Journal of Heat and Mass Transfer Research 2 (2015) 21-29

Journal of Heat and Mass Transfer Research

Journal homepage: http://jhmtr.journals.semnan.ac.ir

Numerical study of fluid flow and heat transfer in a gas-tank water heater

Mohammad Hossein Tavakoli* and Khatereh Moharramkhani

Physics Department, Bu-Ali Sina University, Hamedan, Iran

PAPER INFO

History:
Received 02 April 2014
Received in revised form 06 June 2014
Accepted 27 July 2014

Keywords:
Fluid flow
Heat transfer
Tank water heater
Computer simulation

A B S T R A C T

Influence of a vent hood at the exit of exhaust flue gas and flue baffles in the fire tube on the temperature and flow fields of a gas tank water heater, as well as the structure and amount of heat transferred to the water tank has been studied numerically using two-dimensional steady state finite element simulation. Observations show that without a vent hood, there is a downward gas flow in the flue and a strong vortex in the lower burner chamber. Using a vent hood prevents the gas back flow into the flue, and placing the flue baffles increases the heat delivered to the water

1. Introduction

Among the energy components, heating systems that use mainly fossil fuels are particularly important. Rising demands for gas on one hand, and limited gas reserves on the other hand, are sensible more than before. Consequently, the efficiency of gas consumption in all consumers is a crucial factor [1]. One of these systems is the gas-fired tank water heater that produces domestic hot water at home. Storage water heaters are by far the most common type used in many countries. These systems heat and store water in a tank so that hot water is available at any time. As hot water is drawn from the top of the tank, cold water enters from the bottom of the tank and then is heated [2]. The heating source can be electricity, gas or oil. As Figure 1 shows, a gas tank water heater consists of different parts. These parts include a burner at the bottom of the tank,which heats the water, an inner steel tank that holds the water being heated, fire tube (or chimney) thatis in the middle of system, baffle plates usually made of metal and inside the fire tube, thermal insulation that surrounds the tank to decrease the amount of heat lost to the surrounding, dip tube to allow cold water to enter the tank, pipe to allow hot water to leave the tank, thermostat that reads and controls the temperature of the water inside the tank, and a vent hood that lies at the exit of the combustion products. Air enters the combustion chamber, combines with the gas fuel, the mixture is ignited, and then the resulting combustion heat is transferred to the water through metal surfaces of the storage tank. The vent pipe is an exhaust pipe that carries the by-products of combustion to the outside and serves as a heat exchanger. This pipe is typically surrounded by the water tank and often contains baffles to slow the escape of these gases through the vent, thus

Corresponding author: Mohammad Hossein Tavakoli, Physics Department, Bu-Ali Sina University, P.O.Box: 65174, Hamedan, Iran.
Tel: +98 81 38271541, Fax: +98 81 38280440
Email: *mhtvkl@gmail.com* & *mht@basu.ac.ir*

allowing more time for heat to be transferred to the water around the pipe. Thermostatic controls regulating the temperature of the water areusually located near the mid-section of heaters.

The proper mixture of additional air with combustion or exhaust gasses is important in fuel-fired water heaters to assist the combustion products moving upwardwithin the vent pipe to the outdoors [3-5]. If their escape is impeded or blocked, serious problems can develop. To assure this proper mixture, gas-fired water heaters are equipped with a 'cone-shaped' draft hood on the vent pipe, as it emerges from the tank.

Some gasor oil-fired water heaters may also be equipped with vent dampers on the vent pipe. This energy-saving device automatically 'closes' the vent pipe to prevent the escape of heat to the flue when the main burner is not being fired. This, in turn, slows down the rate at which the water in the tank cools down.

The tank water heaters have been a research topic for many years, and so many mathematical models, experiments and simulation programs have been reported for these systems [6-15].

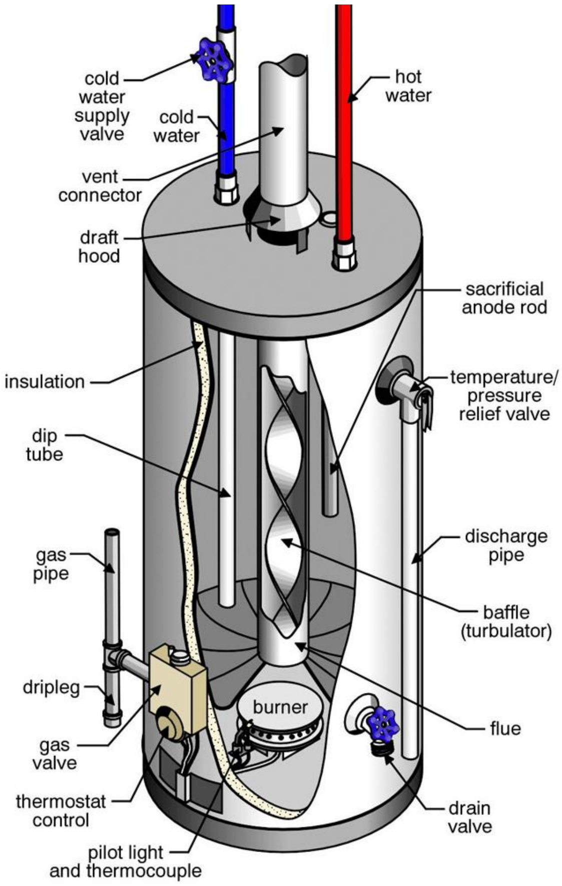

Fig. 1. A schematic view of the gas water-storage heater and its components [16].

In this paper, influence of the flue baffles placed in the fire tube and a vent hood at the exit of the combustion products on the temperature field, fluid flow and heat transfer structure of a gas tank water heater is studied numerically using two-dimensional steady state finite element method. To do so, first a mathematical model containing assumptions, governing equations and boundary conditions is described, and then the obtained numerical results are presented and explained comprehensively.

2. CFD analysis of the tank water heater

2.1 Assumptions and governing equations

The computational model of the considered gas tank water heating system is a three-phase (water, gas and solid) model. The simplifications and assumptions for the model are: (1) The system runs at a steady state and is axially symmetric. Therefore, all temperature and flow fields discussed in this article are limited to two dimensions. (2) Both water and gas flows are incompressible and Newtonian. (3) Specific heat of water and gas is constant. (4) Flows are layered, smooth and stable. (5) The mass flow rate at the inlet and outlet is zero. Due to a density change for heating process, the Boussinesq approximation is used in the equations of momentum conservation. The momentum and energy equations are linked and must be solved simultaneously [17,18]. For a complete solution of the equations, the continuity equation should be considered. Thus, the basic equations in direction of r and z are:

$$\rho(v_r \frac{\partial v_r}{\partial r} + v_z \frac{\partial v_r}{\partial z}) = -\frac{\partial p}{\partial r} + \mu [\frac{1}{r}\frac{\partial}{\partial r}(r\frac{\partial v_r}{\partial r}) + \frac{\partial^2 v_r}{\partial z^2} - \frac{v_r}{r^2}] \quad (1)$$

$$\rho(v_r \frac{\partial v_z}{\partial r} + v_z \frac{\partial v_z}{\partial z}) = -\frac{\partial p}{\partial z} + \mu [\frac{1}{r}\frac{\partial}{\partial r}(r\frac{\partial v_z}{\partial r}) + \frac{\partial^2 v_z}{\partial z^2}] + \rho g \beta (T - T_0) \quad (2)$$

$$\frac{1}{r}\frac{\partial}{\partial r}(r v_r) + \frac{\partial v_z}{\partial z} = 0 \quad (3)$$

$$\alpha[\frac{1}{r}\frac{\partial}{\partial r}(r\frac{\partial T}{\partial r})+\frac{\partial^2 T}{\partial z^2}]=v_r\frac{\partial T}{\partial r}+v_z\frac{\partial T}{\partial z} \quad (4)$$

where $V=(v_r,v_z)$ is the fluid velocity vector, Tis the temperature,ρis the density,βis the thermal expansion coefficient, μ is viscosity coefficient of fluid and α is called the diffusivity coefficient. In order to convert two Navier-Stocks and continuity equations to one equation, we need the stream function:

$$v_z=-\frac{1}{r}\frac{\partial \psi}{\partial r} \text{ and } v_r=\frac{1}{r}\frac{\partial \psi}{\partial z} \quad (5)$$

where ψ is the stream function. In parts that circulating flow is considered, the following equation of vorticity (ω) is applied [19]:

$$\omega=\frac{1}{r}(\frac{\partial^2\psi}{\partial r^2}-\frac{1}{r}\frac{\partial \psi}{\partial r}+\frac{\partial^2\psi}{\partial z^2}) \quad (6)$$

2.2 Boundary conditions

At the rigid surfaces, no-slip condition is used. In addition, the inlet air temperature in the combustion chamber is assumed 27ºC. At the outer surfaces of water heater, heat dissipation via radiation and convection are considered:

$$-k\frac{\partial T}{\partial \hat{n}}=h(T-T_a)+\sigma\varepsilon\left(T^4-T_a^4\right) \quad (7)$$

where k is the thermal conductivity, his the convection (Newtonian) heat transfer coefficient, $\hat{n}$ is the unit normal vector, σ is the Stefan-Boltzmann constant, εis the emissivity and Tais the ambient temperature of the system.

2.3 Computational domain

We consider three cases of a gas tank water heating system for our computation, shown in Figure 2. They are:

case a - without vent hood and flue baffles,

case b - with vent hood and without flue baffles, and

case c – with vent hood, flue baffles and damper.

In case c, in order to enhance the heat exchange between the exhaust flue gas and the water within the tank, baffles are secured in the exhaust flue. A chain of discs is suspended within the flue to retard the rising gases, so that the sidewall of the flue absorbs heat and exchanges it with the liquid disposed thereabout.

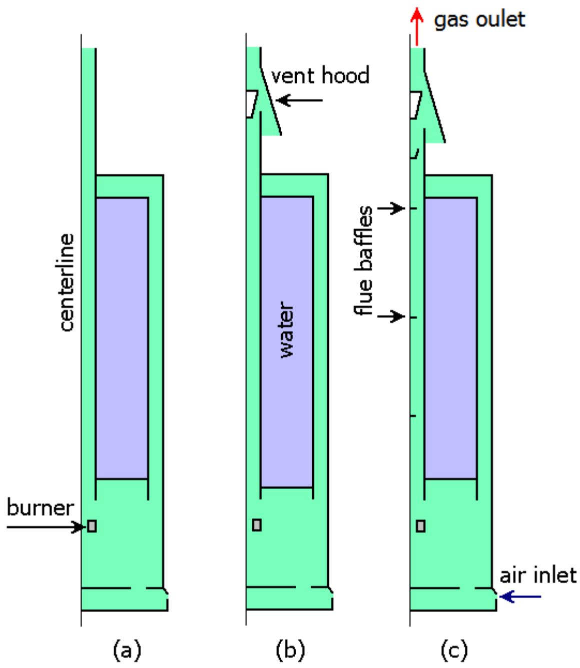

Fig. 2. Sketch of three gas tank water heating systems considered in our computations, (a) without vent hood and flue baffles, (b) with vent hood and without flue baffles, and (c) with both vent hood and flue baffles.

2.4 Grid generation and numerical solution

Calculation of the governing equations with boundary conditions has been made by a 2D steady state finite element method (FlexPDE package [20]). In order to select a suitable computational grid, which can lead to numerical error minimized, different grids have been studied with different structures and node numbers for this problem. Due to complexities in the geometry, the computational domain was meshed with unstructured triangle grids. Grid generation was initially performed for all subsections (e.g. gas side, water tank and solid walls) with different grid resolutions. To achieve the desired level of accuracy on simulation results, mesh independency needs to be obtained. Therefore, a series of grid independency tests have been conducted to ensure that optimized computational mesh was obtained. Grid independency tests have been carried out for each mesh model, considering both convergence time and solution precision, shown in Figure 3.Accordingly, the grid generated for CFD analysis of water heater has different resolutions at different locations.Therefore, for a better performance of the software, an optimal mesh structure with a fine grid in regions with high field gradients such as flue baffles, vent hood, walls and burner has been selected, which could optimize the required memory and computation time with sufficient accuracy. It is also suggested that prior to using an analysis technique on a new configuration, the technique should be validated. The validation process has been done in two ways, (a) Using some known (respected) test cases [FlexPDE-Help], and (b) Influence of changes in boundary conditions on the obtained solutions.

3. Results and Discussion

In Figure 4, the temperature distribution of system is shown for three considered cases. Heat transfer takes place through the system via three mechanisms of conduction, convection and radiation. A part of the energy of combustionproducts raises the temperature of the fire tube and tank walls, and thus the energy is transferred to the water within the tank via conduction. In all cases, cold air inserts into the combustion chamber via intake window (open hole to the atmosphere) positioned at the lowest chamber, and makes this area to be the coldest part of the system. Whenthere is a vent hood (case b), release of the combustion products to the outside of the system is increasing, so less heat is transferred to the tank wall compared to case a. This causes increasing temperature difference between the fire tube and the tank water, which is shown by decreasing the temperature of the water tank in

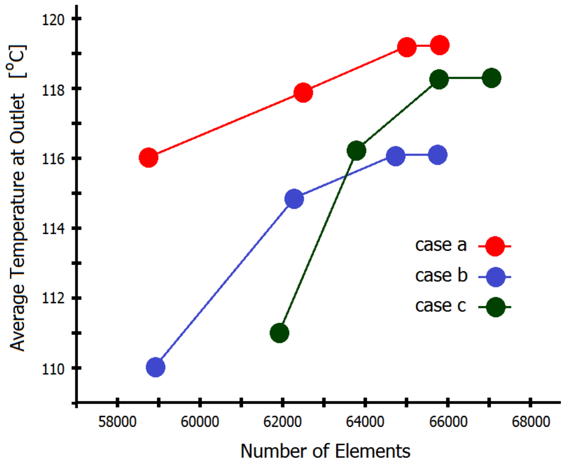

Fig. 3. Grid independency study for the average temperature at outlet.

Figure 4(b). In case c, placing the flue baffles in the fire tube delays heat transferring to the outlet. Therefore, the heat is diverted to the water tank more effectively than other two cases, which is displaced clearly by increasing the temperature of the water tank in Figure 4(c). It can be seenthat the minimum temperature of water is located at the lower left corner of the tank and the maximum at the upper corner of fire tube, which is about the boiling point of water.

Figures 5-7 show the velocity vectors of gas and water for three cases studied here. Gas particles have an upward high speed near centerline in the fire tube. The hot gases from the burner chamber are cooled as they rise up the flue or chimney, which is in contact with the water being heated, i.e., the gas velocity as well as its temperature decreasesdue to heat transport to the water tank. Thus, their energy reduces during their upward motion. Since the left wall of the water tank close to the flame has a higher temperature, density of the water particles decreases and so moves upward. The momentum exchange between particles will replace cold-water particles with higher density. For this reason, an eddy flow is formed within the water domain, which occupies its volume completely. This vortex transfers the heat from the chimney’s wall to the upper and outer surface of the water tank, which is displayed by deflection of the temperature field in that area In case a, the reduction of gas particles' energy is high in the middle of tube and in the radial direction from center line to the sidewall, and the outlet pressure is not so high. Therefore, for particles close to the water tank wall, gravity overcomes buoyancy force and makes them to return downward (called a back draft), Figure 5(b). A back draft can interfere with proper venting of harmful gasses, and could blow out the pilot light or burner. For this reason, a large and strong gas vortex is formed in the lower burner chamber which prevents the incoming air, and thus the inlet cold air is quite weak, Figure 5(c). This vortex also prevents entering and reaching air needed for combustion to the burner. It has been observed when the outlet pressure becomes lower or more negative, the inlet airflow becomes negative, and the combustion products enter the room space from the air’s lower space [21]. It is important to note that the combustion process creates by-product gasses such as carbon dioxide, carbon monoxide, and nitrogen oxide.

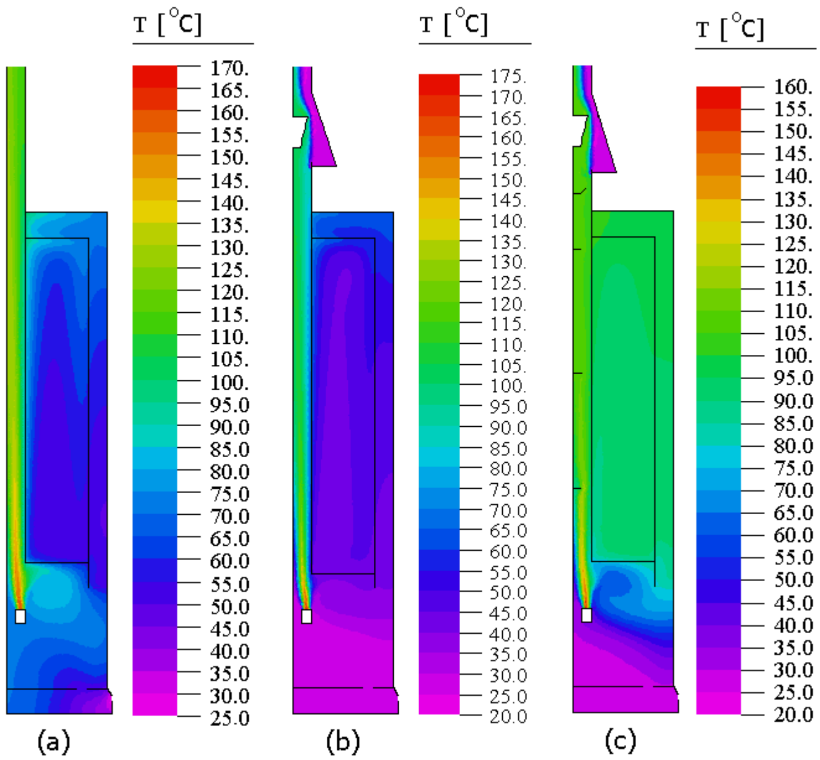

Fig.4. Temperature distribution of the system in three considered cases (a) without vent hood and flue baffles, (b) with vent hood and without flue baffles, and (c) with both vent hood and flue baffles.

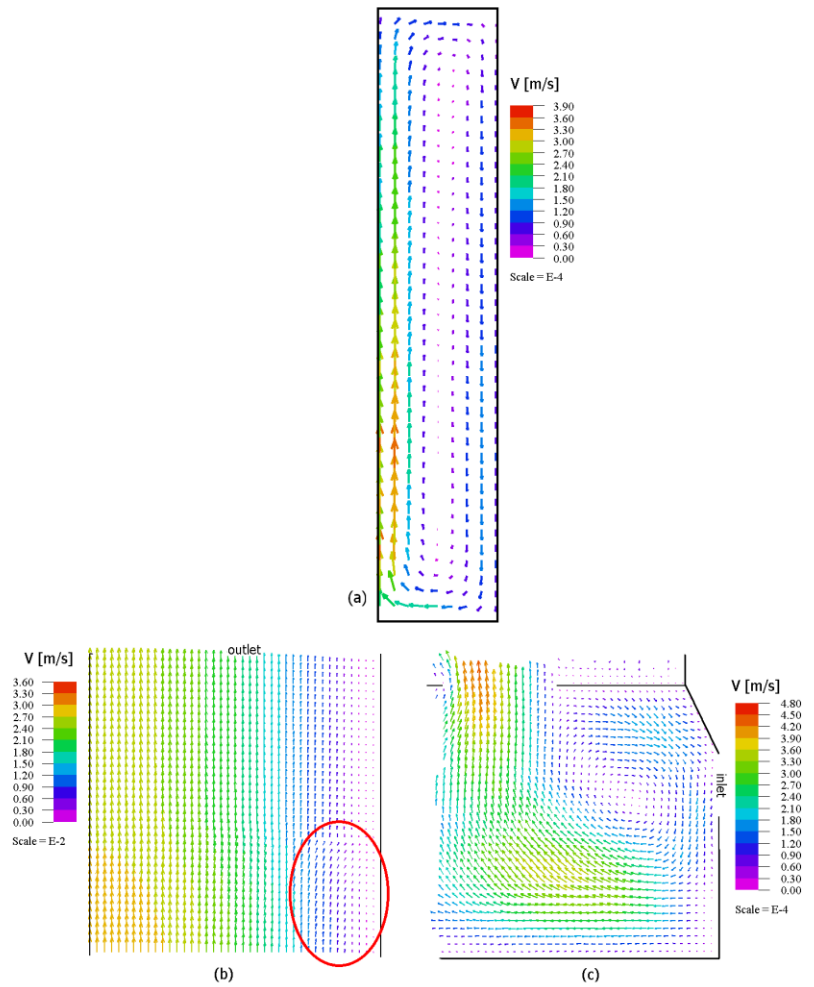

Fig.5. Distribution of velocity vectors of water and gas in *case a* - without vent hood and flue baffles (a) water tank (b) fired tube, and (c) lower space with intake window. The return flow of cold air with high density is clearer in Figure (5-b).

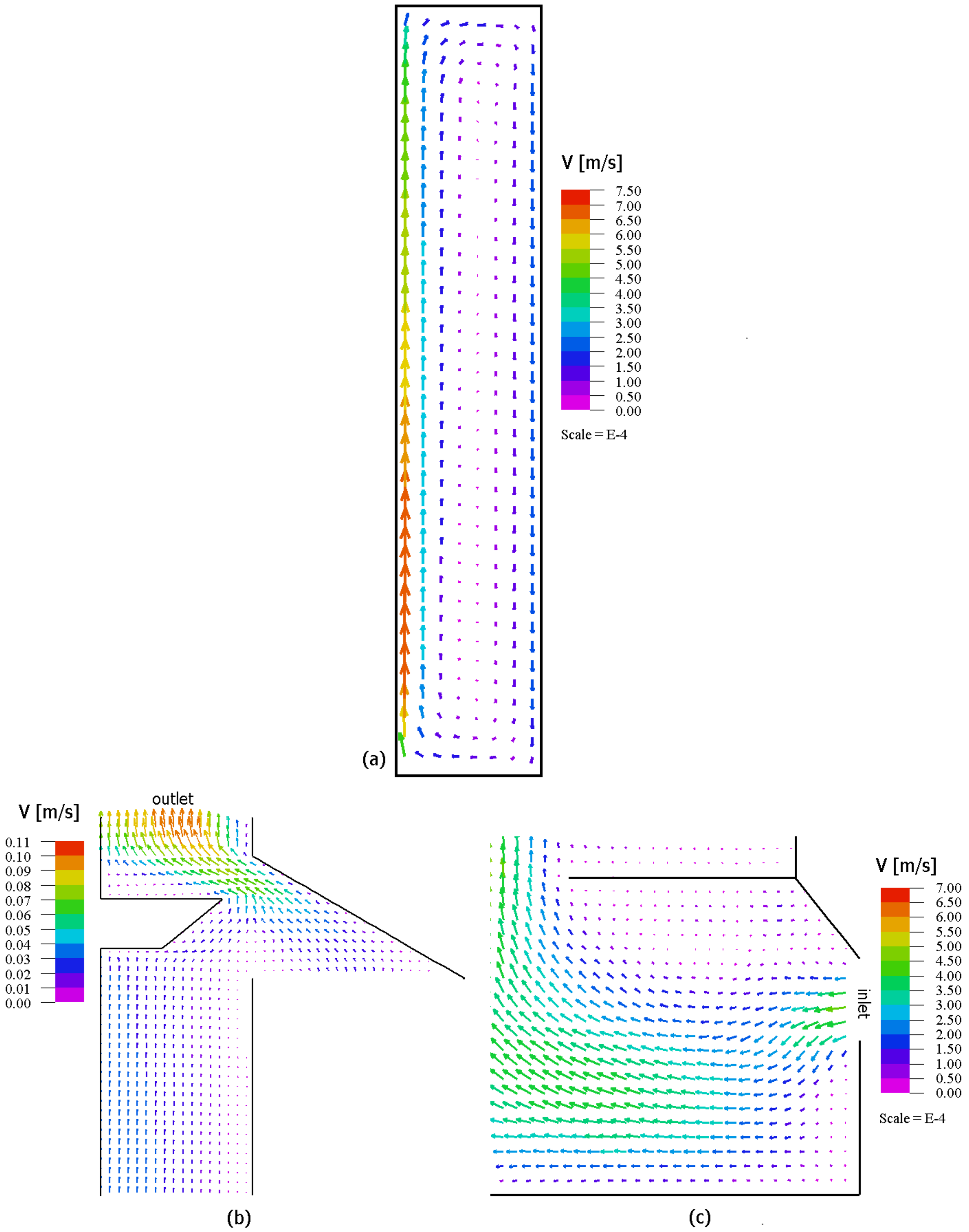

Fig.6. Distribution of velocity vectors of water and gas in *case b*- with vent hood and without flue baffles (a) water tank (b) fired tube, and (c) lower space with intake window.

These gasses are harmful to breathe and need to be vented outside of the house. So, case a is not a safety configuration and is not recommended [3,4,21].In case b, placing a vent hood at the exhaust flue causes an increase in releasing the combustion products from the fire tube, and prevents the downward flow of these products into the flue too, Figure 6(b). In other words, it completely transfers all products of combustion and vents gasses to the outside without considering the vent or spillage at the draft hood. In addition, the related problem of air needed for combustion is solved. For this reason, the formed gas vortex in the lowest chamber is quite small and weak, so the inlet air from the intake window is markedly strong which improves the combustion efficiencies of the tank water heater, Figure 6(c). Because of high outlet flow via the vent hood in this case, the transferred heat to the water is reduced to 540 W

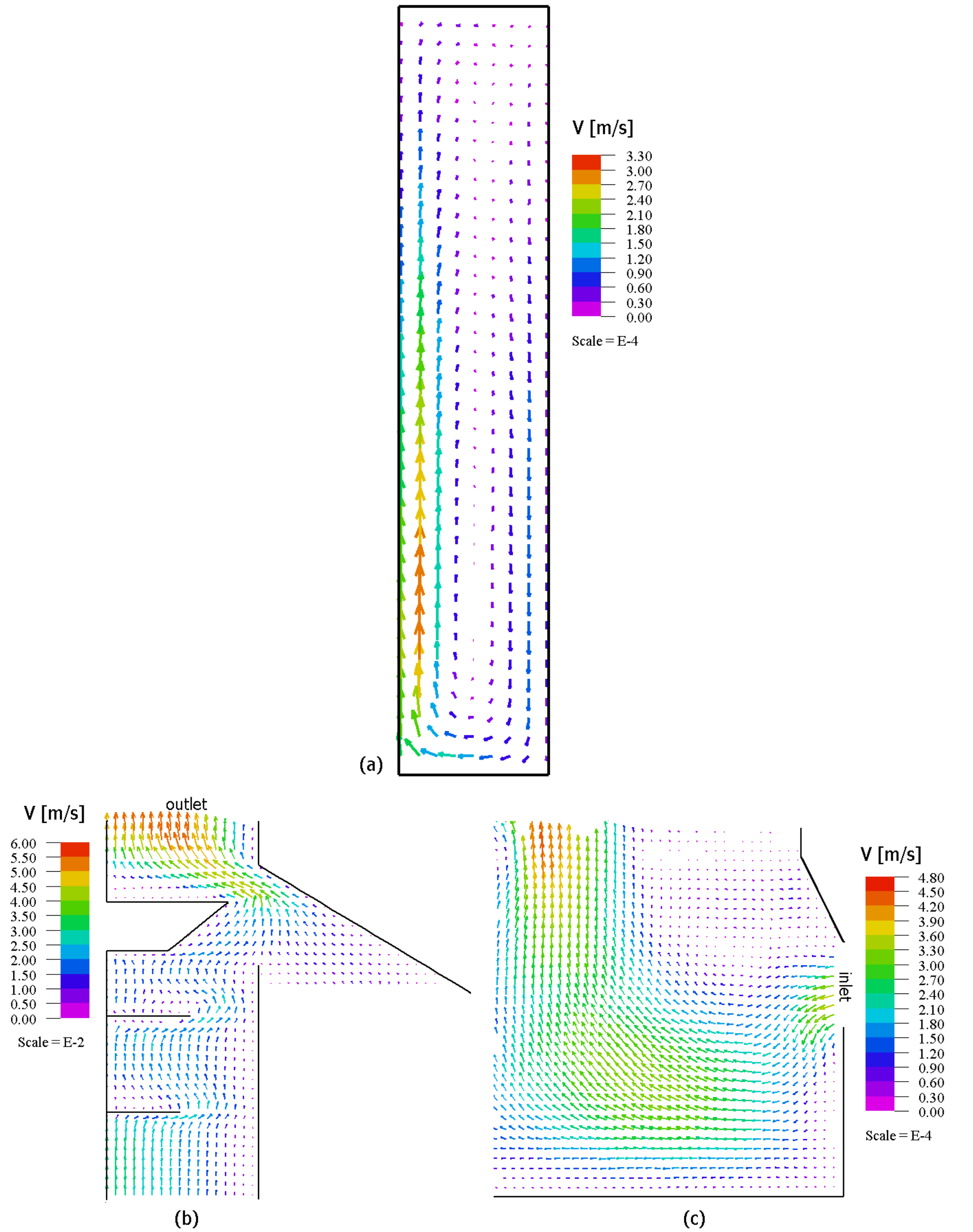

Fig.7. Distribution of velocity vectors for water and gas in *case c*- with vent hood and flue baffles (a) water tank (b) fired tube, and (c) lower space with intake window.

(Nusselt Number $Nu^{case\,b} = 8.9$) from 910 W in case a ($Nu^{case\,a} = 12.6$). For this reason, the required time to reach the final water temperature is increased compared to case a [21], which illuminatesthe low efficiency of this system. It is important to note thatbecause of the zigzag flow pattern caused by the vent hood, there is a dead space just above it and close to the centerline, where gas recirculates with low velocity resulting in a reduction in the amount of combustion products exiting.

In case *c*, three flue baffles, and *a* damper are located in the fire tube in order to increase the delivered heat to the water tank. Baffles and damper help to direct flue gases in radial direction [7,15,22,23].Gas particles within the fire tube are diverted to the tank wall with collisions to the flue baffles, so their energy can be transferred to water simply and then move out with less energy in the fire tube, Figure 7(b). For this reason, these particles spend more time in the chimney,and also a low difference of particles' velocity as well as a thin velocity boundary layer thickness is formed in that space. Therefore, energy is transferred rapidly to the adjacent layers. In this case, the maximum amount of heat can betransferredfrom the hot medium, i.e. flue gases into the water part, i.e. cold medium (1020wor $Nu^{case\ c} = 13.8$), Table 1. This is visible in Figure 4(c), which represents the highest temperature of water tank compared to other cases. In addition, the rate of cold air intakein the combustion chamber is quite high and strong, Figure 7(c), which can provide enough air needed for a complete combustion.

4. Conclusions

In this paper, two-dimensional steady state numerical simulations for three cases of agas storage water heater are performed to reveal the effects of vent hood, damper and baffles on the flow filed and heat transfer. From the above calculations, we can conclude that:

- In case a, there is a downward gas flow in the flue and a strong vortex in the lower burner chamber. The formed gas eddy in the lower combustion chamber prevents arrival of the necessary air fora complete combustion.
- A draft hood positioned at the top of the flue (case b) prevents the backflow of air into the flue,but the delivered heat to the water is not efficient.
- The introduction of the flue baffles and damper (case c) results in an increase in the residence time of hot gases in the flow duct. Consequently, it reduces the temperature of flue gases, and leads to a higher heat transfer rate to the water and therefore makes the water heater more efficient.
- It should be mentioned that a proper baffle design (location and inclination angle)would provide an optimal performance of water heater. The detailed knowledge of the heat transfer and flow distribution provided in this investigation may serve as a basis for further optimization of gas storage water heaters.

Table 1. Detailed information about the total heat transferred to the water and outside for the three considered cases.

Total heat transferred (units)	*Case a*	*Case b*	*Case c*
Output (W)	570	930	370
Water (W)	910	540	1020

References

[1]. M. Krarti, Energy Auditing of Building Systems – An Engineering Approach, CRC Press, (2000).

[2]. http://oee.nrcan.gc.ca/equipment/heating/806?attr=4

[3]. K.F. Michaelsen, K. Taudorf, Danger of Gas Water Heaters, the Lancet, 322,229-230, (1983).

[4]. W.R. Chang, C.L. Cheng, Carbon monoxide transport in an enclosed room with sources from a water heater in the adjacent balcony, Building and Environment, 43,861-870, (2008).

[5]. O. Aydin, Y.E. Boke, An experimental study on carbon monoxide emission reduction at a fire tube water heater, Applied Thermal Engineering, 30, 2658-2662, (2010).

[6]. G. Gutierrez, F. Hincapie, J.A. Duffie, W.A. Beckman, Simulation of forced circulation water heaters; effects of auxiliary energy supply, load type and storage capacity, Solar Energy, 15,287-298, (1974).

[7]. G.N. Tiwari, N.K. Dhiman, Effect of the baffle plate on transient performance of built-in-storage water heater, Energy Conversion and Management,23,151-155, (1983).

[8]. T.W. Abou-Arab, M.O. Othman, Y.H. Najjar, N.T. Ahmad, Combustion and heat transfer characteristics for a dual-fuel cylindrical water heater model,Fuel,69,485-489, (1990).

[9]. A.K. Kar, K.M. Al-Dossary, Thermal performances of water heaters in series, Applied Energy,52,47-53, (1995).

[10]. A.A. Hegazy, M.R. Diab, Performance of an improved design for storage-type domestic electrical water-heaters, Applied Energy,71,287-306, (2002).

[11]. M. Kim, M.S. Kim, J.D. Chung, Transient thermal behavior of a water heater system driven by a heat pump, International Journal of Refrigeration,27,415-421, (2004).
[12]. A.A. Hegazy Effect of inlet design on the performance of storage-type domestic electrical water heaters, Applied Energy, 84, 1338-1355, (2007).
[13]. D.S. Sowmy, R.T.A. Prado Assessment of energy efficiency in electric storage water heaters, Energy and Buildings, 40, 2128-2132, (2008).
[14]. S. Tajwar, A.R. Saleemi, N. Ramzan, S. Naveed Improving thermal and combustion efficiency of gas water heater, Applied Thermal Engineering,31,1305-1312, (2011).
[15]. M. MoeiniSedeh, J.M. Khodadadi Energy efficiency improvement and fuel savings in water heaters using baffles, Applied Energy, 102,520-533, (2013).
[16]. http://www.atlantarealestateinspection.com/
[17]. F.P. Incropera, D.P. De Witt Introduction to Heat Transfer, John Wiley and Sons,(1996).
[18]. I.H. Shames, Mechanics of Fluids, Mc-Graw-Hill, New York, (1982).
[19]. J.P. Holman, Heat Transfer, Mc-Graw-Hill, New York, (1990).
[20]. http://www.Pdesolutions.com
[21]. K. Moharramkhani, M.Sc. Thesis, Bu-Ali Sina University, (2010).
[22]. Technical Manual, T.M. 5-650Central Boiler Plants, Publications of the Headquarters, United States Army Corps of Engineers, (1989).
[23]. S.J. Craig, J.F. McMahon, The effects of draft control on combustion, ISA Transactions,35,345-349, (1996).

بررسی عددی جریان شاره و انتقال حرارت در یک آبگرمکن مخزندار گازی

محمد حسین توکلی[*]، خاطره محرمخانی

گروه فیزیک، دانشکده علوم پایه، دانشگاه بوعلی سینا، همدان

اطلاعات مقاله

دریافت مقاله: ۱۳ خرداد ۱۳۹۳

پذیرش مقاله: ۵ مرداد ۱۳۹۳

واژگان کلیدی:

جریان شاره،
انتقال حرارت،
آبگرمکن مخزندار،
تشعشع حرارتی،
شبیه سازی رایانه ای.

چکیده

تاثیر یک کلاهک تعدیل در خروجی لوله تنوره و صفحات مانع در محفظه احتراق روی میدان های دما و جریان شاره، و همچنین ساختار و مقدار گرمای منتقل شده به مخزن آب در یک آبگرمکن مخزندار گازی بطور عددی با استفاده از شبیه سازی دو بُعدی حالت پایا روش عناصر متناهی بررسی شده است. نتایج بدست آمده نشان می دهد که در حالت بدون کلاهک تعدیل، یک جریان گاز برگشتی به سمت پایین در لوله تنوره و یک چرخش قوی در پایینی محفظه احتراق وجود دارد. استفاده از کلاهک تعدیل مانع از برگشت بسوی پایین گاز احتراق و قرار دادن صفحات مانع، موجب افزایش گرمای منتقل شده به مخزن آب می گردد.