CFD simulation of inlet position of hot water into SDHW thermal storage tank

Assist. Prof. Mainak Bhaumik

***Abstract*— Water should stratify in the hot water storage tank. Stratification is the phenomena which found in the water due to density difference when water gets heated up. Under highly optimum stratification condition hot water should be filled up or supplied or withdrawal from the top of the storage tank. While at the bottom of the cold atmospheric water should be maintain. Due to filling of hot water from the top or top side of the tank, thermal stratification disturbs. This disturbance in thermal stratification leads to mixing of hot and cold water and hence potential of thermal energy storage gets affected. A 2D CFD simulation is carried out in the solar domestic hot water (SDHW) storage tank to investigate the disturbances occurs in the thermally stratified tank. It helps to make study about the position of hot water inlet into the storage tank.**

***Index Terms*— SDHW, CFD, Simulation, thermal stratification, thermocline.**

I. Introduction

Solar domestic hot water storage as a thermal energy is having great potential. Water is highly stratified liquid. Due to density difference and buoyancy effect hot water remains at the top part in any storage tank or naturally in the pond, lake, and ocean etc. Due to this lighter in weight of hot water, it is general practice to supply hot water into the storage tank through top or top side of the storage tank. Cold water we keep from beginning itself at the bottom of the tank. Or else initially the entire tank could be filled with the cold water. Few amounts of water can be supplied to the solar collector. Water started to achieve hot temperature from Sun light through the solar collector cells. As water achieved desire hot temperature, it could be allowed to pass through the pipe line to the storage tank. If required instantly the hot water is consumed for domestic use such as bathing, cooking, drying etc. Whenever instant demand is not there, it is being stored in the storage tank for further use. Atmospheric cold water is denser than hot water which settles down at the bottom of the tank. From the bottom part of the tank the atmospheric cold water is supplied to the solar collector for heating. The cycle continues. A schematic diagram of thermosyphonic type SDHW storage tank is as shown in the Fig.1 with basic equipments. Make up atmospheric water is supplied through the bottom side of the tank, on or before atmospheric water level downs. The hot and cold water is always having the tendency to mix and transfer heat. This mixing and heat transfer of heat in between hot and atmospheric cold water is undesired.

Nomenclatures

SDHW	Solar domestic hot water	ρ_{ref}	Density of water in kg/m^3
CFD	Computational Fluid Dynamics	ρ	Local density of water in kg/m^3
2D	Two Dimensional	th1	Hot water inlet to the tank
ICEM	Integrated Computer Engineering Manufacturing	th2	Hot water outlet from the tank
		tc1	Cold water recirculation temperature to the solar collector

Symbols

		tc2	Make up atmospheric cold water inlet to the tank
T	Local temperature in K		
T_{ref}	Buoyancy reference temperature K	v	Velocity of inlet hot water in m/sec.
β	Thermal expansion coefficient K^{-1}	Re	Reynold's No.

This leads to loss in thermal energy from hot water and thermal stratification of hot and atmospheric cold water. Also due to inlet hot water flow rate from the pipe is the major factor in mixing of hot and atmospheric cold water. The hot water from the top of the storage tank penetrates more as of due to downward velocity in favor of gravity and mixes and creates more turbulence than the top side inlet position. Even though it takes time to lose hot water thermal energy but still this inlet from top of the tank is having more disturbance on thermal stratification of hot and cold water. The hot water inlet from the top side of the storage tank is having lesser contribution of mixing and thermal stratification loss of hot and cold water. Flow inlet at lower velocity and lower Reynolds number also helps to maintain thermal stratification. The study analysis of the inlet position and convection flow behavior is utmost to optimize the location of the inlet ports. Also it involves optimizing the flow behavior and flow rate.

Manuscript received April 04, 2014.
Mainak Bhaumik, B.E.(Mechanical), M.E. (Thermal), working as an Assistant Professor in Bharati Vidyapeeth College of Engineering, Kharghar, Navi Mumbai, India, Tel.: +91 98 19 56 8377

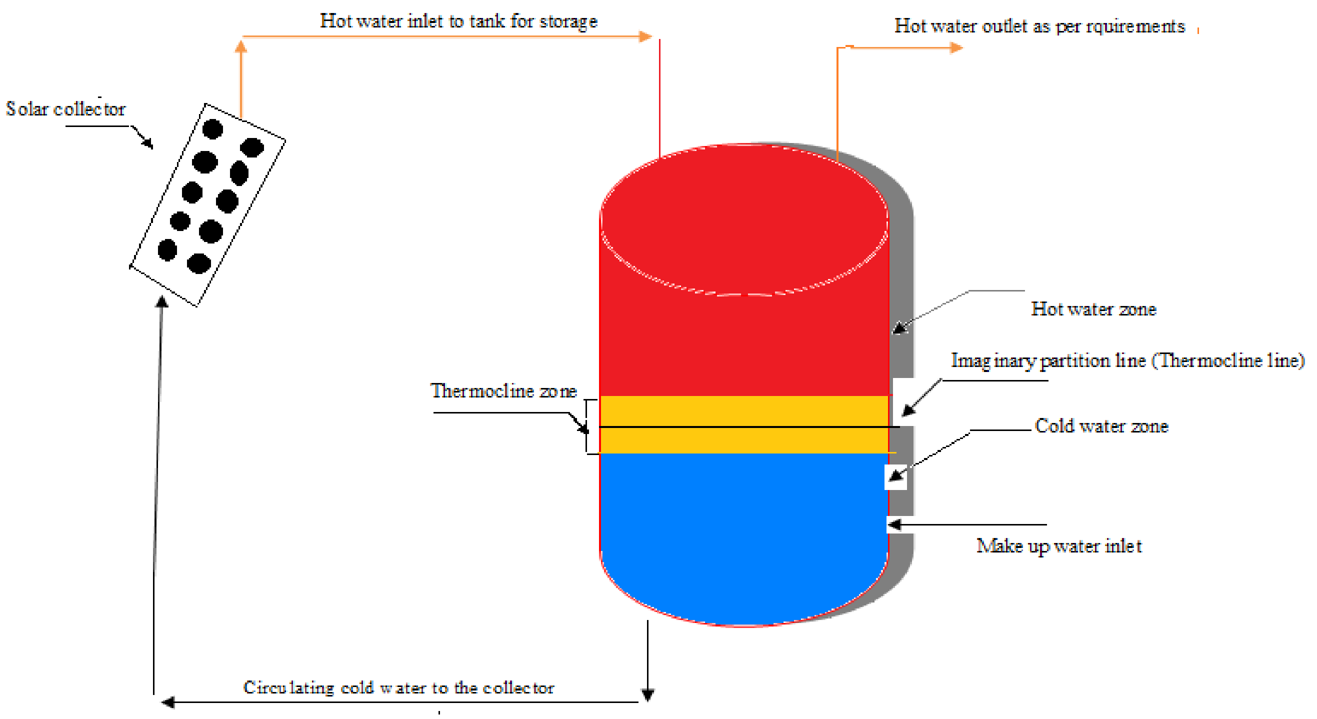

Fig.1: Schematic diagram of SDHW system without using heat exchanger.

II. Problem Definition

The 2D CFD study analysis problem consists of a 2D thermal energy storage tank. The tank dimensions considered are as 1500mm height and 550mm diameter. The size of the inlet and outlet ports are 12mm. the water storage capacity of the tank is 150 liters, sufficient for four persons in a nuclear family. The tank is insulated and considered as ideal insulated adiabatic tank feed in to CFD software as initial condition. Heat loss from the tank side walls are zero. The constant inlet hot water temperature is 80 C. The atmospheric cold water is at room temperature of 25 C. The hot water inlets are from the top side of the tank and top centre of the tank. It has been explained with the schematic diagram as shown in Fig.2.

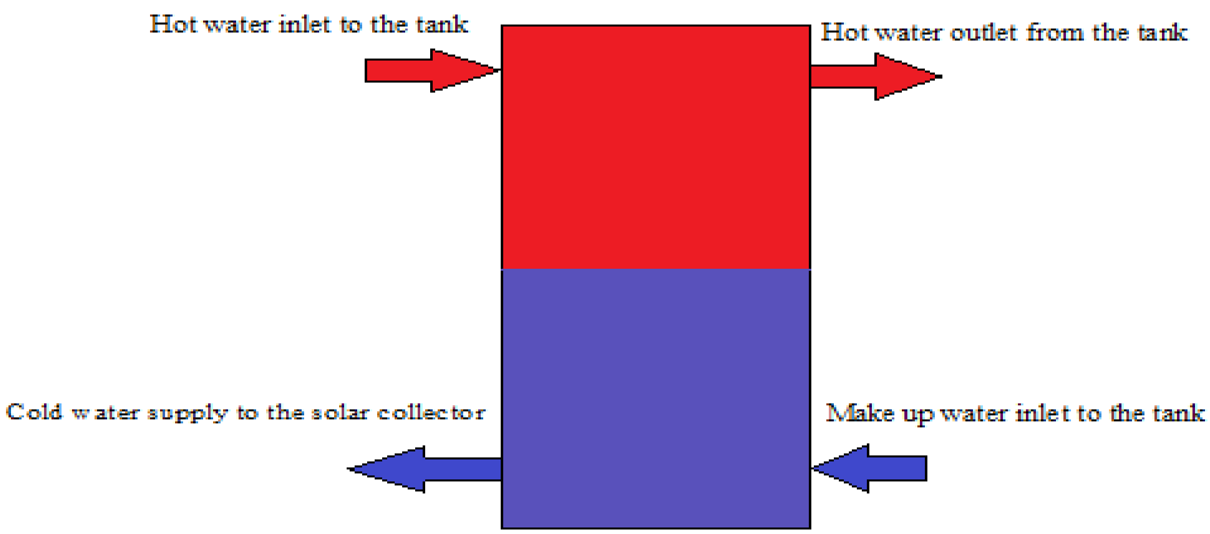

Fig. 2: schematic diagram of problem definition of SDHW storage tank.

Case 01

The hot water inlet from the top centre of the tank at a temperature of 80 C.The inlet velocity of water is 0.0067 m/sec calculated based on Reynold's No. as1000.

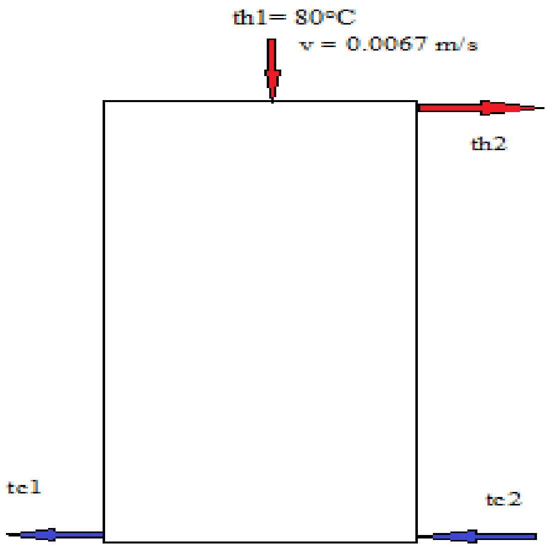

Fig. 3: Hot water inlet at the top centre of the tank.

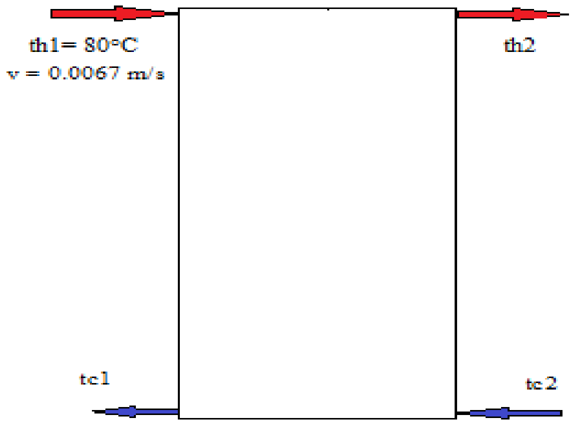

Fig. 4: Hot water inlet from top side of the tank.

Case 02

The hot water inlet from the top side of the tank at a temperature of 80°C. The inlet velocity of water is 0.0067 m/sec calculated based on Reynold's No. as1000.

III. CFD IMPLEMENTATION

The thermal stratification loss due to flow of hot and cold water in the tank is being investigated using commercial CFD Fluent 6.3 software. Transient CFD calculations are performed with a density of water as function of temperature, shown in equation (1). The second order upwind method is used for the discretization of the pressure, momentum and energy equations. The simulation runs with a time step of 1s.

A. Problem setup

Basically problem consists of solid area in which incompressible fluid water is there. So Pressure based solver is selected to solve transient heat transfer flow.CFD implicit scheme is adopted to solve the problem. Unsteady temperature behavior of water need to be obtained from the problem.

B. Solution setup

The buoyancy effect is modeled with Boussinesq approximation during CFD simulation. The model uses a constant density fluid model but applies a local gravitational body force throughout the physical domain which is a linear function of the fluid thermal expansion coefficient (β) and the local temperature difference relative to a datum called the buoyancy reference temperature. The Boussinesq approximation models the change in density using eq.1. A zero velocity field is assumed at the start of all simulations

$$(\rho - \rho_{ref}) = -\rho_{ref}.\ .\beta.\ (T - T_{ref}) \qquad (1)$$

C. Boundary conditions

The boundary conditions applied to case 01 and case 02 are adiabatic walls of the tank with no heat generation inside. The inlet hot water velocity is 0.0068 m/sec.

D. Mesh

2D geometry is created using ICEM software and then orthogonal meshing is also done using the same software. The size of the mesh is 18,360.

IV. RESULTS AND DISCUSSIONS

The ICEM software is used to create the solid modeling geometry and meshing. The time transient CFD analysis is being carried out using Fluent Software. The inlet velocity chosen is laminar. The effect of inlet velocity of water and the movement of the water path is visualized and observed.

Case 01

Inlet hot water from the central inlet port from top part penetrates at for a particular depth and then creates disturbance which in result disturbs the thermal stratification of water

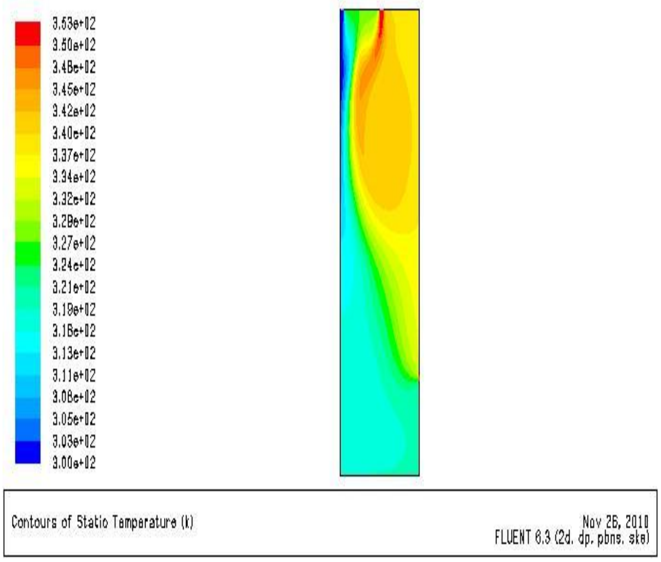

Fig. 5: CFD simulated result of case 01.

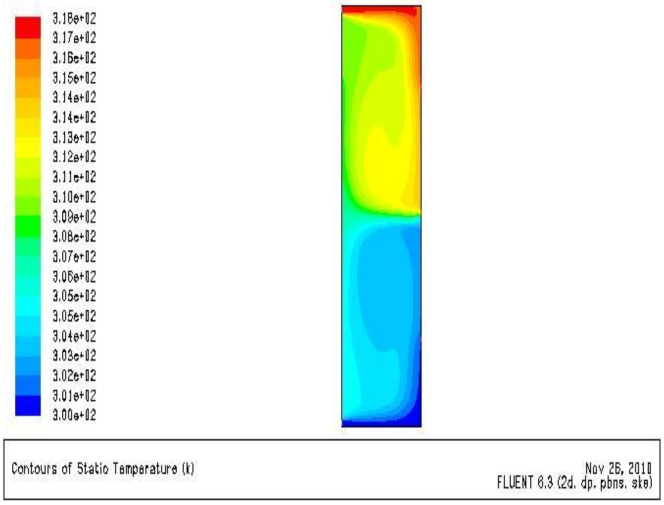

Fig. 6: CFD simulated result of case 02.

Case 02

The hot water is being forced from the top side of the tank. Due to that there is no as such penetration of water and hence no as much more disturbance of thermal stratification from the central region of the water tank. The mixing starts from the side of the tank.

This type of configuration of the inlet system of water is useful. It is better to implement inlet port from the top side of the tank to have lesser thermal stratification disturbance in the tank water.

V. CONCLUSION

The tank inlet hot water is being analyzed by using commercial numerical CFD Fluent software. The inlet hot water from the top centre of the tank creates more disturbances and sooner mixing of hot and cold water which in turn disturbs thermal stratification of the tank. The top side inlet passes the water horizontally and so there is no as such direct downward penetration of hot water and mixing with cold water. Also if we pass the hot water from the top side inlet port at a lower velocity of laminar flow Reynolds no. below 2000, the disturbance of thermal stratification is less. Mixing of hot and cold water is not so frequent. From the analysis it is advisable to construct the storage tank inlet port at the top side of the tank instead to provide at the top centre of the tank. The inlet port from the side of the tank also reduces the length of the pipe and hence the cost is also reduced. Unnecessary movement of water in the pipe can be avoided by top side inlet of hot water in to the tank.

Acknowledgments

The author is greatful to Dr. R.S.Maurya for his technical support and to Professor Shewaz Wanrick, Professor Sameer Khan and my all other colleagues. Also the author is extremely thankful and grateful to his father Chitta Ranjan Bhaumik, mother Gita Bhaumik, brother Chiranjib Bhaumik and sister Rumki Bhaumik for their constant moral and financial support.

REFERENCES

[1] Kenjo et al., Experimental and numerical study of thermal stratification in a mantle tank of a solar domestic hot water system, Applied Thermal Engineering 27 (2007) 1986 – 1995.

[2] Guo et al., Experimental research and operation optimization of an air-source heat pump water heater, Applied Energy 88 (2011) 4128 – 4138.

[3] Mainak Bhaumik, CFD Investigation of Thermocline behavior in Thermal Storage Tank, International Conference on Recent Advances in Engineering, Technology and Management. Vol.01, pp. 125-132, SPICON 2012 MECHANICAL (31st May – 2nd June 2012)

[4] Rao et al., Influencing parameters on performance of a mantle heat exchanger for a solar water heater – a simulation study, International Journal of Engineering, Science and Technology 2 (2010) 155 – 164.

[5] Mainak Bhaumik, CFD simulation of SDHW Storage Tank with and without Heater, International Journal of Advancements in Research & Technology 01 (2012) 124 – 134.

[6] Cengel & Boles, "Thermodynamics – An Engineering Approach", McGraw-Hill, Fifth Edition, 2008.

[7] Patankar S.V.,Numerical heat transfer and fluid flow, Taylor & Francis, 1980.

[8] ANSYS Fluent 6.3.6 User's Manual, ANSYS Inc.

[9] ICEM-CFD 12.0 User's Guide ANSYS,Inc

Mainak Bhaumik, B.E.(Mechanical), M.E. (Thermal), working as an Assistant Professor in Bharati Vidyapeeth College of Engineering, Kharghar, Navi Mumbai, India. The author is having three years of experience in Deepak Fertilizers and Petrochemical Corporation Ltd., Navi Mumbai, India as an Assistant Manager, one year teaching experience in Sardar Patel College of Engineering, Andheri (W), Mumbai, India as a Lecturer in Mechanical Engg. Dept., one year teaching experience in Rizvi College of Engineering, Bandra (W), Mumbai, India as a Lecturer in Mechanical Engg. Dept. one year teaching experience in Saboo Siddik College of Engineering, Byculla (W), Mumbai, India as a Lecturer in Automobile Engg. Dept. and currently working as an Assistant Professor in Bharati Vidyapeeth College of Engg., Kharghar, New Mumbai, India. Residential address is C-6-8-0.3, Sector-18, New Panvel, Navi Mumbai – 410206, Maharastra, India.