

Implementation and Validation of k- ϵ Turbulence Model using C++ Open Source Code Software for Thermal Dispersion in Domestic Water Heater

Wisuwat Matchika and Chakrit Suvanjumrat*

ABSTRACT

The governing equations of the k- ϵ turbulence model were implemented using open source computational fluid dynamic (CFD) codes and open field operation and manipulation (OpenFOAM) software to simulate the thermal dispersion between the water flow and the heater in a domestic water heater. The C++ language was used to develop the CFD codes. The QUICK (Quadratic Upstream Interpolation for Convection Kinetics) scheme was applied with the k- ϵ turbulence model of the thermal dispersion and fluid flow. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm was used to solve the pressure-velocity coupling under turbulence flow. The solutions of the simulations were validated using the experimental results. The transient temperatures at the outlet port of the domestic water heater were obtained using a data acquisition (DAQ) device via K-type thermocouple lines. Twelve cases of the CFD were performed in this study and produced an average error of less than 2.13% when the CFD temperatures were compared with the experimental temperatures.

Keywords: computational fluid dynamics (CFD), turbulence model, open field operation and manipulation (OpenFOAM), water heater

INTRODUCTION

The thermal dispersion rate from a heater to the surrounding water flow in a domestic water heater is a key phenomenon that characterizes product efficiency. Many companies have attempted to develop innovative designs for the heater and chamber shape to obtain higher efficiency in water heaters. Such designs need to be tested before actual implementation and this is reliant on testing prototypes. It was found that these methods were usually time consuming and increased costs (Sezai *et al.*, 2005; Hegazy, 2007). Nowadays, computer-aided design (CAD) software is used to design models of domestic

water heaters and computational fluid dynamics (CFD) software—based on fluid dynamics theory—is used to analyze the thermal dispersion inside the domestic water heater. Many types of CFD software can simulate thermal dispersion behavior to specify the efficiency of water heaters (Soo Too *et al.*, 2004; Morrison *et al.*, 2005; Wang and Shan, 2006; Bhaumik, 2012). Unfortunately, these CFD software tools are commercial software which have an expensive licensing cost.

The open field operation and manipulation (OpenFOAM) software is open source software (OSS) based on the CFD codes. It is written in C++ language under the GNU general public license, which has no license cost (OpenFOAM, 2009). The

Department of Mechanical Engineering, Faculty of Engineering, Mahidol University, Nakhon Pathom 73170, Thailand.

* Corresponding author, e-mail: chakrit.suv@mahidol.ac.th

flow models and solving methods are not restricted and so can be applied to any fluid flow problems. However, the OSS can be difficult to use and does not support the simulation of complex shapes. This research aimed to develop and implement the C++ CFD codes to simulate the phenomenon of thermal dispersion between a heating element and the water flowing in a domestic water heater. The suitable k- ϵ turbulence model was selected from the OpenFOAM software. The three-dimensional volume inside a domestic water heater was created to simulate the thermal dispersion as water flowed in and out. Furthermore, the water heater testing was set to obtain the outlet temperatures for verifying the results of the CFD simulated model. The outcome of the simulations was the ready-made CFD codes to use for supporting the novel design of efficient water heaters in the future. Eventually, this research will result in the users of CFD software developing applications using open source software, which will save the purchasing cost of the commercial software.

MATERIALS AND METHODS

Mathematical model

The basic equations of fluid flow and heat transfer can be obtained by identifying the appropriate fundamental principles from the conservation laws of physics, which involve the conservation of mass, Newton's second law on the conservation of momentum and the first law of thermodynamics for the conservation of energy. The basic mass conservation equation for the thermal dispersion inside a cylindrical storage tank of a water heater is given by Equation 1:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U_i) = 0 \quad (1)$$

where $U_i \equiv [u, v, w]$, t is time, ρ is fluid density and u , v and w are the velocity in the x , y and z direction, respectively. Where the density ρ is constant for an incompressible fluid, Equation 1 can be rewritten as Equation 2:

$$\nabla \cdot U_i = 0 \quad (2)$$

or in the longhand form as shown in Equation 3:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (3)$$

The conservation of momentum can be written in compact form as Equation 4:

$$\rho \left(\frac{\partial U_i}{\partial t} + U_i \cdot \nabla U_i \right) = \nabla P_i + \nabla \tau_{ij} + S_M \quad (4)$$

where ∇P_i is the pressure gradient, $\nabla \tau_{ij}$ is the shear stress and S_M is the rate of increase due to sources.

The shear stress may be taken to be proportional to the time rate of strain for the Newtonian fluid. The shear stress component according to Newton's law of viscosity is written as Equation 5:

$$\tau_{ij} = \mu \left(\frac{\partial U_i}{\partial x_j} \right) + \left(\frac{\partial U_j}{\partial x_i} \right) \quad (5)$$

where x_i is denoted mutually perpendicular coordinate direction and μ is the dynamic viscosity.

On the basis of a consideration of the first law of thermodynamics, the energy equation can be derived as the rate of change of energy of a fluid particle which is equal to the rate of heat addition plus the rate of work done. The conservation of energy can be simplified by considering the fact that density is constant for incompressible flows and the kinetic energy is negligible. The energy equation can be expressed as Equation 6:

$$\rho \left(\frac{\partial i}{\partial t} + U_i \cdot \nabla i \right) = \text{div}(k \text{ grad } T) + \Phi \quad (6)$$

where i is the internal energy, T is the absolute temperature, k is the thermal conductivity of the fluid, Φ is the dissipation function representing the work done against viscous forces, which is irreversibly converted into internal energy and defined by Equation 7:

$$\Phi = \tau_{ij} \frac{\partial U_i}{\partial x_j} \quad (7)$$

The governing equations contain as a further unknown the viscous stress components, τ_{ij} . The most useful forms of the conservation equations

for fluid flows are obtained by introducing a suitable model for the viscous stresses, τ_{ij} . In three-dimensional flows, the local rate of deformation is composed of the linear deformation rate and the volumetric deformation rate. In a Newtonian fluid, the viscous stresses are proportional to the rates of deformation. The nine viscous stress components, of which six are independent, are shown in Equation 8:

$$\begin{aligned}\tau_{xx} &= 2\mu \frac{\partial u}{\partial x} & \tau_{yy} &= 2\mu \frac{\partial v}{\partial y} & \tau_{zz} &= 2\mu \frac{\partial w}{\partial z} \\ \tau_{xy} &= \tau_{yx} = \mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) & \tau_{xz} &= \tau_{zx} = \mu \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \\ \tau_{yz} &= \tau_{zy} = \mu \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)\end{aligned}\quad (8)$$

Substitution of Equation 8 into Equation 4 yields Equation 9 which is called the Navier-Stokes equation.

$$\rho \left(\frac{\partial U_i}{\partial t} + U_i \cdot \nabla U_i \right) = \nabla P_i + \text{div}(\mu \text{ grad } U_i) + S_M \quad (9)$$

The Newtonian model for viscous stresses for the internal energy equation, $i = cT$, in Equation 6 can be re-arranged into Equation 10:

$$\rho c \left(\frac{\partial T}{\partial t} + U_i \cdot \nabla T \right) = -\nabla(PU_i) + \Phi + \nabla \cdot (k \nabla T) + S_E \quad (10)$$

where c is the specific heat.

All the effects due to viscous stresses in this internal energy equation are described by the dissipation function Φ which, after considerable algebra, can be shown as Equation 11:

$$\begin{aligned}\Phi &= \mu \left\{ 2 \left[\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 + \left(\frac{\partial w}{\partial z} \right)^2 \right] + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 \right. \\ &\quad \left. + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)^2 \right\}\end{aligned}\quad (11)$$

The classical averaging method of the turbulence flow equations is the ensemble average, which produces the Reynolds Average Navier-Stokes equation (RANS). The time average properties of fluid flow as shown by Equation 12 were applied to the mass, momentum and energy

conservation equation (Versteeg and Malalasekera, 2007; Yeoh and Tu, 2010):

$$U_i = \bar{U}_i + U_i', \quad P_i = \bar{P}_i + P_i' \quad (12)$$

Often, only the mean flow is of interest, so the RANS equation for the conservation of mass is given by Equation 13:

$$\nabla \cdot (\rho \bar{U}_i) = 0 \quad (13)$$

and hence the conservation of momentum and energy are written as Equations 14 and 15, respectively:

$$\begin{aligned}&\rho \left(\frac{\partial \bar{U}_i}{\partial t} + \bar{U}_i \cdot \nabla \bar{U}_i \right) \\ &= -\nabla \bar{P}_i + \nabla \cdot \left((\mu - \mu_t) \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) - \frac{2}{3} (\mu - \mu_t) \delta_{ij} \right. \\ &\quad \left. \frac{\partial \bar{U}_j}{\partial x_j} \frac{\partial \bar{U}_i}{\partial t} - \frac{2}{3} \rho k \delta_{ij} \right) + \bar{S}_M\end{aligned}\quad (14)$$

$$\rho c \left(\frac{\partial T}{\partial t} + \bar{U}_i \cdot \nabla T \right) = -\nabla(P\bar{U}_i) + \Phi + \nabla \cdot (k \nabla T) + \bar{S}_E \quad (15)$$

where k is the turbulent kinetic energy, μ is the viscosity. μ_t is the turbulent viscosity which can be obtained using Equation 16:

$$\mu_t = \rho C_\mu \frac{k^2}{E} \quad (16)$$

where ε is the turbulent energy dissipation.

The values of k and ε were solved according to their respective transport equations obtained by Equations 17 and 18, respectively.

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho k U) = \nabla \cdot \left(\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right) + P_k - \rho \varepsilon \quad (17)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \varepsilon U) = \nabla \cdot \left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right) + C_{1\varepsilon} P_k \frac{\varepsilon}{k} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (18)$$

where P_k is the positive source term defined by Equation 19:

$$P_k = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial U_k}{\partial x_k} \right) \frac{\partial U_i}{\partial x_j} - \frac{2}{3} \rho k \frac{\partial U_j}{\partial x_j} \quad (19)$$

Launder and Spalding (1974) specified the constants for the k - ε model through comprehensive data fitting for a wide range of turbulence flows at $C_\mu = 0.09$, $\sigma^k = 1.00$, $\sigma_\varepsilon = 1.30$, $C_{1\varepsilon} = 1.44$ and $C_{2\varepsilon} = 1.92$, respectively.

Finite volume discretization

The physical mathematical model must be manipulated into a suitable form for the computer calculations. The main task of the discretization approach is to convert the partial differential equations into a set of discrete algebraic equation. This process was facilitated by using the governing equations which are written in terms of the general transport equation for property ϕ (Versteeg and Malalasekera, 2007) as shown by Equation 20:

$$\underbrace{\frac{\partial(\rho\phi)}{\partial t}}_{\text{Time Derivative}} + \underbrace{\nabla \cdot \rho U_i \phi}_{\text{Convection Term}} = \underbrace{\nabla \cdot \Gamma \nabla \phi}_{\text{Diffusion Term}} + \underbrace{S_\phi}_{\text{Source Term}} \quad (20)$$

where ϕ is the dependent variable, U_i is the vector of velocity, Γ is the diffusion coefficient and S_ϕ is the source term.

The corresponding values of ϕ , Γ and S_ϕ for the mass, momentum, energy and k- ϵ equation of the general transport equations are summarized in Table 1.

The implementation of the finite volume method (FVM) in this study used cells with an arbitrary number of faces as shown in Figure 1. Point P is in the center of a cell of interest, which is bounded by the triangular flat faces that are shared with the neighboring cells. The vector S_i^f is normal

to the flat faces and pointing outward from the cell. The vector $\overline{PN}_i$ connects the computational point, P , with its neighbor points, N_i (Chaichanasiri and Suvanjumrat, 2013).

The finite volume discretization is formulated by integrating over the interested control volume ($\forall_p$) and time and is written as Equation 21:

$$\int_t^{t+\Delta t} \left(\int \frac{\partial \rho \phi}{\partial t} d\forall_p + \int \nabla \cdot (\rho U \phi) d\forall_p \right) dt = \int_t^{t+\Delta t} \left(\int \nabla \cdot (\rho \Gamma \nabla \phi) d\forall_p + \int S d\forall_p \right) dt \quad (21)$$

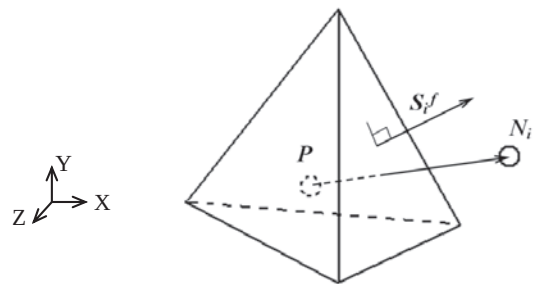


Figure 1 Typical space domain of discretization of FVM. P = Center point of a cell of interest; N_i = Center point of neighboring cells; S_i^f = Cell face vector.

Table 1 Generic form of the single-phase flow differential equation.

Conservative Equation	ϕ	Γ	S_ϕ
Mass	1	0	0
Momentum	U_i	μ	$-\frac{1}{\rho} \nabla p + S_U$
Energy	T	$\frac{k}{C}$	$\frac{1}{C} (-\nabla P U_i + \phi + S_E)$
Kinetic energy	k	$\mu + \frac{\mu_t}{\sigma_k}$	$P_k - \rho \epsilon$
Dissipated energy rate	ϵ	$\mu + \frac{\mu_t}{\sigma_\epsilon}$	$C_{1\epsilon} P_k \frac{\epsilon}{k} - C_{2\epsilon} \rho \frac{\epsilon^2}{k}$

ϕ = Dependent variable; ρ = Density; U = Vector of velocity; Γ = Diffusion coefficient; S = Source term; k = Turbulent kinetic energy; ϵ = Rate of dissipation of turbulent energy; μ_t = Turbulent viscosity; P_k = Positive source term.

The linear variation of ϕ_p of a cell of interest within a time step was used in this study based on the Euler implicit time differencing scheme and is given by Equation 22:

$$\int \frac{\partial \rho \phi_p}{\partial t} d\forall_p = \frac{(\rho \phi_p)^{t+\Delta t} - (\rho \phi_p)^t}{\Delta t} \forall_p \quad (22)$$

where ϕ_p is the general variable at point P of the interested cell and Δt is the time step.

Then, Gauss's theorem was applied to the convection term for transforming the volume integral to a surface integral as shown in Equation 23:

$$\int \nabla \cdot (\rho U \phi) d\forall_p = \sum_{i=1}^n (\rho U) \phi_f S_i^f \quad (23)$$

where ϕ_f is the general variable on flat faces of the interested cell and S_i^f is the surface area of the interested cell.

The QUICK (Quadratic Upstream Interpolation for Convective Kinetics) scheme was employed in this study to determine on the face from the direction of flow as written by Equation 24:

$$\phi_f = \frac{6}{8} \phi_{N_{i-1}} + \frac{3}{8} \phi_{N_i} - \frac{1}{8} \phi_{N_{i-2}} \quad (24)$$

The diffusion term was integrated over a cell and reduced to Equations 25 and 26:

$$\int \nabla \cdot (\rho \Gamma \nabla \phi) d\forall_p = \sum_{i=1}^n (\rho \Gamma) \nabla \phi_f S_i^f \quad (25)$$

when

$$\phi_f S_i^f = \frac{\phi_{N_i} - \phi_p}{|\overline{PN_i}|} |S_i^f| \quad (26)$$

where $\overline{PN_i}$ is any length between the center of the cell of interest and the center of a neighboring cell.

The source terms are defined as Equation 27:

$$\int S d\forall_p = S \forall_p = (S_U + S_p \phi_p) \forall_p \quad (27)$$

where S_U is the explicit source term and S_p is the implicit source term.

Using Equations 22, 23, 25 and 27, Equation 21 was reformulated as Equation 28:

$$\begin{aligned} & \frac{(\rho \phi_p)^{t+\Delta t} - (\rho \phi_p)^t}{\Delta t} \forall_p + \sum_{i=1}^n (\rho U_i) \phi_f S_i^f \\ &= \sum_{i=1}^n (\rho \Gamma) \phi_f S_i^f + S_U \forall_p + S_p \phi_p \forall_p \end{aligned} \quad (28)$$

The discretized equations yielded an equation for ϕ in each cell written in terms of neighboring cells as described by Equations 29 and 30:

$$\alpha_p (\phi_p)^{t+\Delta t} = \sum_{i=1}^n \alpha_{N_i} (\phi_{N_i})^{t+\Delta t} + S_U \quad (29)$$

and

$$\alpha_p = \sum_{i=1}^n \alpha_{N_i} - S_p \quad (30)$$

where ϕ_{N_i} is the general variable of neighboring cells, α_p is the central coefficient and α_{N_i} is the neighboring coefficient.

A problem associated with the calculation of velocity and pressure on co-located cells can be solved by interpolating the discretized equation onto the cell faces and applying the continuity constraint to the cell face velocities (OpenFOAM, 2009). The discretized equation was modified by assuming all the source terms, except the pressure gradient, are contained in the $H(U)$. The momentum equation was written in a semi-discretized form as Equations 31 and 32:

$$(U)^{t+\Delta t} = \frac{H(U)}{\alpha_p} - \frac{1}{\alpha_p} \Delta P \quad (31)$$

where

$$H(U) = \sum_{i=1}^n \alpha_{N_i} (U_{N_i}) \frac{U^t}{\Delta t} \quad (32)$$

For simplicity, the superscript $t + \Delta t$ of Equation 31 can be dropped and the velocity value at the cell face was obtained by interpolation of the semi-discretized form and is given by Equation 33:

$$U_f = \left(\frac{H(U)}{\alpha_p} \right)_f - \left(\frac{1}{\alpha_p} \nabla P \right)_f \quad (33)$$

The discretized form of the continuity equation was obtained from Equation 34:

$$\nabla \cdot U_i = \sum_{i=1}^n S_i U_f = 0 \quad (34)$$

Substitution of Equation 33 into Equation 34, provided the pressure equation as Equation 35:

$$\nabla \cdot \left(\frac{\nabla P}{\alpha_p} \right) = \nabla \cdot \left(\frac{H(U)}{\alpha_p} \right) = \sum_{i=1}^n S_i \left(\frac{H(U)}{\alpha_p} \right)_f \quad (35)$$

Fluxes are obtained by Equation 36:

$$F = n_i \cdot U_f \quad (36)$$

The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm for solving the couple pressure-velocity in the transient flow calculations developed by Patankar and Spalding, (1972) was used in this study. It has the following steps:

(a) To initiate the SIMPLE calculation, a pressure field was guessed (p'). From the momentum equation, using the guessed pressure field, the velocity field was solved. The equation was under-relaxed using Equation 37:

$$\frac{\alpha_p}{\alpha} \phi_p + \sum_{i=1}^n \alpha_{Ni} \phi_{Ni} = S_U + \frac{1-\alpha}{\alpha} \alpha_p \phi_p^\circ \quad (37)$$

where ϕ_p° is the value from the previous iteration, α is the under-relaxation factor ($0 < \alpha < 1$).

(b) The pressure equation was solved to obtain the new pressure distribution.

(c) A new set of fluxes was calculated using Equation 36. The new pressure field included both the pressure error and convection-diffusion error. In order to obtain a better approximation of the pressure field, the coefficients of $H(U)$ were recalculated with the new set of conservative fluxes.

(d) The pressure solution was under-relaxed to include the velocity part of the error as Equation 38:

$$p = p' + \alpha_p(p'' - p') \quad (38)$$

where p is the approximation of the pressure field which will be used in the next momentum predictor, p' is the pressure field used in the momentum predictor, p'' is the solution of the pressure equation, α_p is the pressure under-relaxation factor.

This SIMPLE algorithm loop was continued until convergence.

Development and implementation

In this study, the thermal dispersion simulation in the domestic water heater was performed using OpenFOAM. The C++ language of CFD was written into two categories of applications: solvers that were each designed to solve a particular problem in continuum mechanics; and utilities that were designed to perform tasks that involved data manipulation. The domestic water heater consisted of a cylindrical storage tank and a rectangular heater. The cylindrical storage tank had a height and diameter of 124 mm and 51 mm, respectively. The inlet and outlet ports of heater tank had a diameter of 13 mm and 7 mm, respectively. The heater was rectangular in shape with width, length and thickness of 16.8, 111.5 and 3.8 mm, respectively. The rectangular heater was assembled in the cylindrical storage tank as shown by the detailed drawing in Figure 2.

The space domain for investigating the water flow and thermal dispersion was the volume inside the water heater. The third-party OSS, Discretizer, was imported to build the cell structure. This volume was divided by non-uniform cells based on the cell- or grid-independent study to check the mesh or cell quality with respect to the accuracy of the CFD solutions. The appropriate total number of cells for an investigated volume was 174,315 as shown in Figure 3. The water properties are shown in Table 2. The no-slip boundary condition was treated at the sidewall of the cylindrical tank and silicon heater because the turbulence boundary layer or Y^+ was less than 5 (Versteeg and Malalasekera, 2007). Consequently, the velocity at boundaries was zero.

The initial conditions of the study were the water flow rates at room temperature (297 K) and the temperatures of the heater. Four water flow rates and three heater temperatures were used to implement the CFD codes. The different initial velocities of water flow were defined at the inlet port of the CFD model. The pressure at the outlet port of the storage tank was defined as atmospheric

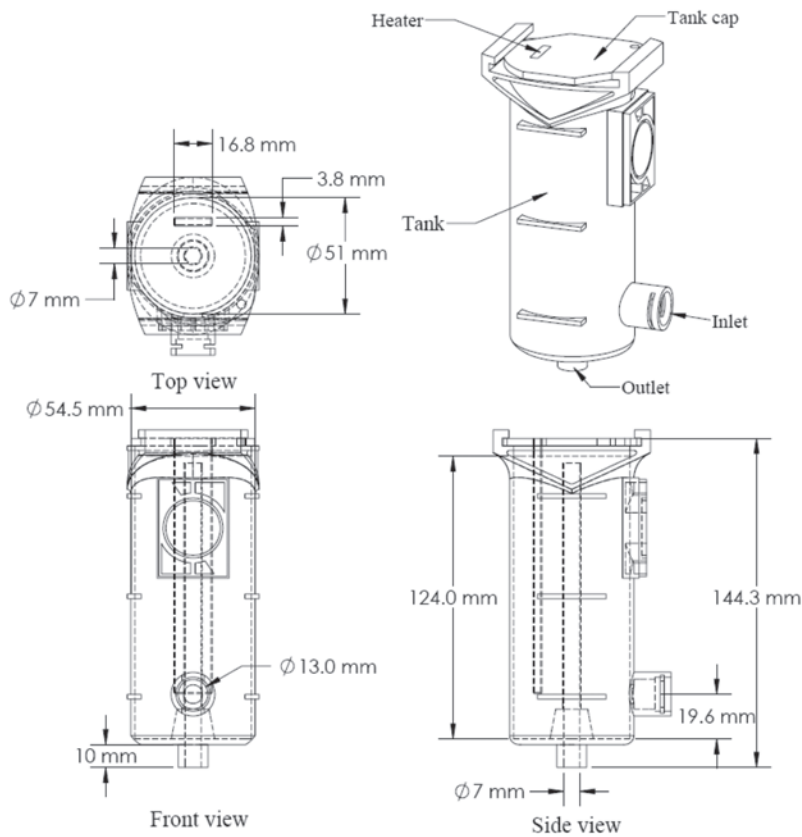


Figure 2 Tank dimensions of the domestic water heater.

pressure. The twelve cases of thermal dispersion studied are described in Table 3.

Validation setting

The experiment was set up to verify the CFD models. The rectangular heater and cylindrical storage tank in Figure 4 were selected as a temporary water heater chamber. It was assembled within the testing bench of a domestic water heater which is shown in Figure 5. The K-type thermocouples were inserted at the inlet and outlet ports of the tank to record the water temperature. The water flow rates were adjusted using a glove valve and levels were shown using a flow meter which distributed the water into a cylindrical tank. The hot water flow out of the tank after heating was cooled down by a cooling tower which was installed outside the testing room.

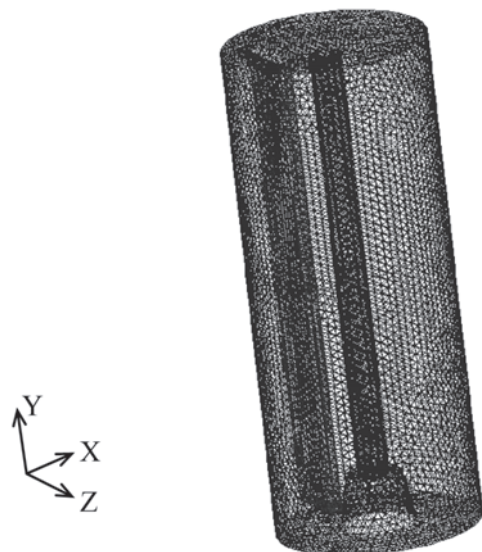


Figure 3 Cell structure for a water heater including the cylindrical storage tank and the rectangular heater.

The procedure of an experiment started by opening the inlet valve to allow water to flow at the adjusted flow rate into the cylindrical storage tank until it came out from the outlet port of the tank. The power supply of the heater was turned on to heat the flowing water and then the

Table 2 Properties of fluid (Cengel and Cimbala, 2010).

Fluid type	k (W/(m.K))	ρ (kg.m ⁻³)	C_p (J/(kg.K))	ν (m ² .s ⁻¹)
Water	0.61	1,000	4.1×10^3	9×10^{-7}

k = Thermal conductivity, ρ = Fluid density, C_p = Specific heat capacity, ν = Kinematic viscosity.

Table 3 Computational fluid dynamics (CFD) cases studied.

CFD case	Inlet flow rate (L.min ⁻¹)				Temperature of heater (K)		
	2.0	3.0	4.0	5.0	364	371	375
1	⊙				⊙		
2		⊙			⊙		
3			⊙		⊙		
4				⊙	⊙		
5	⊙					⊙	
6		⊙				⊙	
7			⊙			⊙	
8				⊙		⊙	
9	⊙						⊙
10		⊙					⊙
11			⊙				⊙
12				⊙			⊙

The symbol ⊙ indicates the characteristic set in the CFD models.

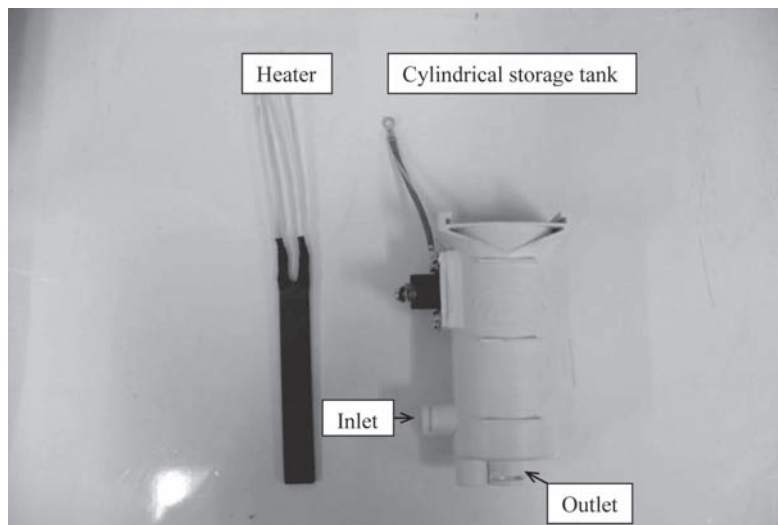


Figure 4 Layout of cylindrical storage tank with silicon heater.

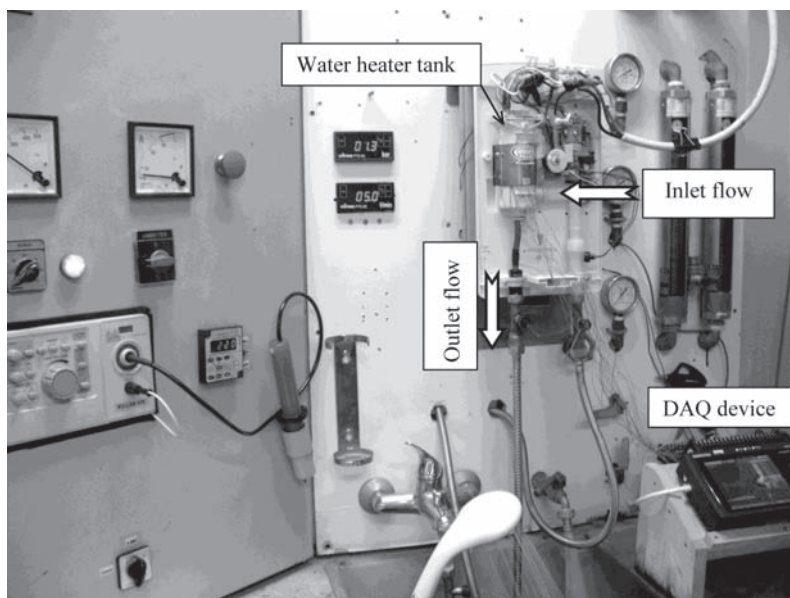


Figure 5 Apparatus for the thermal dispersion experiment.

temperatures were recorded using a GRAPHTEC data acquisition (DAQ) device model GL820 via the thermocouple line. Temperatures were sampled every 2 s at the outlet water port over a 240 sec period.

RESULTS AND DISCUSSION

The velocity profiles inside the water heater tank are illustrated clearly in Figure 6 where the symmetry plane shows detail at the inlet and outlet ports of the tank (Figure 6).

Magnitudes of calculated velocities in the symmetry plane of the storage tank are shown by gray contours. Maximum velocity magnitudes of water flow inside the tank are represented in white, while minimum velocity magnitudes of water flow inside the tank are shown in black. Arrows indicate flow directions of the water inside the tank; therefore, the inlet flow occurred on the left hand side of the symmetry plane at the inlet port. Outlet flow occurs at the outlet port (the vertical pipe) inside the tank; water flows in and out at the top and bottom of the vertical pipe, respectively.

The computational fluid dynamic results of the thermal dispersion in the water heater tank are illustrated by the gray contours of temperature on the section view of the cell structure. Temperature increases from cool to hot as shown from black to white, respectively.

The temperatures of the heater transferred to the flowing water were investigated using the CFD results. Figure 7 shows the sequences of the thermal dispersion inside the tank with the heater temperature at 364 K. Inlet water flow rates of 2, 3, 4, and 5 L.min⁻¹ with a constant initial temperature of 297 K effected the thermal dispersion inside the tank as shown in Figures 5A, 5B, 5C and 5D, respectively. The outlet temperatures of water were reduced when the inlet water flow rates were increased. Figures 8 and 9 show the temperature contours of the flowing water passing through the heater at temperatures of 371 and 375 K, respectively. The temperature contours appear to be similar to those in Figure 7 but the tank with the higher temperature of heater had a greater number of higher temperature contours inside the tank.

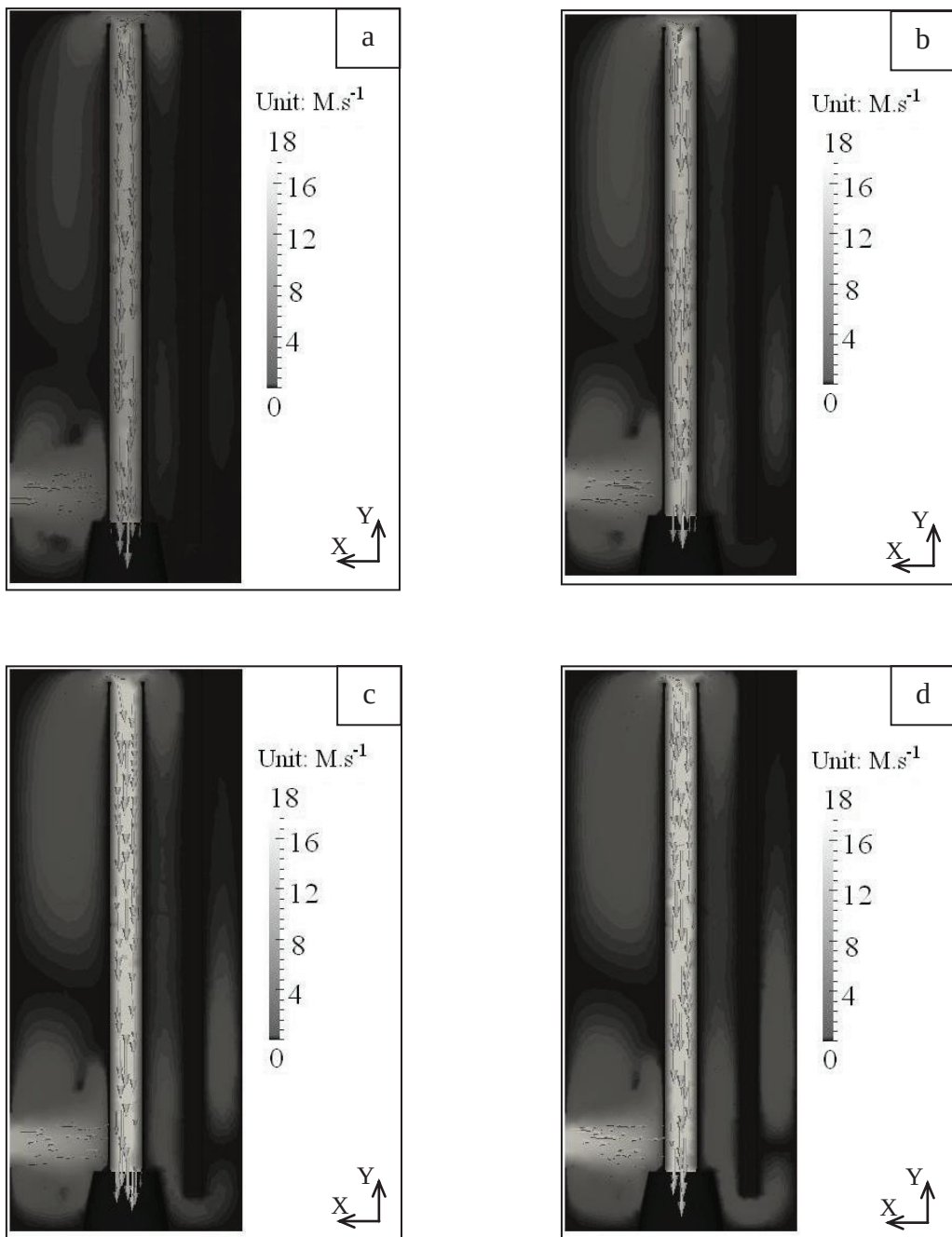


Figure 6 Velocity distribution of water flow inside the tank with a water inlet flow-rate of: (a) 2 $L.min^{-1}$; (b) 3 $L.min^{-1}$; (c) 4 $L.min^{-1}$; and (d) 5 $L.min^{-1}$. The X and Y axes show the tank directions.

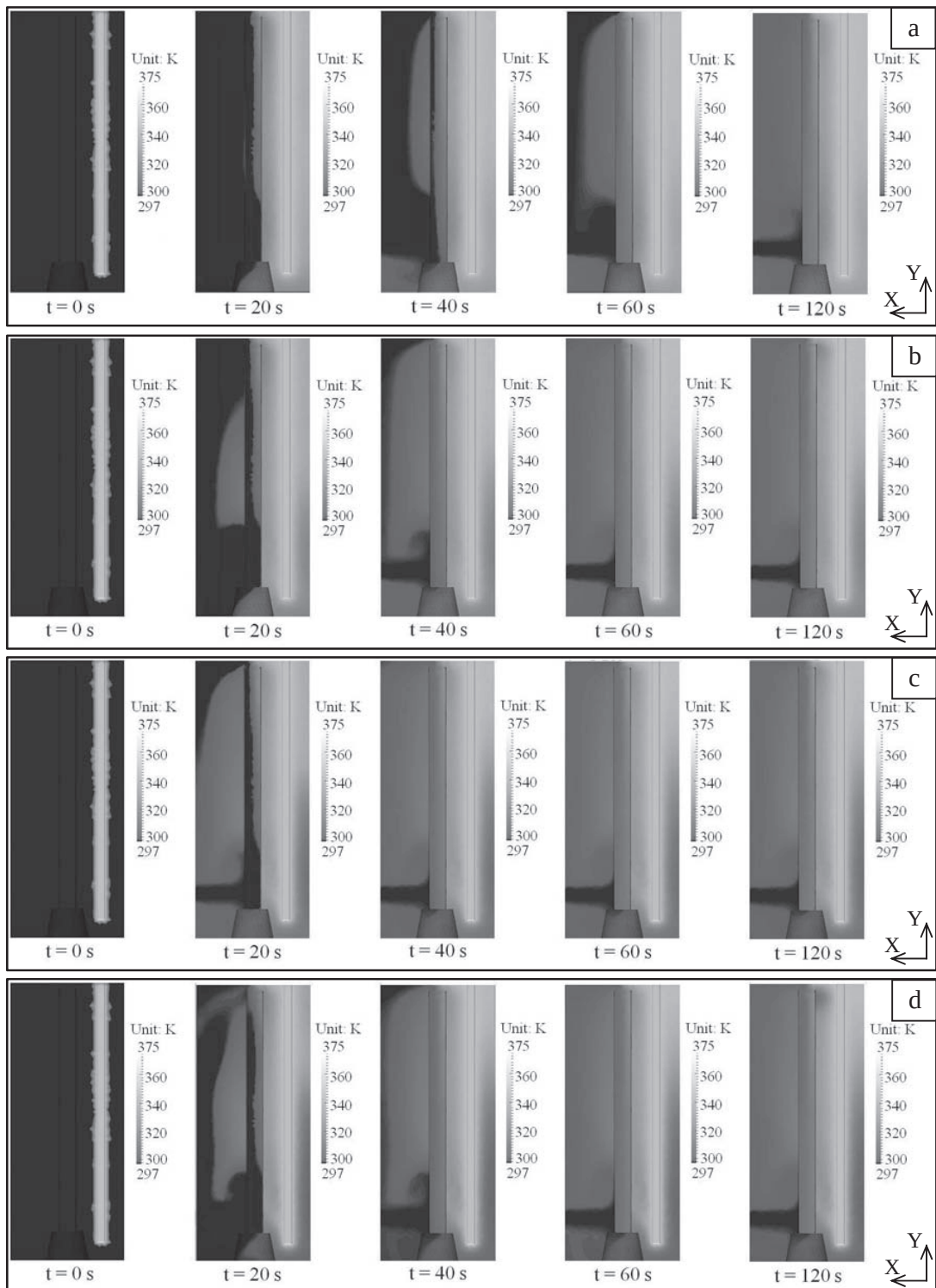


Figure 7 Sequential images of thermal dispersion of inlet water passing through the 364 K heater at a rate of: (a) 2 L.min⁻¹; (b) 3 L.min⁻¹; (c) 4 L.min⁻¹; and (d) 5 L.min⁻¹. The X and Y axes show the tank directions.

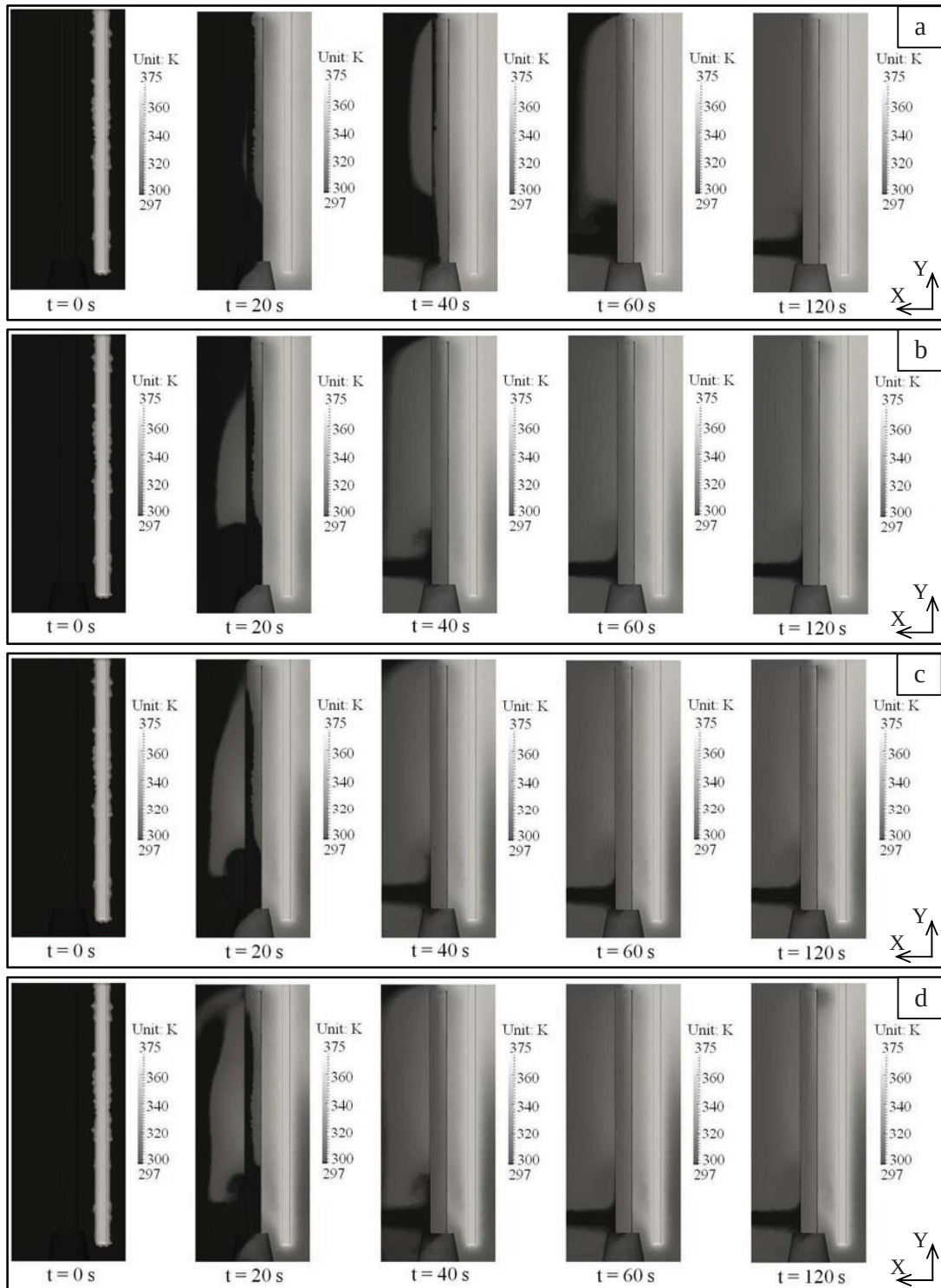


Figure 8 Sequential images of thermal dispersion of inlet water passing through the 371 K heater at a rate of: (a) 2 L.min⁻¹; (b) 3 L.min⁻¹; (c) 4 L.min⁻¹; and (d) 5 L.min⁻¹. The X and Y axes show the tank directions.

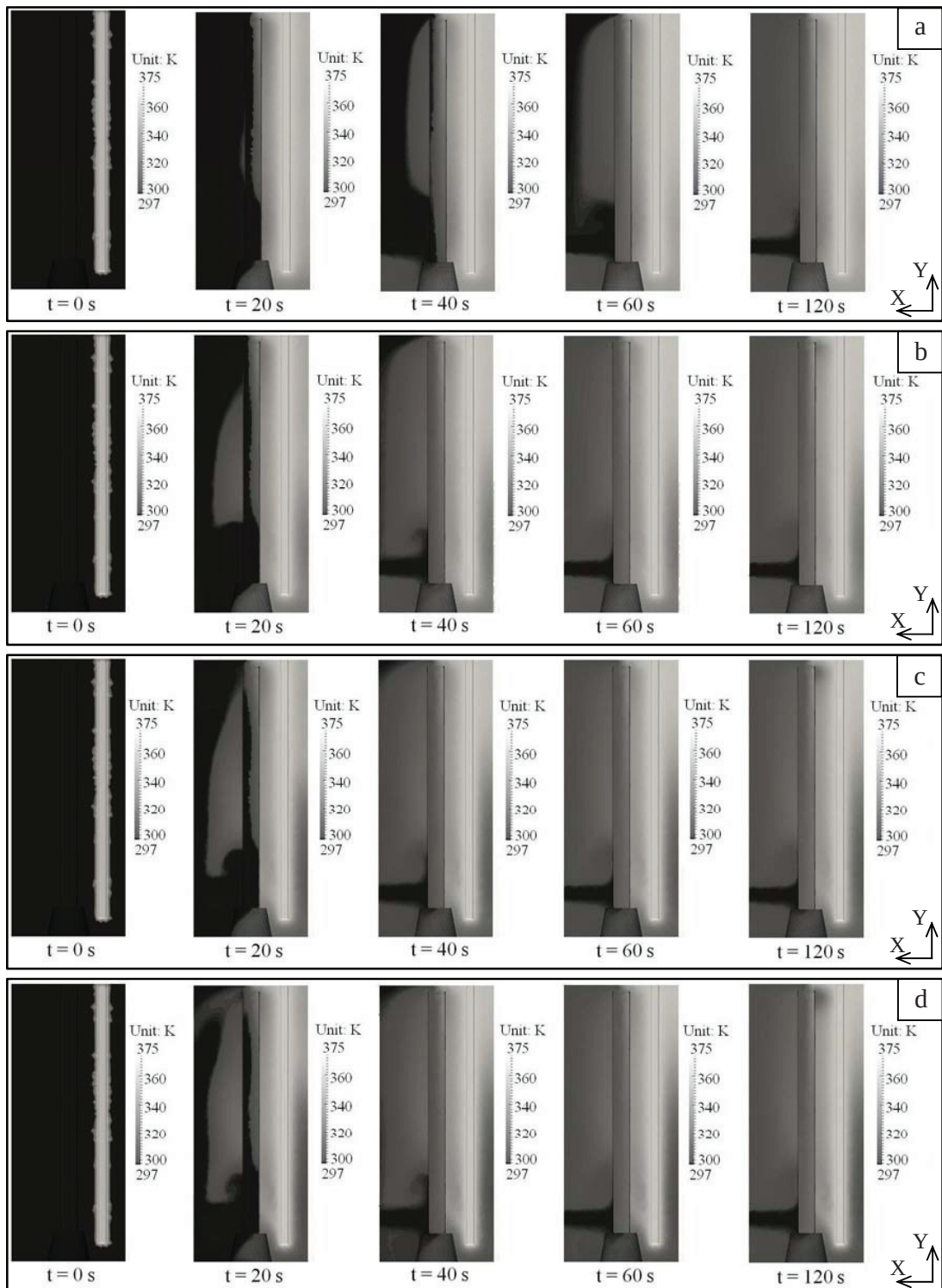


Figure 9 Sequential images of thermal dispersion of inlet water passing through the 375 K heater at a rate of: (a) 2 L.min⁻¹; (b) 3 L.min⁻¹; (c) 4 L.min⁻¹; and (d) 5 L.min⁻¹. The X and Y axes show the tank directions.

The temperature results of CFD were extracted from the cell having the same position as the thermocouple. Comparisons between the CFD and experimental temperatures at the outlet port for the inlet water flow rates of 2, 3, 4, and 5 L.min⁻¹ passing through the heater which had a temperature of 364 K are shown by the graphs in Figure 10. The temperatures of CFD and experimental results were plotted by dashed line and solid line, respectively. The comparison of variable temperatures throughout the experimental period of 240 s showed good correlations between the CFD and the experimental results. The comparisons of the temperature distribution between CFD and experiments for the 371 and 375 K were also correlated as shown in Figures 11 and 12, respectively. The temperatures of thermal dispersion increased at a rapid rate when the heater was adjusted to temperatures of 371 and 375 K.

The mean percentage error (MPE) method was used to compute the average percentage errors which was used to forecast CFD differences from

the experiment. The MPE of the temperature of all 12 cases which was measured at the outlet of the cylindrical tank throughout the 240 s period are shown in Table 4.

CONCLUSION

The developed code using the C++ language in the open source software Open FOAM was utilized for the simulation of thermal dispersion and water flow in a domestic water heater. The k- ϵ turbulence model was implemented and found to be suitable for simulation of the heat exchange between the heater and water flow inside the small tank of the water heater. Tetrahedral cells were used successfully to develop and build the finite volume model of the complicated shape of the water heater tank. The QUICK (Quadratic Upstream Interpolation for Convection Kinetics) scheme was applied successfully to reduce the wiggle results from the effect of water flow. The SIMPLE (Semi-Implicit Method

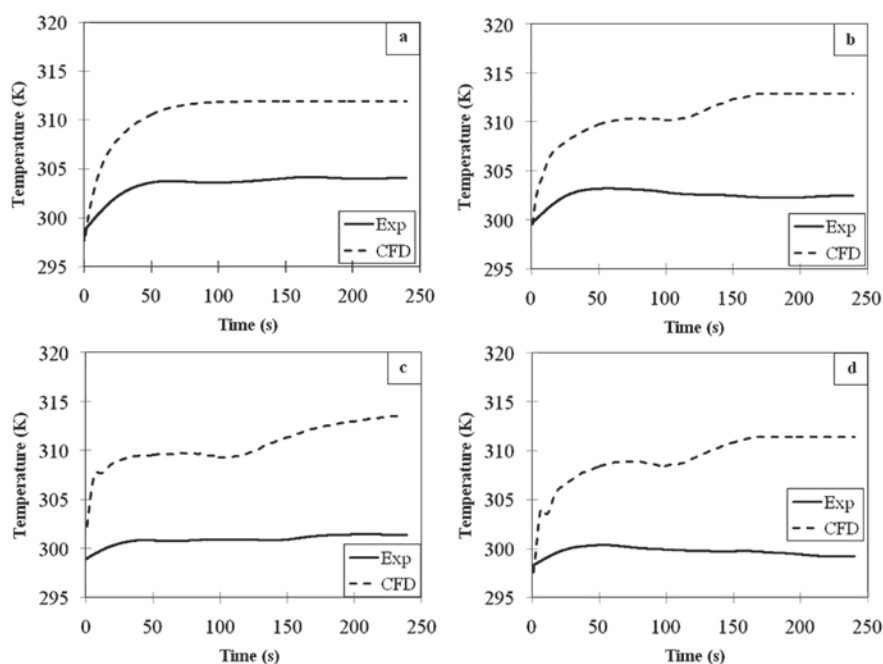


Figure 10 Comparison between the experiment (Exp) and the computational fluid dynamics (CFD) temperatures at outlet port of a water heater with 364 K heater temperature over a period of 240 s at a flow rate of: (a) 2 L.min⁻¹; (b) 3 L.min⁻¹; (c) 4 L.min⁻¹; and (d) 5 L.min⁻¹.

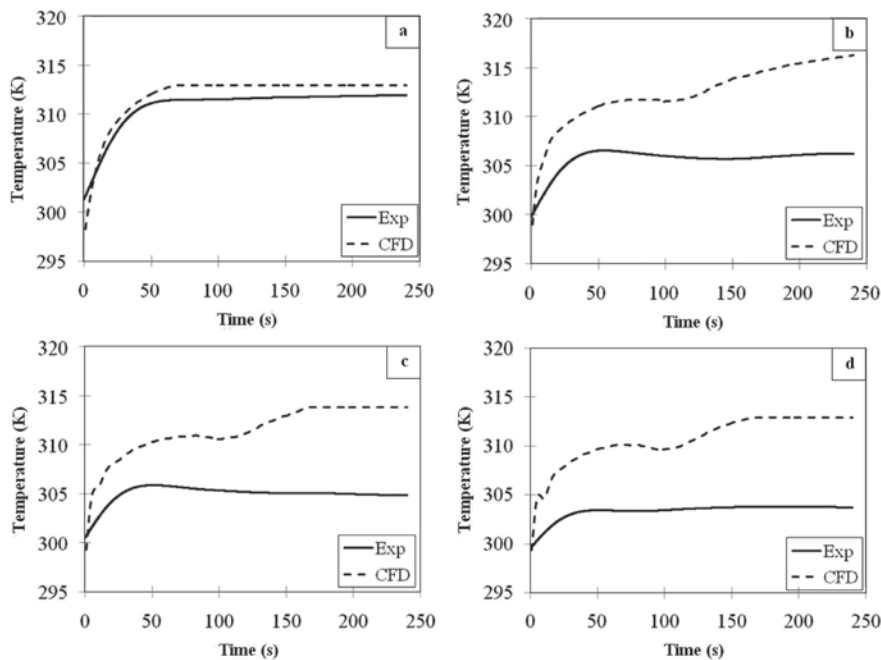


Figure 11 Comparison between the experiment (Exp) and the computational fluid dynamics (CFD) temperatures at outlet port of a water heater with 371 K heater temperature over a period of 240 s at a flow rate of: (a) 2 L.min⁻¹; (b) 3 L.min⁻¹; (c) 4 L.min⁻¹; and (d) 5 L.min⁻¹.

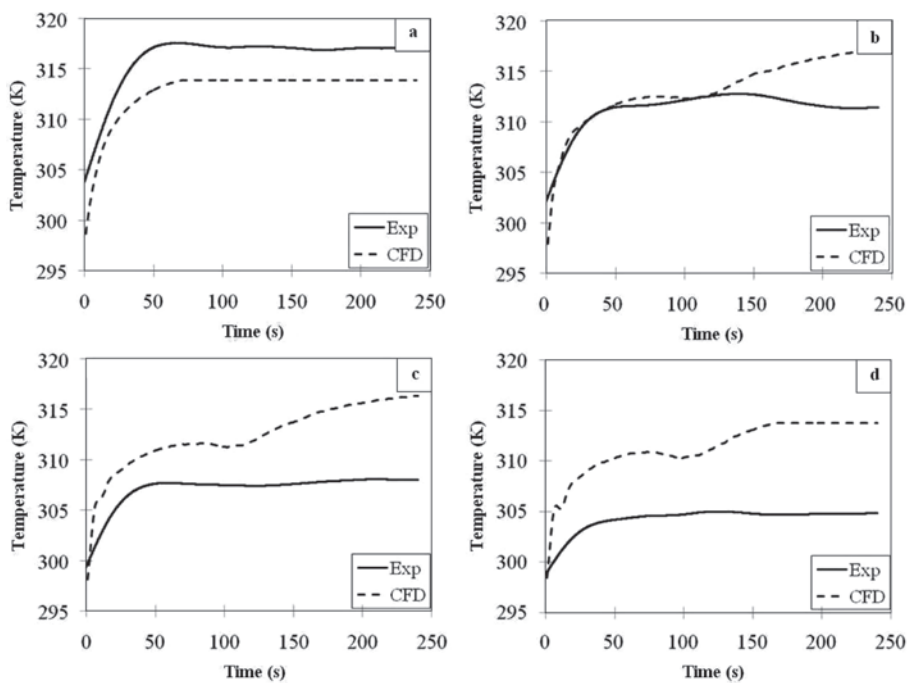


Figure 12 Comparison between the experiment (Exp) and the computational fluid dynamics (CFD) temperatures at outlet port of a water heater with 375 K heater temperature over a period of 240 s at a flow rate of: (a) 2 L.min⁻¹; (b) 3 L.min⁻¹; (c) 4 L.min⁻¹; and (d) 5 L.min⁻¹.

Table 4 Mean percentage error of the temperature comparison between the calculated computational fluid dynamics (CFD) and experiment results.

Heater temp. (K)	Mean percentage error of CFD compared with experiment (%)			
	Flow rate at 2 L.min ⁻¹	Flow rate at 3 L.min ⁻¹	Flow rate at 4 L.min ⁻¹	Flow rate at 5 L.min ⁻¹
364	2.47	2.91	3.47	3.43
371	0.56	2.43	2.33	2.45
375	1.01	0.67	1.76	2.06

for Pressure-Linked Equations) algorithm was used successfully to solve the pressure-velocity coupling in the water flow. The solutions of the simulations were validated using the 12 cases of the experimental results. The CFD temperatures of water flow at the outlet port of the tank had values greater than those from the experiments, with an average error of 2.13%. Therefore, the CFD codes developed in this research provided good estimates and can be used for the design of novel domestic water heaters tanks with regard to changes in the tank and heater shape.

ACKNOWLEDGEMENTS

The authors wish to thank STIEBEL ELTRON ASIA Ltd. for the use of the experimental devices.

LITERATURE CITED

- Bhaumik, M. 2012. CFD Simulation of SDHW storage tank with and without heater. **Int. J. of Advancements in Research & Technol.** 1(2): 1–11.
- Cengel, Y.A. and J.M. Cimbala. 2010. **Fluid Mechanics; Fundamentals and Applications**. McGraw-Hill. Singapore. 980 pp.
- Chaichanasiri, E. and C. Suvanjumrat. 2013. The k- ϵ turbulence model to simulate the two-phase flows of fluids in flumes using C++ Open Source Code computational fluid dynamic software. **Kasetsart J. (Nat. Sci.)**. 47(3): 460–477.
- Hegazy, A.A. 2007. Effect of inlet design on the performance of storage-type domestic electrical water heaters. **Appl. Energ.** 84: 1338–1355.
- Launder, B.E. and D.B. Spalding. 1974. The numerical computation of turbulent flows. **Comput. Methods Appl. Mech. Eng.** 3: 269–289.
- Morrison, G.L., Y.C. Soo Too and M. Behnia. 2005. Heat transfer and flow visualization in vertical mantle heat exchangers for solar water heaters. *In The 6th World Conference on Experimental Heat Transfer, Fluids Mechanics, and Thermodynamics*. 17-21 April 2005, Miyagi, Japan.
- Open FOAM. 2009. **Programmer's Guide**. OpenCFD Limited. USA. 95 pp.
- Patankar, S.V. and D.B. Spalding. 1972. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. **Int. J. Heat Mass Transfer**. 15: 1787–1806.
- Sezai, I., L.B.Y. Aldabbagh, U. Atikol and H. Hecisevki. 2005. Performance improvement by using dual heaters in a storage-type domestic electric water-heater. **Appl. Energ.** 81: 291–305.
- Soo Too, Y.C., G.L. Morrison and M. Behnia. 2004. Vertical mantle heat exchanger for solar water heaters. *In The 42nd annual conference of the Australian and New Zealand Solar Energy Society*. 1-3 December 2004, Perth, Australia.

- Versteeg, H.K. and W. Malalasekera. 2007. **An Introduction to Computational Fluid Dynamics: The Finite Volume Method.** Prentice Hall. London, UK. 503 pp.
- Wang, D. and S. Shan. 2006. Numerical analysis of water temperature distribution in the tank of ASHPWH with a cylindrical condenser. **In The 6th International Conference for Enhanced Building Operations.** 6-9 November 2006, Shenzhen, China.
- Yeoh, G.H. and J. Tu. 2010. **Computational Techniques for Multi-Phase Flows.** Butterworth-Heinemann Publications. Waltham, MA, USA. 643 pp.