

## **"CFD ANALYSIS OF CONVECTIVE FLOW IN A SOLAR DOMESTIC HOT WATER STORAGE TANK"**

**Mr. Mainak Bhaumik**

M.E. (Thermal Engg.)

Automobile Engineering Department

MHSSCOE, Mumbai - 400008

+91 9819568377

mainak\_bhaumik@yahoo.com

### **ABSTRACT**

Two dimensional computational fluid dynamics (CFD) analysis of heat and fluid flow is carried out in a solar domestic hot water (SDHW) storage tank. Commercial software ICEM (Integrated Computer-aided Engineering and Manufacturing) is used for modelling and meshing and FLUENT 6.3 for analysis. A positional change of hot water inlet geometry which is horizontally from the vertical downward position of hot water inlet into the tank. The heat growth and movement and flow disturbance of water layer in the tank is the main study of the CFD research.

### **INTRODUCTION**

Storage tank is used to store thermal energy. Thermal energy can be stored either as a sensible heat or latent heat. In sensible heat is stored in the fluid especially water or oil or solid media rock or pebble is used. Free solar radiation heat energy storage technique is used using solar collector. Water is very popularly used to store thermal energy. It needs one solar collector, piping and storage tank for this process. As the water gets heated up it becomes lighter which results in the movement of flow and temperature from the collector to the tank. This movement of hot water and storing in the tank can be either thermosyphonic or forced type. In thermosyphonic type there is involvement of any external force by introducing pump. But in case of forced type there is the involvement of pump to boost up the movement, filling and flow of hot water inside the tank. At varying flow rates for different Reynold's number the flow path, mixing of water and boundary layer growth are changes and also affects the efficiency of the collector. The schematic diagram of SDHW system is as shown in Fig.1.

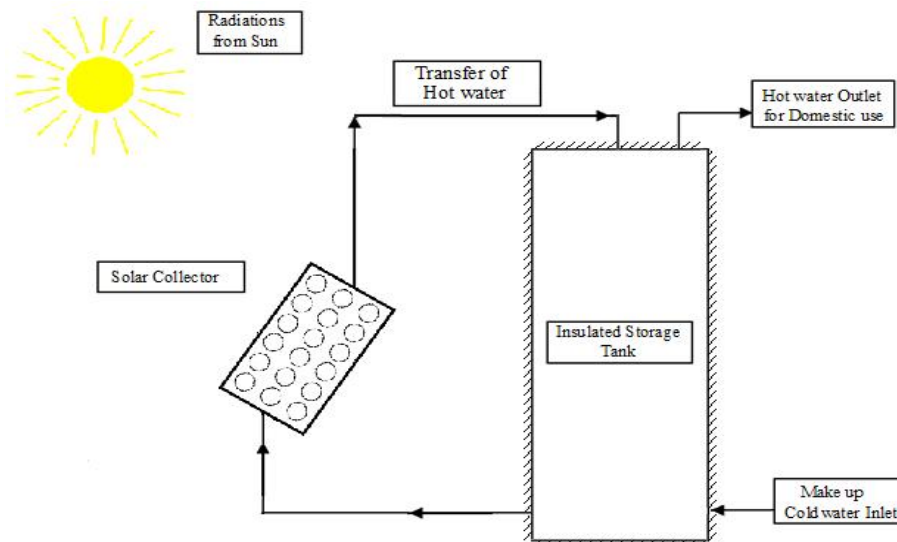


Fig.1: Schematic diagram of SDHW system.

## PROBLEM DEFINITION

Problem under the study contains an insulated vertical hot water storage tank. The storage tank is having height of 1500mm and diameter 500mm. Pipe inlet diameter is 15mm. The storage tank is initially filled with cold water. Later on the hot water inlet takes place from the top side of the tank at a temperature of 60°C. The walls of the tank are adiabatic wall. Inside the storage tank the water experience heating, cooling and mixing process due to temperature gradient present there.

The study consists of hot water inlet from the top of the tank wall. Hot water inlet temperature is 60°C and the laminar velocity developed at a flow rate of 3 L/min.

## NUMERICAL IMPLEMENTATION

The movement of water and heat from transfer profile in the tank is investigated using numerical techniques by commercial Fluent 6.3 software. CFD calculations are carried out to theoretically investigate the fluid flow in the hot water tank. The tank model includes the steel tank wall as a solid region and the hot and cold water volume of the tank as a fluid region. The SIMPLE algorithm is used to treat the pressure-velocity coupling. The calculation is considered as convergent for the continuity equation, the momentum equations and the energy equation. The simulation runs with a time step of 1 s. Unsteady time analysis is carried out.

$$\text{Density, (kg/m}^3\text{)} \quad \rho = 863 + 1.21 * T - 0.00257 * T^2 \quad (1)$$

Where, T is fluid temperature, [K].

### Problem setup

Software need problem data set up is configured as per different cases of problem. Basically problem consists of solid area in which incompressible fluid water is there. Pressure based solver is selected to solve transient heat transfer of selected incompressible fluid. CFD implicit scheme is adopted to solve the problem. The geometry of the problem is 2

dimensional. Unsteady temperature behavior of water need to be obtained from the problem. All such problem setup data is tabulated in the Table 1.

**Table 1: Problem Setup**

Sr. No	Define problem Setup Options	Problem Setup Adopted
01	Solver	Pressure Based Solver
02	Formulation	Implicit
03	Spatial discretization	2D
04	Temporal discretization	unsteady
05	Material	water
06	Cell zone material	water

**Solution setup**

Natural-convection flow is modelled with Boussinesq approximation during CFD simulation. The buoyancy for an incompressible fluid with constant fluid properties is modelled by using the Boussinesq approximation in ANSYS FLUENT 6.3. 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 ( $\beta$ ) and the local temperature difference relative to a datum called the buoyancy reference temperature. The Boussinesq approximation models the change in density using eq.4.

$$(\rho - \rho_{ref}) = -\rho_{ref} \cdot \beta \cdot (T - T_{ref}) \tag{4}$$

Where, T is the local temperature in K,

$T_{ref}$  is the buoyancy reference temperature in K,

$\beta$  is the thermal expansion coefficient in  $K^{-1}$ ,

$\rho_{ref}$  is the density of water in  $kg/m^3$  and

$\rho$  is the local density in  $kg/m^3$

A zero velocity field is assumed at the start of all simulations. The calculation is considered convergent for the continuity equation, the momentum equations and energy equations. The simulation runs with a time step of 1second and a duration of 1 hour and 24 hours. Table 2 gives solution control and solution setup.

**Table 2: Solution Setup**

Sr. No	Solution Control Parameters	Solution Setup Adopted
01	Pressure	SIMPLE
02	Energy equation activation	Second Order Upwind

03	Momentum equation activation	Second Order Upwind
04	Time step	01s

### Boundary conditions

The boundary condition applied is the walls of the tank is adiabatic wall. Initial hot water temperature is 60°C.

### Mesh

2D geometry is created using ICEM software and then meshing is also done using the same software. The size of the mesh is 36,780. In order to better resolve the heat transfer and fluid flow in the region adjacent to the tank wall, a boundary layer mesh is applied so that there is a fine and dense mesh in the area close to the wall. Fig. 2 illustrates the 2-D mesh on the vertical cut-plane of the tank.

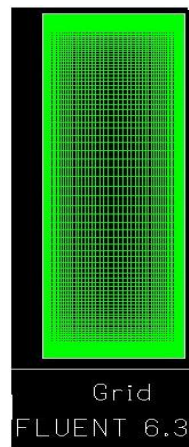


Fig.2: Complete 2-D view of Coarse Mesh size is 18,360.

## RESULTS AND DISCUSSION

Present work uses ICEM for solid modelling and meshing purpose. Fluent 6.3 has been used as solver. The convective heat transfer and boundary layer development due to heat transfer is captured. Natural convection considered is negligible. Grid independent test also done to check result independence on grid. This analysis permits qualitative examinations of the intrinsic nature of the flow and temperature values in the tank. Fig. 2 and Fig. 3 show the pictorial view of the temperature and velocity distributions for the flow rate of 3 L/min, respectively.

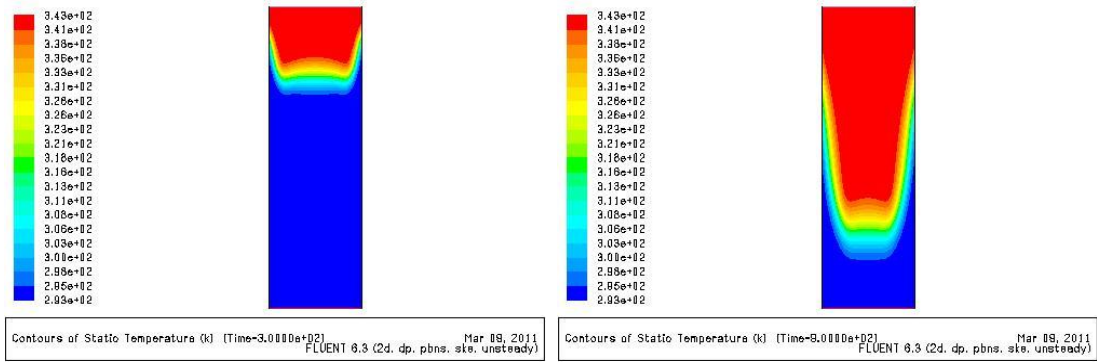


Fig. a: After 05 min

Fig. b: After 10 min.

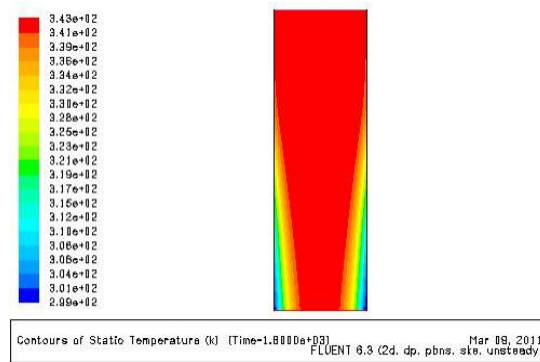


Fig. C: After 20 min.

Fig. 2: Temperature distribution (in K) in the tank at a flow rate of 3L/min.

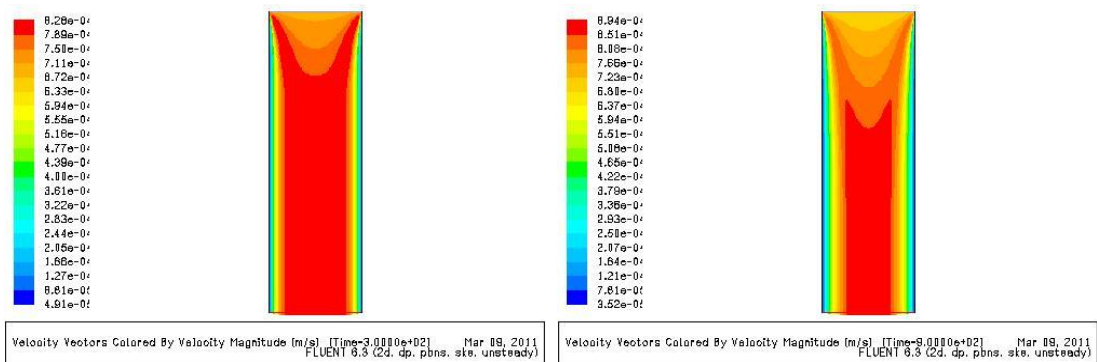


Fig. a: After 05 min.

Fig. b: After 10 min.

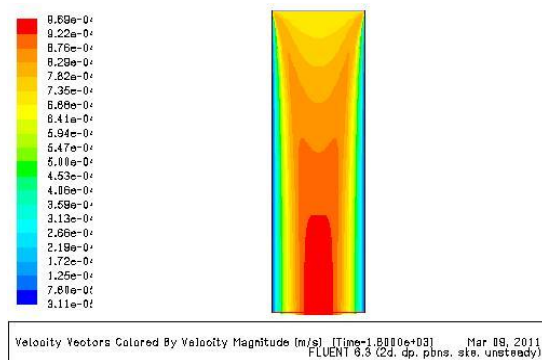


Fig. C: After 20 min.

Fig. 3: Velocity contour (in K) in the tank at a flow rate of 3L/min.

## CONCLUSION

It is observed that at a lower laminar velocity the temperature profile develops and the thermal stratification of the tank is maintained. Thermal stratification of the tank is important as thermal stratification increases the efficiency of the collector. Lower velocity to be applied at inlet and exit port to avoid mixing and thermal stratification loss.

## REFERENCES

1. Lavan and Thompson, "Experimental study of thermally stratified hot water storage tank", Solar Energy., Vol. 19, Issue 5, pp., 519-524, 1977.
2. Sliwinshi et al., "Performance for stratified thermal storage tanks a theoretical model of the charging process", Solar Energy., Vol. 51, Issue 6, pp., 513-519, 1979.
3. Jaliria Y. and Gupta S.K., "Decay of thermal stratification in a water body for solar energy storage", Solar Energy., Vol. 28, Issue-2, pp., 137-143, 1982.
4. Hollands and Lightstone, "A review of low flow, stratified tank solar water heating system," Solar Energy., Vol.43, Issue-2, pp., 97-105, 1989.
5. Sukhumi S. P., "Solar Energy", McGraw-Hill, New Delhi, 1996.
6. ANSYS FLUENT 6.3.26 User's Manual, ANSYS, Inc.
7. ICEM-CFD 12.0 User's Guide, ANSYS, Inc.