Design and optimization of thermal storage tank using CFD

Satish Kumar singh[1], Jayant Chaudhari[2]
Lecturer, SESGOIFE, Karjat, India
[1]*satish.140788@gmail.com*,[2]*chaudharijay2000@gmail.com*

Abstract:- Thermal stratification in hot water tanks of solar thermal energy system has significant positive effect on their efficiency. The heated water from the collector is discharge at top inlet port of the tank and a thermal stratification arrives because of temperature variation which gives rise to a density variation in the medium. The results of three-dimensional (3D) transient/unsteady Computational Fluid Dynamics (CFD) simulations to study, during charging operation effect of several design and operating parameters on the flow behaviour, thermal stratification and performance of a hot water storage tank installed in solar thermal energy systems. Results were validated with experimental data, found in the literature, showed good agreement. To observe the effects of essential geometrical and operating parameters on the system performance, different computation test cases were run. The results showed that an appropriately designed storage tank can provide improved stratification conditions. Also, at early design stages, 3D unsteady CFD simulations could be used as an effective tool to optimize thermal storage tank parameters, so that it may add value to the storage tank performance and efficiency, by optimizing the whole solar thermal energy storage system design and size.

***Keywords*:** *3D unsteadyCFD simulation, ansys fluent tool, thermal stratification, thermal energy storage tank.*

*****

I. INTRODUCTION

Thermal stratification in hot water tanks of solar thermal energy systems has a significant positive effect on their efficiency. Solar hot water tanks, which keep the hot and cold water by means of gravitational stratification, are widely used for load management and energy conservation applications. The single stratified tank is attractive in low to medium temperature applications, due to its simplicity and low cost. Solar collector performance increases as the collector inlet temperature decreases, due to reduced convective heat loss from the collector plate. However, a hotter storage tank temperature is required in order to meet the energy demand such as hot water, space heating and air cooling applications. The heated water from the collector is discharged at the top inlet port of the tank and a thermal stratification arises because of temperature variation which gives rise to a density variation in the medium. Therefore, the cold and hot water are separated by means of gravitational effect; the intermediate region is called thermo cline. The efficiency increases when the temperature difference between the top and bottom is high. The storage performance can be improved by maintaining an effective stratification in the fluid [1-10].

Many theoretical and experimental studies have been per formed in order to understand the transient phenomena during the charging and discharging modes in the stratified tanks, and to identify the fundamental mechanisms of stratified tank operation. Despite the significant progress that has been achieved, and due to the flow complexity resulting from simultaneous interaction of numerous parameters, thermal energy storage tank modelling remains a problem that has yet to be resolved. One of the main difficulties still to overcome, is a reliable numerical modelling and simulation on the unsteady internal flow of a thermal energy system for the prediction of its performance, taking into account stratification and mixing between the hot and cold thermal layers.

Objective

1. To appropriately designed storage tank such that stratification conditions improves.
2. Model validation with the experimental data.
3. CFD simulations can be used as an effective tool to optimize thermal storage tank parameters so that it may add to the value of the storage tank performance and efficiency, by optimizing the whole solar thermal energy storage system design and size.

II. PROBLEM DEFINITION

The mixing process, which consists of fluid streams that enter the tank from the collector loop and from the load, is one of the major causes of the degradation of thermal stratification inside the storage devices. Several numerical and experimental studies that analyses various alternatives so as to preserve the thermal stratification can be found in the literature (Y.M. Han, 2009; Y Tian, 2014; A Zachar, 2003). With regard to the unloading processes, the mass flow rate delivered to the load depends on the hourly load profiles. Often, daily loads are approximately one renovation of the tank per day. Usually, the maximum requirement for consumption is about 20–30% of the total daily mass flow rate (Morrison and Sapsford, 1983; Arbel and Sokolov, 1994). Considering a domestic hot water system with a 4 m 2 collector area and a 0.3 m3 storage tank (_300 l), the peak mass flow rate could be around 0.0167–0.025 kg/s (_60–90 l/h), sometimes even larger. These peaks of cold water entering the tank can affect its thermal stratification adversely. As the performance of storage tanks depends on various factors, there still remain a issues in the design methodology to be addressed. Despite diverse research efforts made so far,

ISSN: 2321-8169
126 - 131

the effects of design and operating parameters on the performance of thermal storage systems has partially undertaken, making design engineers rely greatly on simple numerical tools and method.

In this present work, the various configurations and operating conditions influence on the stratification performance for thermal energy storage tanks are simulated, using CFD. A 3D unsteady CFD model has been developed and validated. The various effects of the thermal storage tank geometry and operating conditions on stratification were analyzed and optimum design and operating condition of tanks were find out. The hot water is extracted from the highest point in the tank to make sure the outlet temperature is maximum and thus efficiency and performances of thermal storage tank increases. In order to do so, the experimental set-up constructed by A.Zachar (2003) was taken into account. The device tested was horizontal tanks of length 0.8m with an internal diameter (D) of 0.4m. Simulations were performed with inlet and initial temperature of 20 and 41℃

Governing Equations of Fluid Dynamics.

Each CFD software package has to produce a prediction of the way in which a fluid will flow for a given situation. To do this the package must calculate numerical solutions to the equations that govern the flow of fluids. For the analyst, it is important to have an understanding of both the basic flow features that can occur and so must be modeled and the equations that govern fluid flow. The physical aspects of any fluid flow and heat transfer are governed by three fundamental principles:

1 Continuity Equation

A. For an incompressible fluid, the continuity equation for a steady three dimensional flow can be written as:

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} + \frac{\partial W}{\partial Z} = 0$$

2. ***Momentum Equation***

$$[\text{Mass}]\begin{bmatrix}\text{Acceleration in}\\ \text{i direction}\end{bmatrix} = \begin{bmatrix}\text{Body forces acting in}\\ \text{i direction}\end{bmatrix} + \begin{bmatrix}\text{Surface force acting}\\ \text{in i direction}\end{bmatrix}$$

$$\rho\left[u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right] = F_x - \frac{\partial p}{\partial x} + \mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right]$$

$$\rho\left[u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right] = F_y - \frac{\partial p}{\partial y} + \mu\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right]$$

$$\rho\left[u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right] = F_z - \frac{\partial p}{\partial z} + \mu\left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right]$$

3.Energy Equation

$$\begin{bmatrix}\text{rate of energy}\\ \text{input due to}\\ \text{conduction}\end{bmatrix} + \begin{bmatrix}\text{rate of energy}\\ \text{input due to}\\ \text{workdone by}\\ \text{conduction}\end{bmatrix} + \begin{bmatrix}\text{rate of energy}\\ \text{input due to}\\ \text{workdone by}\\ \text{surfacestress}\end{bmatrix} = \begin{bmatrix}\text{rate of increase}\\ \text{of energy in the}\\ \text{element}\end{bmatrix}$$

$$u\frac{\partial T}{\partial x} + V\frac{\partial T}{\partial Y} + W\frac{\partial T}{\partial Z} = \alpha\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right)$$

Discretization Methods

The dependability of the picked discretization is for the most part settled numerically as opposed to systematically as with basic direct issues. Uncommon consideration should likewise be taken to guarantee that the discretization handles intermittent arrangements smoothly. The Euler mathematical statements and Navier-Stokes comparisons both concede stuns, and contact surfaces. A portion of the discretization strategies being utilized are talked about beneath:

1 Finite Volume Method (FVM)

This is the "classical" or standard approach used most often in commercial software and research codes. The governing equations are solved on discrete control volumes. FVM recasts the PDE's (Partial Differential Equations) of the N-S equation in the conservative form and then discretize this equation [1].

$$\frac{\partial}{\partial t}\iiint Q\, dV + \iint F\, dA = 0$$

2 Finite Element Method (FEM)

This strategy is prevalent for basic investigation of solids, but on the other hand is material to liquids. The FEM plan requires, nonetheless, extraordinary consideration to guarantee a traditionalist arrangement. The FEM detailing has been adjusted for use with the Navier-Stokes mathematical statements. In spite of the fact that in FEM protection must be dealt with, it is a great deal more steady than the FVM approach (4.2) therefore it is the new course in which CFD is moving. By and large dependability/power of the arrangement is better in FEM however for some cases it may take more memory than FVM techniques. In this strategy, a weighted remaining comparison is framed:

$$R_i = \iiint W_i Q\, dV^e$$

Where, R_i is the equation residual at an element vertex i, Q is the conservation equation expressed on an element basis, W_i is the weight factor and V^e is the volume of the element.

3 Finite Difference Method (FDM)

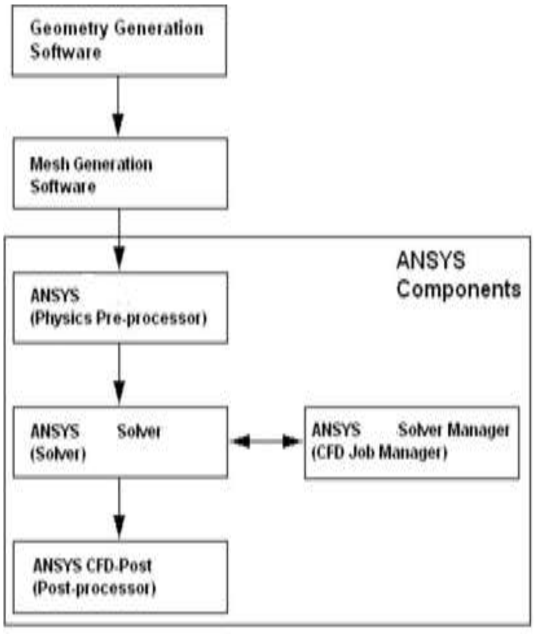
This method has historical importance and is simple to program. It is currently only used in few specialized codes. Modern finite difference codes make use of an embedded

ISSN: 2321-8169
126 - 131

boundary for handling complex geometries making these codes highly efficient and accurate

4 Boundary Element Method (BEM)
In this method the boundary occupied by the fluid is divided into surface mesh. Volume mesh is not applicable in this particular method

The Structure of ANSYS Fluent



III. METHODOLOGY

In the present work, effects of the thermal storage tank geometry as well as the operating conditions on stratification performance will be investigated in detail. Distinct characteristics of significance to thermal energy storage tank optimum design and operation that have effects on stratified tanks will be identified and analyzed**.**

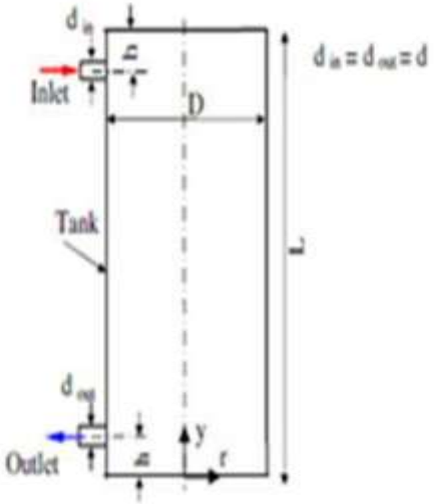


Fig.1 Thermal storage tank

Fig.4.1 illustrates the geometric model of the solar hot water storage tank. The physical model for charging process in cylindrical tank consists of inlet and outlet ports. The inlet hot water is from top of the tank and the outlet cold water is from the bottom of the tank as shown in Fig. 4.1. The inlet supply has uniform velocity and temperature. The external walls are insulated. The system is symmetrical, thus the computational domain is modeled for half cylinder with symmetric boundary conditions.

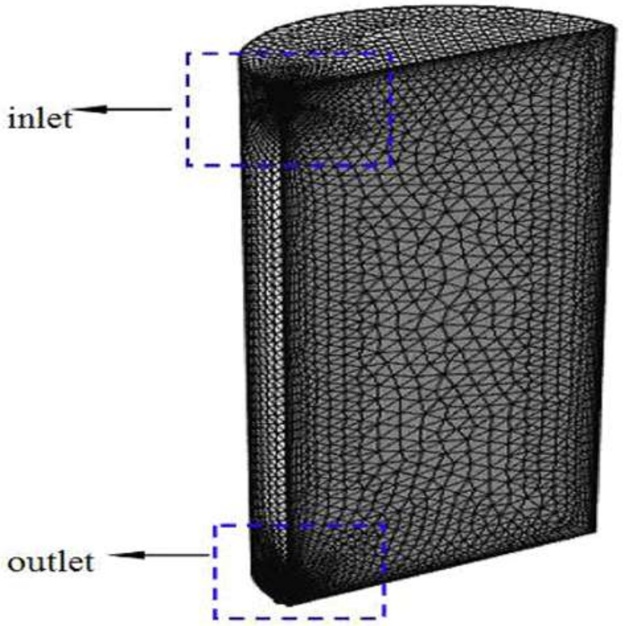


Fig.2 Computational domain with unstructured grid of the modelled thermal storage tank

.

1 Effects of geometrical factors: Temperature stratification inside the capacity hot tank is influenced by various components, for example, tank measurement (tallness and width), channel and outlet ports area and kind of liquid impact of viewpoint proportion (L/D) at various tank statures and breadth and the gulf and way out port areas on the temperature conveyance inside the tank at various time history will be contemplated.
Impact of operational variables: The target of the study is to distinguish the working condition envelope for the best warm proficiency. The dynamic method of operation of the tank considering the variety of the delta stream rate, the underlying temperature distinction in light of the bay heated water temperature variety and an altered starting temperature or on the underlying water temperature variety with the channel boiling hot water temperature settled, will be numerically examined, for example, impact of mass stream rate, impact of gulf high temp water temperature variety, impact of beginning water temperature variety.

ISSN: 2321-8169
126 - 131

Fig. 3 Geometric model of storage tank

As shown in fig.3 the physical model for charging process in cylindrical tank consists of inlet and outlet ports. The inlet hot water is from top of the tank and the outlet cold water is from the bottom of the tank. The inlet supply has uniform velocity and temperature. The external walls are insulated

Following assumption are made

1. The working fluid is incompressible.
2. The viscous dissipation is negligible.
3. The influence of pressure in temperature is negligible.
4. The thermo-physical properties are constant, except for the density variation with temperature

Boundary Conditions:

Heat flux	0
Adiabatic condition	u = v =w = 0 *and* $\partial T/\partial r$=0
Outside temperature	Atmospheric temperature (300K)
Slip condition	No slip

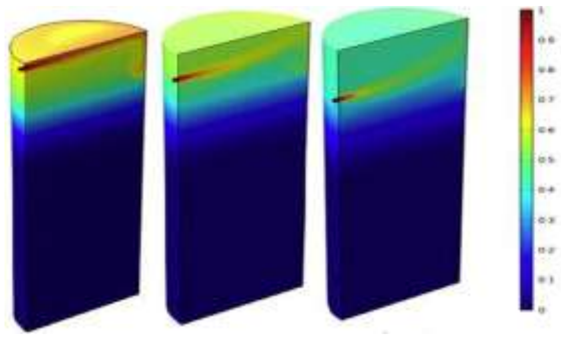
INLET CONDITION

Inlet temperature	333k
Mass flow rate	0.1 kg/s

I V. CFD SIMULATION RESULT

1 Effect of delta and way out areas on temperature shapes (L =1000 mm, D = 400 mm) for T= 100s

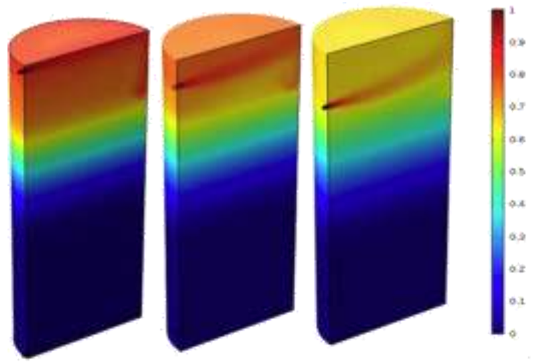
Three diverse cases with h = 30, 100, 200 mm are mulled over to watch the impact of delta and way out port areas on warm stratification in the tank. The way out and delta ports are situated on the same side divider at separation of h from the tank base and top, as delineated in Fig. 4.1. Fig. 5a and b indicates temperature shapes of the accusing procedure of Three diverse gulf and way out port positions at t = 100 and 200 s, separately. The outcomes demonstrate that there is a blending stream inside the tank because of the plane bay energy and the lightness impact. On the blending stream impact of lightness is not critical for the primary case (h =30 mm) since channel port is near the top, as gulf vitality energy is prevailing on the blending stream, increment in the bay and leave port areas, the impact of lightness on the blending stream increments and the plane force impact diminishes at various time history, as appeared in Fig. 5a and b for cases with h= 100, 200 mm. For the primary case (h = 30 mm) the warm layer grows speedier at the top, contrasted with alternate cases.



CASE 1	CASE 2	CASE 3
h =30	**h = 100**	**h = 200**

T= 100s

2. Effect of inlet and exit locations on temperature contours (L =1000 mm, D = 400 mm) for T= 200s



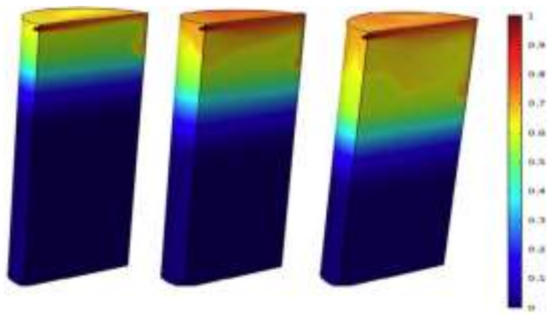
CASE 1	CASE 2	CASE 3
h =30	**h = 100**	**h = 200**

T= 200s

2. Effect of mass flow rate on temperature contours (L =1000 mm, D = 400 mm): (a) t =100 s

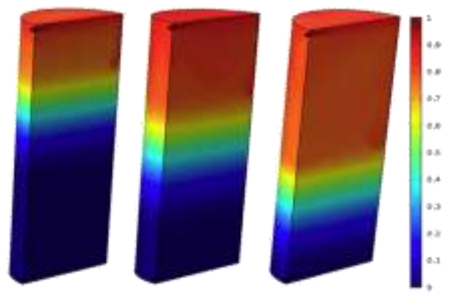
The effect of mass flow rate of the incoming fluid to the tank on the thermal stratification within the tank is investigated; here the storage tank water is initially at a uniform temperature of 30 ^oC, the inlet hot water at a temperature of 60 ^oC, from the top of the solar storage tank. Simulation was run for the mass flow rates of 0.1, 0.15 and 0.20 kg/s. Fig. 6a and b depicts the temperature distribution profile of hot water on the mid-plane of the solar hot water tank for different mass flow rates at 100 and 200 s respectively. It can be seen, flow of inlet hot water to the top of the solar tank flows along the side walls and moves in the radial direction down towards the bottom colder water layers with increase in flow rates, the incoming hot water progresses further into the tank along the axial direction, which results in the formation of vortex at this level. The inlet jet produces high hydrodynamic and thermal gradients. The incoming hot water to the storage

Tank penetrates deeper inside the tank and hit the back wall of the tank. This enhances mixing of hot and cold water or with the stratified fluid layers in the tank and causes the deterioration of the thermal stratification at the initial stages of charging operation. With progress of time, the cold fluid gradually fills the lower space of the tank while maintaining the stable thermal stratification. As expected, the thermocline forms initially when hot water enters the tank. The thermocline thickness increases as a function of the charging time. This is mainly due to the heat conduction across the thermocline, axial wall conduction and mixing at the inlet port. A close examination of the flow field reveals that the thermocline deterioration is more pronounced at low flow rates. Thermal diffusion, axial wall conduction and heat exchange with the ambient environment are rate controlled processes. The energy degradation due to these effects is directly proportional to the charging time. Hence the degradation of energy due to heat conduction across the thermocline and along the axial wall of the storage tank increases with the increase in charging time, i.e., at low flow rates. Stratification improves with increasing flow rate, up to a certain value and then remains constant. Nevertheless, as the flow rate increases, the mixing may be intense therefore, there is a competition between the effect of mixing and thermal diffusion resulting in an optimal flow rate.



CASE 1	CASE 2	CASE 3
m=0.1kg/s	m=0.15kg/s	m=0.2kg/s

T= 100s



CASE 1	CASE 2	CASE 3
m=0.1kg/s	m=0.15kg/s	m=0.2kg/s

T= 200s

It can be concluded that for various range of flow rates, a stable thermal performance is observed at high flow rates, no mixing occurs specially at the top region of the storage tank. Which also confirm those found in the literature.

V. Result validation.

The overseeing mathematical statements of mass, force and vitality protection were fathomed by utilizing the limited component strategy, in view of the presumptions made above. The overseeing mathematical statements are illuminated with weight based solver. The second request upwind discretization volume division is unraveled utilizing QUICK plan. Framework autonomy test is done before CFD calculation, network freedom study is finished by utilizing changing the cross section sizes before settling to a lattice size of 85676 cells for half of the geometry.

Zachar experimental data

TABLE NO. 1

Serial no.	Height of tank (in meter)	$(T-T_{in})/(T_{ini}-T_{in})$
1.	0.00	0.05
2.	0.05	0.05
3.	0.1	0.06
4.	0.15	0.06
5.	0.2	0.1
6	0.26	0.18
7	0.3	0.3
8	0.32	0.7

ISSN: 2321-8169
126 - 131

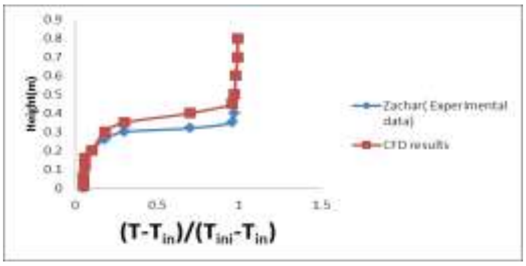
9	0.35	0.96
10	0.4	0.97
11	0.6	0.98
12	0.7	0.99
13	0.8	0.99

CFD SIMULATION RESULT

TABLE NO. 2

Serial no.	Height of tank (in meter)	$(T-T_{in})/(T_{ini}-T_{in})$
1.	0.01	0.05
2.	0.12	0.06
3.	0.16	0.06
4.	0.2	0.1
5.	0.3	0.18
6.	0.35	0.3
7.	0.4	0.7
8.	0.45	0.96
9.	0.5	0.97
10.	0.6	0.98
11.	0.7	0.99
12	0.8	0.99

RESULT VALIDATION GRAPH.



Results were verified with A. zachar [12] experimental data, Good agreement is obtained between experimental and the simulation results, which are within less than 10% of discrepancy. The difference between the present numerical results and the experimental data, due to experimental uncertainty.

VI. CONCLUSION

- The CFD model is validated with the experimental data available in the literature. Very good agreement found between the numerical results and the experiments.
- For the case with h = 30 mm as compared to other cases h > 30 mm the thermal layer develops faster on the top tank. Also, the thermal layer develops more with further increase in time
- In addition, it has been observed that the thermal degradation increases with the increase in flow rates cause of mixing at the inlet and outlet port.
- Also it has been observed that the thermal degradation increases with the increase in flow rates due to mixing at the inlet and outlet ports. It has been seen that thermal stratification increases with the initial temperature difference, due to increased density difference. However, depending of the operating and geometrical parameters, the thermocline deterioration could increase with the increase in the initial temperature difference due to increased energy transfer across the thermocline.
- Finally, CFD simulations can be used as an effective tool to optimize thermal storage tank parameters, thus it may add to the value of the solar thermal energy system performance and efficiency, by optimizing the whole solar thermal energy storage system design.

REFERENCES

[1] Mi-Soo Shin, Hey-Suk Kim, Dong-Soon Jang, Sang-Nam Lee, Young-Soo Lee, Hyung-Gi Yoon, Numerical and experimental study on the design Of a stratified thermal storage system

[2] Necdet Altuntop, Mevlut Arslan, Veysel Ozceyhan a, Mehmet Kanoglu, Effect of obstacles on thermal stratification in hot water storage tanks.

[3] Michael W. Jack a, Jan Wrobel, Thermodynamic optimization of a stratified thermal storage. 10 December 2008

[4] Y. Tian, C.Y. Zhao, A review of solar collectors and thermal energy storage in solar thermal applications. 23 December 2012

[5] Jose Fernandez-Seara, Francisco J. Uh, Jaime Sieres, Experimental analysis of a domestic electric hot water storage tank. Part I: statics mode of operation, 12 July 2006

[6] Jose Fernandez-Seara, Francisco J. Uh, Jaime Sieres, Experimental analysis of a domestic electric hot water storage tank. Part II: dynamic mode of operation, 13 July 2006

[7] Lana Kenjo, Christian Inard, Dominique Caccavelli, Experimental and numerical study of thermal stratification in a mantle tank of a solar domestic hot water system. 8 January 2007

[8] C. Cristofari , G. Notton, P. Poggi, Influence of the flow rate and the tank stratification degree on the performances of a solar flat-plate collector, 8 July 2002

[9] Mi-Soo Shin, Hey-Suk Kim, Dong-Soon Jang , Sang-Nam Lee, Young-Soo Lee, Hyung-Gi Yoon, Numerical and experimental study on the design Of a stratified thermal storage system, 21 July 2003

[10] E. Papanicolaou, V.Belessiotisa, Transient natural convection in a cylindrical enclosure at high Rayleigh numbers. 16 July **2001**