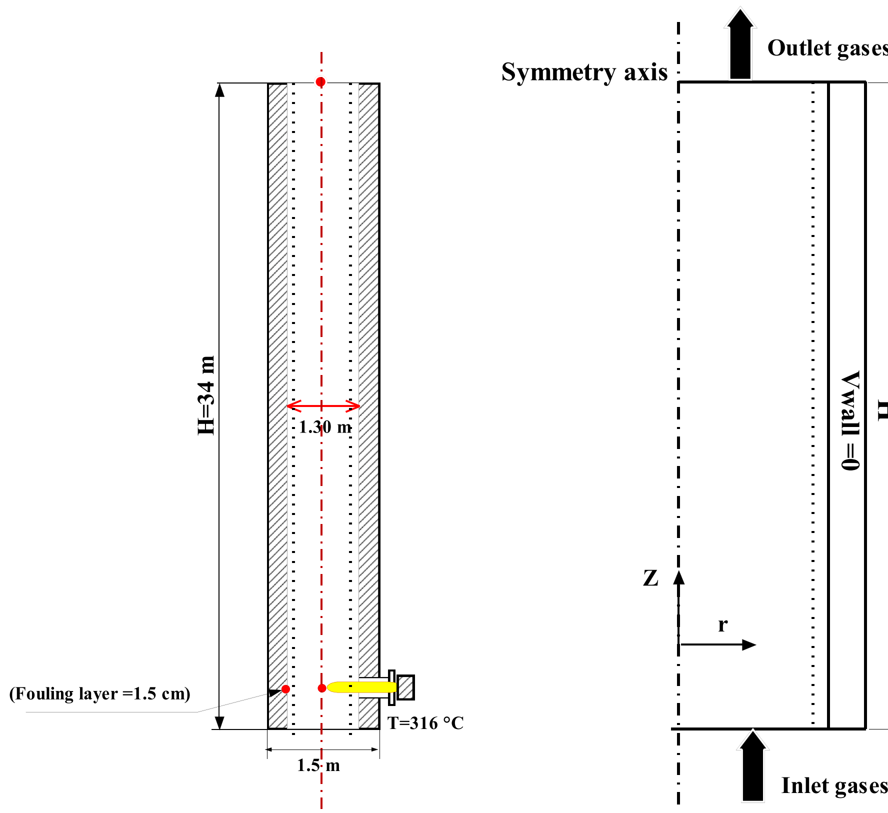


Fig. 1. Schematic diagram of 2D-geometry and boundary conditions of the chimney.

Table 1
Chimney boundary condition.

Boundary condition	Variable	Value
Inlet and outlet	Static pressure (P)	0 Pa
Chimney walls	No slip (V_{wall})	0
Ambient temperature	T_{amb}, T_{ref}	298 K
Feeding mass flow rate	$\dot{m}$	0.8642 kg/s
Inlet gas temperature	T_{in}	589 K
Insulation material	Aluminium silicate	$k = 0.9$ W/m.K
Deposit or fouling layer	Thermal conductivity	$k = 0.45$ W/m.K
Surface emissivity	ε	0.9

is sufficient for describing the temperature decay along the industrial chimneys. Two mesh types "coarse", and "normal" are investigated in the current 2D simulation model. We select the normal mesh because the velocity and temperature profiles show a deviation lower than 1.12% with respect to the coarse mesh results. Moreover, the normal mesh type can give more accurate results for the current simulation. The high-resolution mesh can be found in supplementary materials Fig. S2, the generated mesh is close to the deposit layer and thermal boundary layer. 'Normal' mesh type was tested and consists of 116,468 elements with minimum quality of 0.1722.

The k-ω turbulence model was used to study the exhaust gas behaviour along the chimney duct. The turbulence model equations were solved for incompressible and non-isothermal flow. The natural convection flow was calculated using the Boussinesq approximation [27]. The momentum equation included the buoyancy force F_z which can be determined as [28].

$$F_z = -g(\rho - \rho_{ref}) = g\rho_{ref}\,\beta(T - T_{ref}), \tag{21}$$

where the acceleration due to gravity is g in m/s^2, ρ is the gas density in kg/m^3, ρ_{ref} is the reference density in kg/m^3, β is the thermal expansion coefficient in K^{-1}, the exhaust gas temperature at a given position is T, measured in K, and T_{ref} is the reference or ambient temperature in K. The thermal expansion coefficient is taken at a value of 3.47×10^{-3} K^{-1}.

The convection, conduction and radiation heat transfer are calculated to predict the outlet temperature. Both analytical models results are compared with the k-ω turbulence model simulated by COMSOL program. The bulk or average temperature of the exhaust gases on a chimney duct cross section were estimated according to [29].

$$T_b = \frac{1}{\rho_m A_c u_m}\int \rho u.T dA_c \tag{22}$$

where ρ_m is the average density in kg/m^3, u is the mean velocity vector in m/s, the radial direction temperature profile is T, and the average velocity u_m is determined as

$$u_m = \frac{\int \rho u.dA_c}{\rho_m A_c} \tag{23}$$

3.2. k-ω turbulence modelling and solver convergence

The El-Araby chimney, with outside diameter of 150 cm and inside diameter of 130 cm, was simulated using the k-ω turbulence model, solved by the COMSOL Multiphysics™ package using a finite element method [30]. The thermodynamic properties of exhaust gases were determined and used as input data to the simulation program. The following properties of inlet exhaust gases were used: the thermal conductivity $k_g = 0.06313$ W/m.K, the molecular viscosity $\mu_g = 3.076 \times 10^{-5}$ Pa.s, the density $\rho_g = 0.496$ kg/m^3, and the heat capacity $C_p = 1.1523$ kJ/kg.K. Thus, the chimney's dimensionless group parameters are $Pr = 0.561$, $Re = 21473.21$ and $Ra = 8.8 \times 10^{10}$. Because $Pr < 1$ and Reynolds number is $2300 < Re \leq 5 \times 10^6$, the thermal boundary layer is out of the viscous boundary layer. The Wilcox k-ω turbulence model was applied to study the eddy viscosity models [25]. The temperature decay and distribution inside and along the chimney wall were simulated, taking into account heat transfer processes in solids and fluids. The flow of exhaust gases through the chimney was solved as a natural convection problem including the body force and gravity effect. The natural convection of exhaust gases includes the buoyancy force, which depends on the density variation and acceleration due to gravity. The Kays–-Crawford model was used to evaluate the thermal conductivity and determine the turbulence Prandtl number *Pr* [31]. The viscous heat dissipation and pressure effects were neglected in the CFD simulation modelling. The initial temperature values of the fluid and solid zones were set to 25 °C. The radiation and convective heat transfer from outside walls were modelled; a second-order discretization was used for all variables. The internal Nusselt number was determined for the modified lumped model and the Cortés model correlations, and the simulated value was compared with both the analytical models. The modified lumped model Nusselt number is a function of the friction factor *f*, the inner Reynolds number *Re* and the Prandtl number *Pr*.

4. Results and discussion

We used the analytical Cortés and modified lumped models to determine the temperature distribution along the chimney duct, and exhaust gas parameters such as mass flow rate and Reynolds number were calculated based on chemical composition analysis. These models were used to predict the bulk temperature, the pressure drop and the natural convection behaviour of flue gases inside the chimney. The convection, conduction and radiative heat transfer were calculated to predict the outlet temperature. Both analytical model's results are compared with the *k*-*ω* turbulence model simulated by COMSOL program. A CFD simulation analysis was performed to investigate the performance of the chimney. The Wilcox *k*-*ω* turbulence model was solved to predict the temperature and velocity distribution along the industrial chimney duct. Moreover, the numerical simulation results were experimentally validated.

4.1. Numerical simulation results

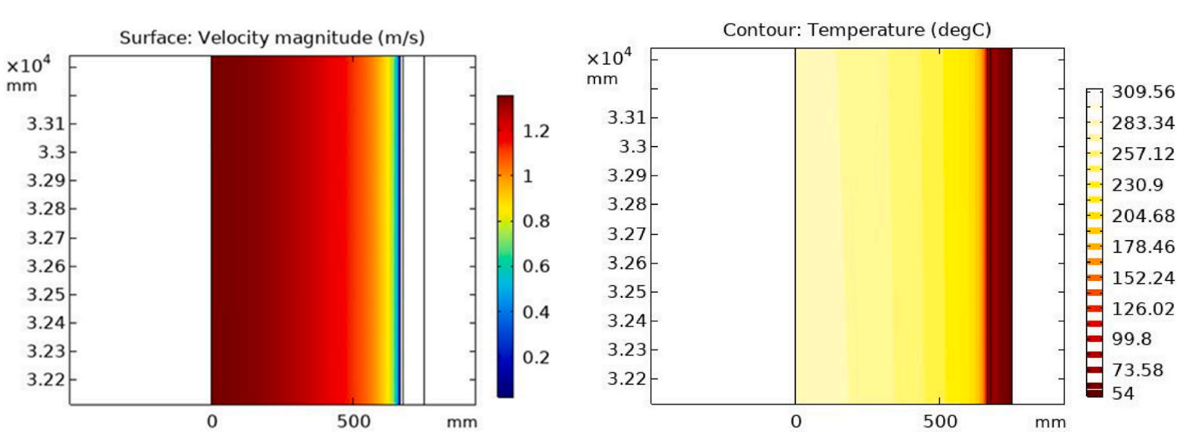
In this section the results of the simulations performed using the *k*-*ω* turbulence model for the glass furnace industrial chimney are presented. Fig. 2a shows a plot of the 2D-geometry velocity distribution. The velocity distribution takes its maximum value at the chimney centre, and reaches zero at the chimney wall. A plot of the temperature distribution of hot gases inside the chimney duct and the chimney walls is shown in Fig. 2b). The temperature gradient reaches the maximum value at the centre. The temperature gradient decreases from the chimney plume centre to the outside wall; the cooling effect results from the heat transfer between the outside walls and the ambient air. It can be seen that the highest temperature is in the centre of the chimney duct and the lowest temperature is at the outside wall, as expected.

The cross-sectional temperature distributions from the numerical simulation results at different positions are shown in Fig. 3a. It can be seen that the temperature decreased as the chimney height increased. Also, the temperature gradient decreased from the chimney duct centre to the outside wall, due to the heat transfer loss to the outside air. Furthermore, the temperature gradient decreased as the chimney height increased. Fig. 3b shows the temperature decay along the chimney height for different chimney radii. The ratio of the simulated temperature to the inlet gas temperature decreased with the chimney height. Also, it was found that the temperature ratio increased as the chimney simulated radius increased from $r/R_i = 0.25$ to $r/R_i = 1$ because of the effect of heat exchange with the outside air. These simulation results are presented for the radial direction from $z = 0$ m to $z = 34$ m; the temperature at the chimney inlet was relatively high to facilitate natural convection of the flue gases.

Fig. 4 shows the cross-sectional velocity profile inside the chimney duct at different heights. The velocity profile gradient typically decreased from the centre to the chimney wall: the velocity was high at the chimney centre and reached zero at the chimney wall. Also, the velocity gradient at the chimney inlet was lower than at the chimney exit. It was found that the velocity distribution increased in the r-direction or inside the chimney duct with the chimney height.

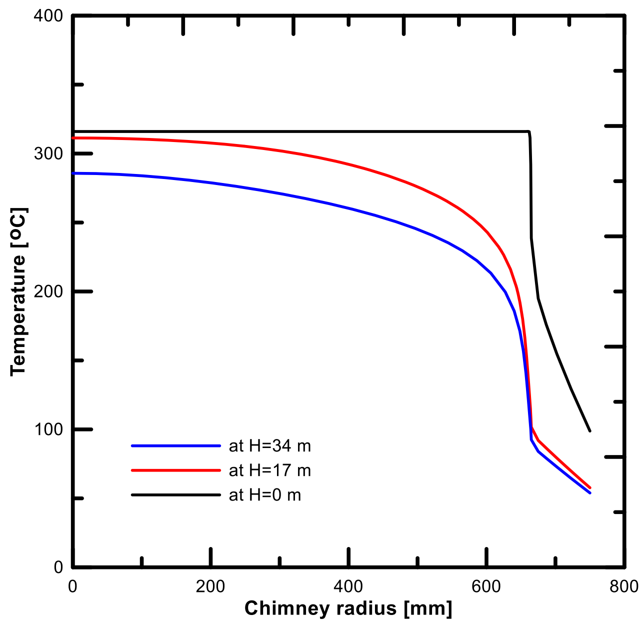
4.2. Comparison between analytical, numerical, and experimental results

The comparison between the analytical and numerical turbulence models was validated with empirical measurements from an industrial chimney. The dimensionless variables were determined for both analytical models; it was found that the Reynolds number result was relatively high, implying that the flow inside the chimney duct was turbulent. The dependence of the density ratio parameter and the dimensionless group parameters *Re* and *Pr* on the chimney duct inlet temperature is presented in supplementary materials Fig. S3. These analytical results were calculated to investigate the effect of inlet temperature on the natural convection behaviour. It was clear that the inlet flue gas temperature had a great effect on the dimensionless parameters. It was found that the density ratio, Reynolds and Prandtl numbers decreased as the inlet flue gas temperature increased. The Prandtl number is considered the most important heat transfer parameter, and controls the thermal boundary layer thickness of the momentum equation. The finding that $Pr < 1$ means that heat was quickly diffused compared with the velocity of the gas, and that the thickness of the

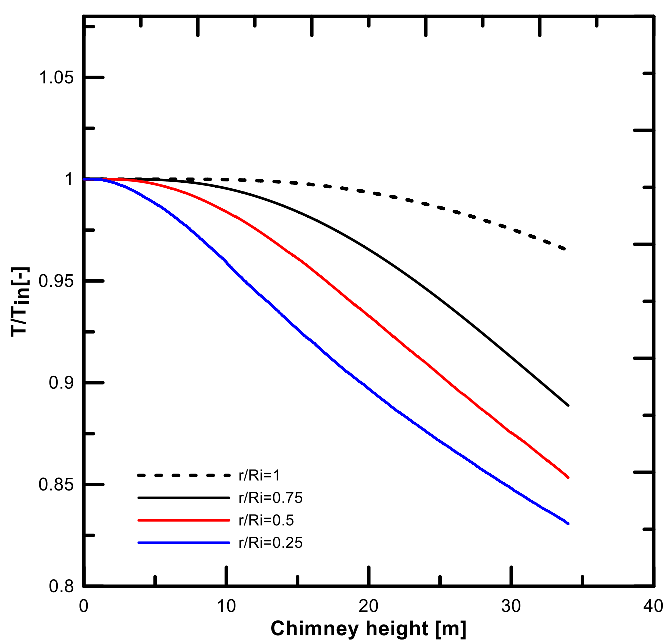


(a) Velocity profile. (b) Temperature profile.

Fig. 2. Velocity plot and temperature distribution along the chimney duct.



(a)



(b)

Fig. 3. The temperature distribution profiles: (a) a cross-sectional plot at different heights, and (b) along the height of the chimney.

thermal boundary layer was much larger than that of the momentum boundary layer. The Prandtl number can be defined as the ratio of momentum diffusivity to thermal diffusivity. Exhaust gases with low Prandtl numbers have high thermal conductivities, leading to faster heat diffusion than at higher values of *Pr*. Furthermore, the Reynolds number increased with temperature ratio, corresponding to higher heat transfer rates.

The outlet temperature of exhaust gases from the chimney is shown in Fig. 5. It was measured at two points at the inlet and outlet of the real chimney, using thermocouples. The comparison between both analytical and simulation models with the measured industrial chimney data illustrates that the modified lumped model is closer to the empirical data

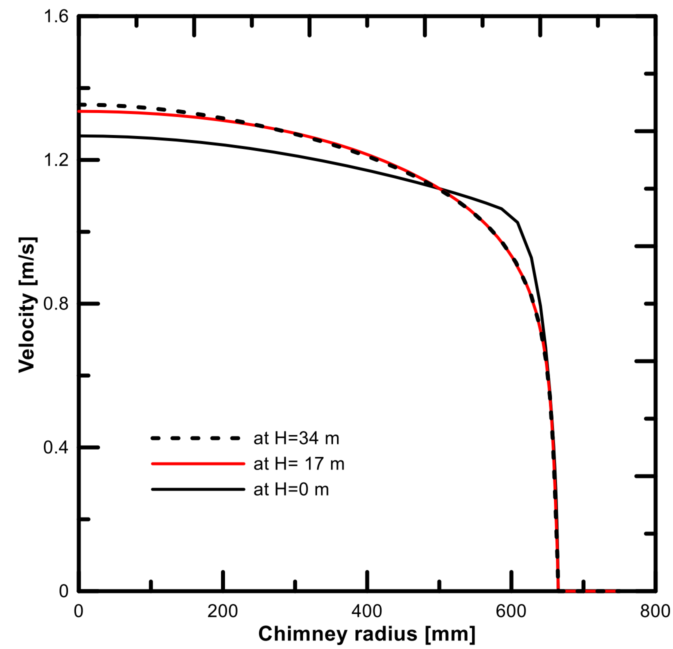


Fig. 4. Cross-sectional plot showing the velocity distribution at different heights.

than the *k-ω* and Cortés models. It is clear that the *k-ω* turbulence model can be used to predict the outlet temperature from industrial chimneys. The results demonstrate that the modified lumped model is better able to predict the temperature distribution because it takes into account the friction factor and the fouling resistance resulting from deposits formed by the outlet exhaust gases.

Fig. 6a shows the inner wall temperature distribution along the chimney height. The inner wall temperature was measured at two points. Furthermore, the inner wall temperature was determined by both analytical models. While the turbulence model was simulated and considered all of the heat transfer types. It can be concluded that the modified lumped model is able to give a good analytical prediction of

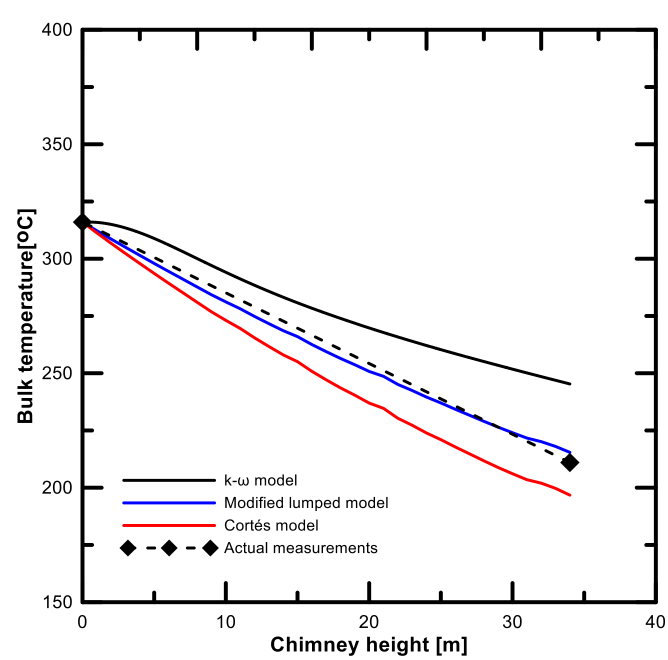


Fig. 5. Comparison between measured bulk temperature results with the analytical and turbulence models versus chimney height.

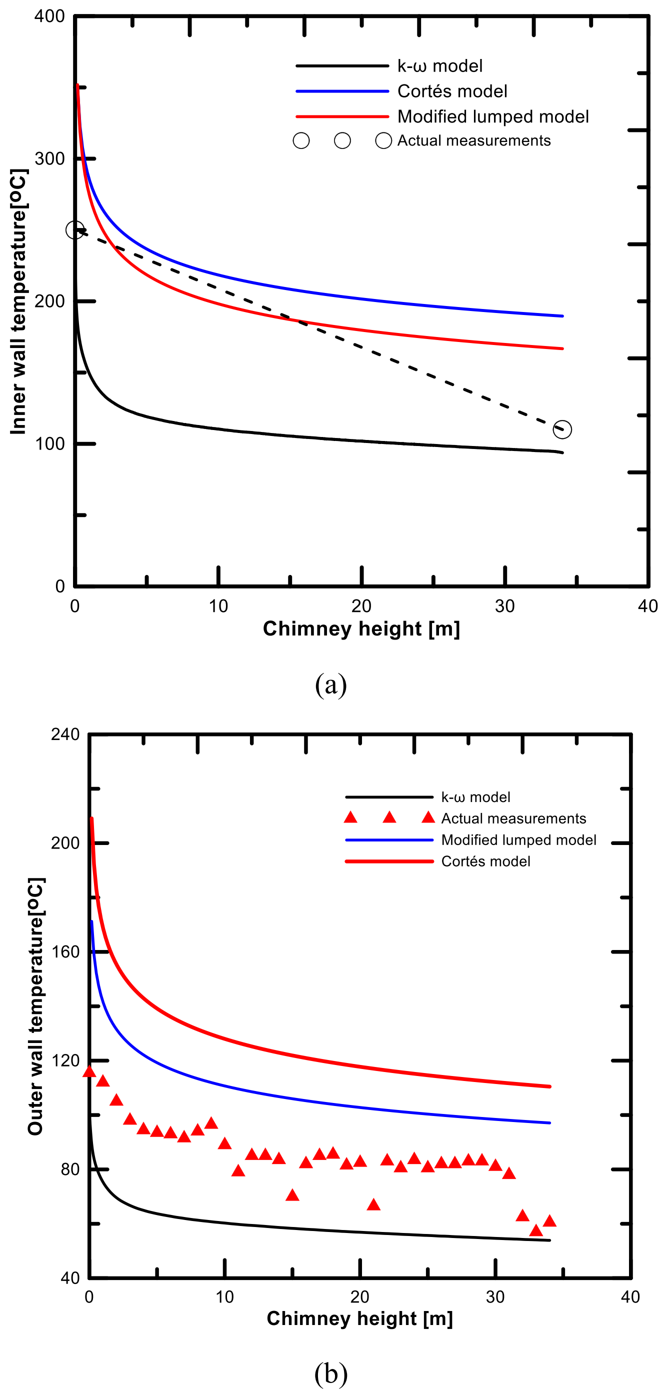


Fig. 6. Comparison between measured wall temperatures with the analytical and turbulence models: (a) inner wall temperature, and (b) outer wall temperature.

the temperature distribution results. Moreover, the *k-ω* turbulence model showed simulation results close to the empirical data. An optical laser pyrometer was used to measure the inside wall temperature at the chimney exit with an accuracy of ±2 °C. The temperature distribution on the inner wall is influenced by the flue gas thermodynamic properties as well as the heat transfer modes (conduction, convection, and radiation) along the chimney inner wall and outside air.

Fig. 6b shows the simulated results of the chimney outer wall temperature. The outside chimney wall is exposed to natural/forced convection heat loss. The outer wall temperature is an indicator of the performance of chimney wall insulation, the excellent performance of the chimney insulation can be defined by the larger temperature difference between inner gases temperature and outer wall temperature. These results indicate that the *k-ω* turbulence model and modified lumped model more accurately predicted the measured outside wall temperatures. The measured outside wall temperatures were taken at equal distances at the chimney outside wall and at the opposite side of the wind direction. The results show that the *k-ω* turbulence model can be adequately used to simulate and predict the temperature decay along the industrial chimney height. The temperature distribution inside and outside the chimney walls was investigated to evaluate the chimney design parameters. The simulated flue gases of the industrial chimney were modelled as a natural convection draft system depending on the buoyancy effect to remove gases to the outside air.

To clarify the differences between the analytical and turbulence models, the internal Nusselt number results were used as the main point of comparison. Fig. 7 shows the simulated results of internal Nusselt number for the *k-ω* turbulence model and modified lumped analytical models. The internal Nusselt number was estimated according to the modified lumped model. The Nusselt number was determined at various chimney vertical positions for the turbulence model and compared with the analytical model results. It was found that both the internal Nusselt number and the flue gas temperature decreased as the chimney height increased. The heat transfer rate increased from the chimney wall to reach the maximum value at the chimney centre. The *Nu* number results of the current simulation showed a good agreement with the natural convection finite element analysis of a uniform square cavity [32]. Furthermore, the simulated results of the *k-ω* turbulence model at different positions inside the chimney's 2D geometry showed that the Nusselt number increased with distance from the chimney wall.

The Nusselt number is an important dimensionless parameter that can provide key information about heat exchange as it is the ratio between the convective and conductive heat transfer. Also, the internal Nusselt number is expressed about the temperature gradient inside the chimney duct. The analytical models results showed a good agreement with the *k-ω* turbulence model. It can be concluded that the *k-ω* turbulence model can be also used to predict and evaluate the chimney design parameters, in addition to the analytical models. It will support industry/further research and to which direction.

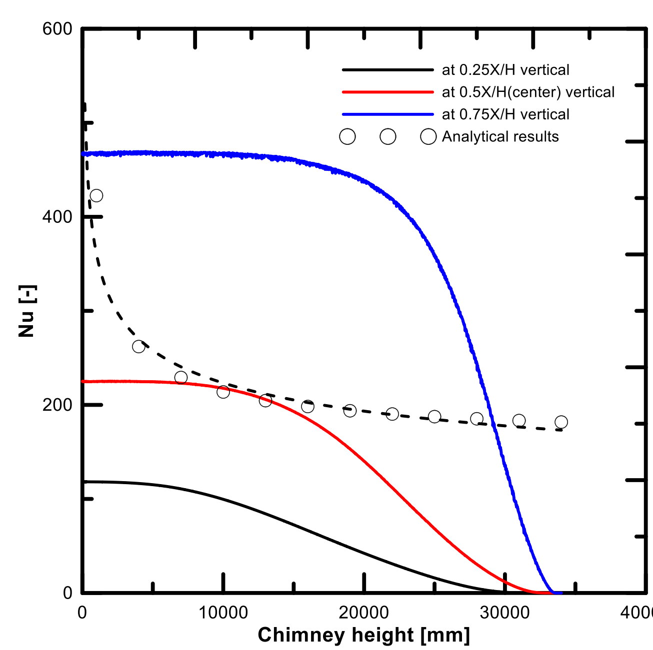


Fig. 7. Comparison between analytical and simulated Nusselt number results at different vertical chimney positions.

5. Conclusion

Chimney design is an important consideration in the operation of industrial furnaces. The design procedure requires the prediction of temperature decay along the chimney height. Two different analytical models were developed to obtain heat transfer distributions. Moreover, the chimney natural convection design was simulated using the CFD *k-ω* turbulence model. The analytical and numerical model results were validated with empirical measurements from a glass furnace chimney. Because it takes into account additional parameters such as the friction factor and fouling resistance calculations, the analytical results produced by the modified lumped model have better agreement with the empirical measurements than the Cortés model [1]. Furthermore, both analytical models were compared with the simulation results obtained from the *k-ω* turbulence model. The cross-sectional temperature and velocity profile distributions at different chimney height positions were investigated. The outlet temperature from the chimney exit was compared with the empirical data; it was found that the modified lumped model results were more accurate than those from the other analytical and computational models. The inner and outer wall temperature distributions were also compared with the empirically measured values. The inner and outer wall temperature decreased with the chimney height. It was found that the modified lumped model and the *k-ω* turbulence model were much near to the actual measurement data. The outer wall temperature of the industrial chimney can express the chimney wall insulation performance. It can be concluded that the modified lumped model can be used as an analytical model and the *k-ω* turbulence model as a numerical simulation model for efficient design of industrial chimneys, because they both showed good agreement with the empirical data. The main roles of a good simulation prediction of the temperature distribution along the industrial chimneys will participate in better environmental impact and reduction of energy consumption. Moreover, the simulation analysis of chimney or draft systems is the most significant issue to determine appropriate insulation materials and manage energy consumption that will achieve great cost savings and climate protection. Finally, it can be concluded that the *k-ω* turbulence model and the modified lumped model are both suitable for use by designers for the quick prediction and simulation of the heat transfer characteristics of chimneys.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data availability

The authors do not have permission to share data.

Acknowledgements

The authors would like to express their thanks to *Dr. Khaled Al-Farhany* (Department of Mechanical Engineering, University of Al-Qadisiyah, Ad-Diwaniyah 58001, Iraq) for his kind support to finalize this simulation.

Appendix A. Supplementary data

Supplementary data to this article can be found online at https://doi.org/10.1016/j.cscee.2022.100264.

References

[1] C. Corte's, A. Campo, Rapid computation of the exit temperature of hot combustion gases flowing inside chimneys, Appl. Therm. Eng. 19 (1999) 969–990.
[2] M.K. Bassiouny, A.A. Hussien, Mostafa El Shafie, Calculations of temperature decay for industrial chimney by using modified analytical model, J. Mech. Civil Eng. (IOSRJMCE) 11 (Issue 1) (2014) 13–24. Jan. 2014.
[3] A. Campo, L. Campo, An algebraic solution for a 2-D partial differential energy equation with a robin boundary condition generated from the simpler solution for adirichlet boundary condition, Comput. Math's Appl. 34 (1997) 101–114.
[4] W. Marefa, H. Cherkaoui, P. Crausse, H. Boisson, Heat transfer in the turbulent flow through a conduit for removal of combustion products, Build. Environ. 38 (2003) 763–770.
[5] C. Ming Chu, M. Mizanur Rahman, S. Kumaresan, Effect of cold inflow on chimney height of natural draft cooling towers, Nucl. Eng. Des. 249 (2011) 125–131.
[6] J. Zinoubi, R. Ben Maad, A. Belghith, Experimental study of the resulting flow of plume–thermo siphon interaction: application to chimney problems, Appl. Therm. Eng. 25 (2005) 533–544.
[7] S. Kazansky, V. Dubovsky, G. Ziskind, R. Letan, Chimney-enhanced natural convection from a vertical plate: experiments and numerical simulations, Int. J. Heat Mass Tran. 46 (2003) 497–512.
[8] A. Campo, A. Sanchez, Convective heat transport of high pressure flows inside active, thick walled tubes with isothermal outer surfaces: usage of Nusselt correlation equations for an inactive, thin walled tube, Appl. Therm. Eng. 18 (1998) 157–169.
[9] G. Verboom, H. Koten, Vortex excitation: three design rules tested on13 industrial chimneys, J. Wind Eng. Ind. Aerod. 98 (2010) 145–154.
[10] J. Kawecki, J. Zuranski, Cross-wind vibrations of steel chimneys-A new case history, J. Wind Eng. Ind. Aerod. 95 (2007) 1166–1175.
[11] A. Andreozzi, B. Buonomo, O. Manca, Numerical investigation of transient natural convection in a vertical channel-chimney system symmetrically heated at uniform heat flux, Int. J. Heat Mass Tran. (2012). Article in Press.
[12] G. Ruud, C. Beerkens, P. Hendrikus, Modeling of the aging of glass furnace regenerators, Glastech 65 (1992) 247–255.
[13] N. Hirata, J. Barrera, A unifying view for stack filter design based on graph search methods, Pattern Recogn. 38 (2005) 2088–2098.
[14] M. Krishna, K. Satyanarayana, S. Hussain, K. Hari, V. Kausik, stack-gas monitoring for nuclear power plants, Int. J. Eng. Sci. Adv. Technol. 2 (2012) 23–29.
[15] M. Bhatt, S. Vasanwala, Comparison of ultimate capacities of RC chimney sections under wind loading, Trends Sci. 19 (1) (2022) 1722.
[16] Mostafa El-Shafie, M. Khalil Bassiouny, Shinji Kambara, M. Samy, El-Behery, A. A. Hussien, Design of a heat recovery unit using exhaust gases for energy savings in an absorption air conditioning unit, Appl. Therm. Eng. 194 (2021), 117031.
[17] A. Eldesoukey, H. Hassan, Study of the performance of thermoelectric generator for waste heat recovery from chimney: impact of nanofluid-microchannel cooling system, Environ. Sci. Pollut. Res. (2022) 1–22, https://doi.org/10.1007/s11356-022-21015-1.
[18] O.O. Alfred, F.M. Tamer, O. Shinichi, H. Hassan, Comprehensive Review in Waste Heat Recovery in Different Thermal Energy-Consuming Processes Using Thermoelectric Generators for Electrical Power Generation. Processes, 2022.
[19] S. Churchill, H. Chu, Correlating equations for laminar and turbulent free convection from a vertical plate, Int. J. Heat Mass Tran. 18 (1975) 1323–1330.
[20] A. Zukauskas, J. Ziugzda, Heat Transfer of a Cylinder in Cross Flow, Hemisphere, New York, 1985.
[21] J. Lienhard, Heat Transfer Text Book, Phlogiston Press, 2005.
[22] B. Larock, R. Jeppson, G. Watters, Hydraulic Pipe Line, 2000. Boca Raton London New York Washington.
[23] E. Sparrow, J. Gregg, Laminar free convection heat transfer from the outer surface of a vertical circular Cylinder, ASME 78 (1956) 1823–1830.
[24] F. Incropera, D. DeWitt, in: Fundamentals of Heat and Mass Transfer, third ed., Wiley, New York, 1990.
[25] David C. Wilcox, Turbulence Modeling for CFD, Griffin Printing., Glendale, CA, 1994.
[26] J.H. Ferziger, M. Perić, Computational Methods for Fluid Dynamics, vol. 3, Springer, 2002.
[27] E. Spiegel, G. Veronis, On the Boussinesq approximation for a compressible fluid, Astrophys. J. 131 (1960) 442.
[28] S. Tieszen, A. Ooi, P. Durbin, M. Behnia, Modeling of natural convection heat transfer, Proceed. Summer Program (1998) 287.
[29] D.M. Aliaga, R. Feick, W.K. Brooks, M. Mery, R. Gers, J.F. Levi, C.P. Romero, Modified solar chimney configuration with a heat exchanger: experiment and CFD simulation, Therm. Sci. Eng. Prog. 22 (2021), 100850.
[30] A.J. Baker, Finite Element Computational Fluid Mechanics, Taylor & Francis US, 1983.
[31] W.M. Kays, Turbulent Prandtl number—where are we? J. Heat Tran. 116 (1994) 284–295.
[32] S. Roy, Tanmay Basak, Finite element analysis of natural convection flows in a square cavity with non-uniformly heated wall(s), Int. J. Eng. Sci. 43 (2005) 668–680.