UPPSALA
UNIVERSITET

THERMAL ENERGY STORAGE

Vincent Larribe

Project in Computational Science: Report

Abstract

It is now widely accepted that renewable energy is a key concern for the future. However, fossil energy sources such as oil or gases are still much more used than the environmental friendly solutions and we face difficulties to develop renewable energy…

We can quote the two main reasons for this:

-It is more expensive to produce electricity with renewable energy than with fossil energy

-We do not chose when we want to produce more or less energy, it depends on Mother Nature's willingness. This is especially true for wind and solar power. It is thus very difficult or even impossible to adapt the supply to the demand.

One solution of the second point would be to have an efficient way to store the energy produced when we do not need it and then, to return it when the energy source is not available anymore (during the night for solar power)

In this project, I study the possibility to store the energy produced as thermal energy.

I will show which parameters are important to select an efficient system and answer some fundamental questions on the chosen system.

Note that, even if it is a "project in computational science", an important part of my work was a pre-analysis to decide what to simulate. Thus, you will find in this report both the theoretical formulae and the computational work.

Table of contents

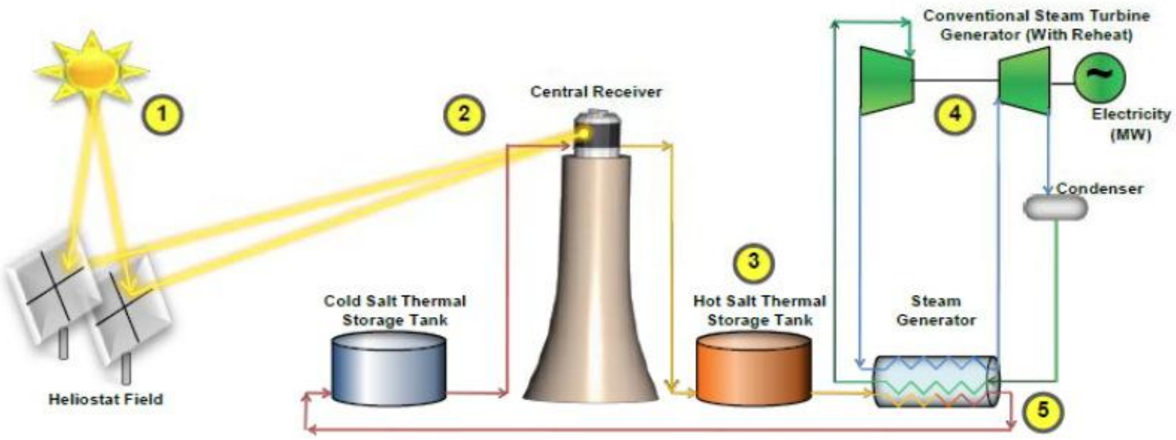
I. Presentation of the system.

The principle is very simple: we heat a fluid with the excess of energy and we take back the energy when we need it by cooling down the fluid.

a. An experimental example

Some experimental projects have already been built. One of them is Solar II in California.

Figure 1 : Scheme of the experimental power plant



The fluid used to store the energy is a mixture of melted salt. We will see later why this is a good choice.

In this system, two tanks are used, one to store the cold fluid and the other one to store the hot fluid. The fluid flows in a closed circuit; the cold fluid receives heat from the sun on his way to the hot reservoir and releases it on its way back to the cold reservoir. The energy released is used to heat steam and run a turbine to produce electricity. In this project, I will not study the electricity production part (steam generator and turbine) but I will focus on the tanks used to store the heat.

As it is an experimental system, they do not look for the most efficient one but they want to obtain conclusions from different tests. This is why they opted for two reservoirs; it allows knowing easily which amount of the stored energy is lost in the environment and which part is taken back by the heat exchanger. However, they know that using only one reservoir would be more efficient. Of course for the same volume of fluid, i.e. for the same amount of energy stored, the surface in contact with the environment is more important when using two reservoirs. This means higher losses and more material to build the reservoirs. The resulting system is more expensive and less efficient.

These reasons prove that if we want to use this system for industrial applications we should design a system with only one reservoir. This system has not been built experimentally and in order to build it we must know what the important parameters are and how to choose an efficient system.

CFD (Computational Fluid Dynamics) will help us to do this.

b. Advantages of using only one reservoir and first sizing approach

Let's have an estimation of how big is the efficiency difference between these two cases:

- Case1: one reservoir

We use one tank of volume V_1, the surface of heat exchange with the environment (losses) is S_1

- The equation of heat exchange by conduction gives us the heat flux ϕ between the fluid and the environment

$$\phi_1 = \frac{\lambda S}{e}(T_{hot} - T_{out}) \quad (1)$$

ϕ is the exchange with the environment by conduction

λ is the thermal conductivity of the material used to build the reservoir,

e is the thickness of the tank

S is the surface of the tank

T_{hot} is the temperature of the fluid

- The equation of heat exchange by radiation gives us:

$$R_1 = \varepsilon S \sigma \left(T_{hot}{}^4 - T_{out}{}^4\right) \qquad (2)$$

R is the exchange with the environment by radiation

ε is the emissivity,

S is the surface of the tank

σ is the Stefan-Boltzmann constant

- Case2: two reservoirs

We use two reservoirs which have the same capacity. When the maximum storage capacity is used, all the heated fluid is in the so called "hot reservoir" this reservoir must have the same capacity as if only one reservoir was used. Therefore, each reservoir is exactly the same as the one used in the first case. The exchange surface is then multiplied by 2 but the losses are not multiplied by 2! Indeed, the temperature in each tank is different, let's call them T_{cold} and T_{hot}

In this case, we have:

- Losses by conduction in this case:

$$\Phi_2 = \frac{\lambda S}{e}(T_{hot} - T_{out}) + \frac{\lambda S}{e}(T_{cold} - T_{out})$$

We compare the losses:

$$\frac{\Phi_2}{\Phi_1} = \frac{(T_{hot} - T_{out}) + (T_{cold} - T_{out})}{(T_{hot} - T_{out})}$$

$$\frac{\Phi_2}{\Phi_1} = 1 + \frac{(T_{cold} - T_{out})}{(T_{hot} - T_{out})}$$

To have an estimation, we take the values of SolarII:

$$T_{hot} = 550°C$$
$$T_{cold} = 250°C$$

$$\frac{\Phi_2}{\Phi_1} = 1{,}45$$

- Losses by radiation in this case:

$$R_2 = \varepsilon S\sigma\left({T_{hot}}^4 - {T_{out}}^4\right) + \varepsilon S\sigma\left({T_{cold}}^4 - {T_{out}}^4\right)$$

$$\frac{R_2}{R_1} = 1 + \frac{\left({T_{cold}}^4 - {T_{out}}^4\right)}{\left({T_{hot}}^4 - {T_{out}}^4\right)}$$

To have estimation, we take the values of Solar II:

$$T_{hot} = 550°C$$
$$T_{cold} = 250°C$$

$$\frac{\Phi_2}{\Phi_1} = 1{,}45$$

$$\frac{R_2}{R_1} = 1{,}15$$

To have an estimation of these losses, we take S=500m², λ=0,035W.K^{-1}.m^{-1}, e=20cm and ε=0,02 (we will estimate more precisely this parameters later in the report).

R_1≈400kW R_2≈460kW

ϕ_1≈100kW ϕ_2≈145kW

These estimations only take into account the losses in the reservoirs, in the second case, many losses will occur in the pipes. These quick estimations show that for an industrial application we will prefer using only one reservoir.

c. The important parameters

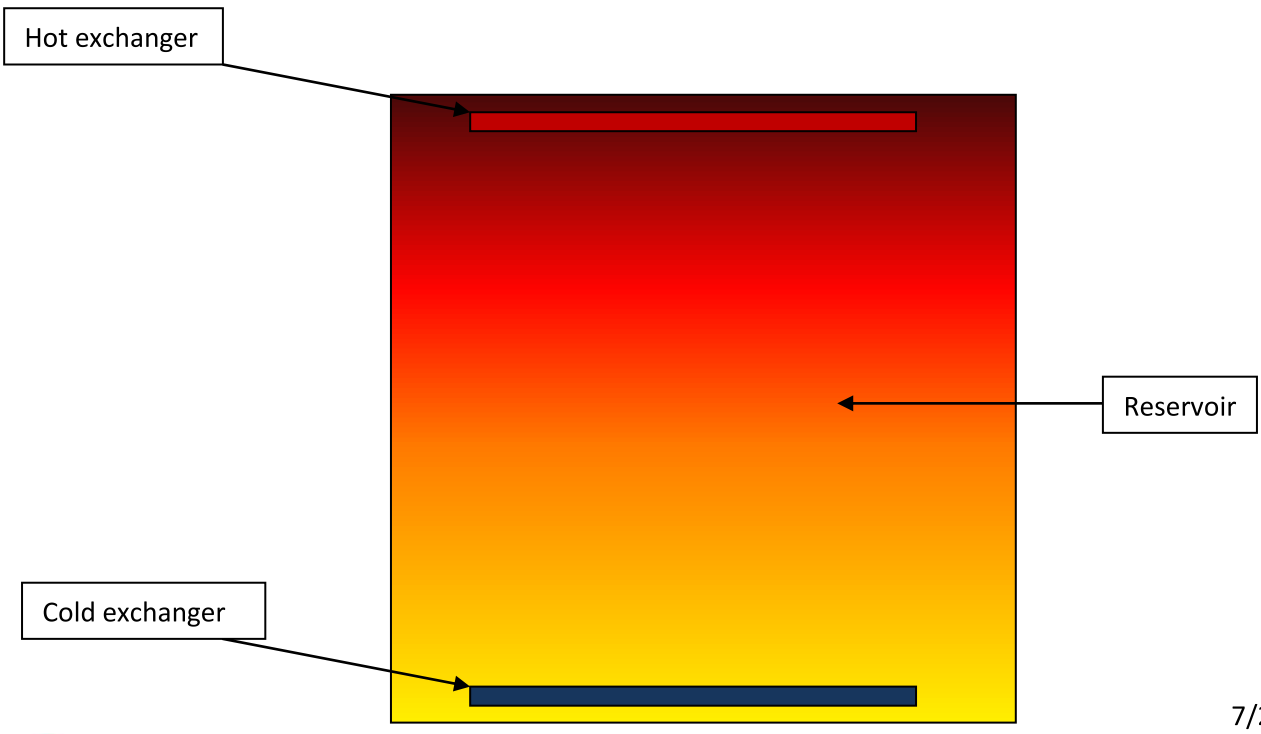
There are many important parameters to guarantee the efficiency of the system. The most important ones are the viscosity, the thermal capacity, the compressibility, the density of the fluid, the thermal conductivity of the fluid and of the reservoir's wall. The shape, the size and the place of heat exchangers in the tank are important as well.

The final goal of this project is to do calculations and simulations to evaluate how large the influence of these parameters is on the global efficiency, to choose values for them and check the behavior of the system with the chosen parameters.

The simulations tools used are OpenFOAM and Fluent. OpenFOAM has been used to create the geometry and the mesh as it is a freeware and Fluent for its post-processing efficiency.

As we first assume the flow to be slow enough to be laminar, the 3D effects can be neglected and the simulations can be done in 2D. In these simulations, we make sure that the velocity remains low and this assumption true.

Figure 2: system simulated



In this first system, we consider a cylindrical reservoir (in 2D) with two heat exchangers, one is situated in the coolest part of the reservoir and brings heat to the fluid, and the other is situated in the hottest part which absorbs the heat by cooling the fluid.

Future simulations could include also the effects of varying the shape of the reservoir and the position of the heat exchangers.

II. Choosing the parameters

a. Properties of the fluid

Choosing a “good” fluid for the system means to determine the influence of the characteristics of the fluid on the processes.

Respectively, we must know what is important in the phenomenon occurring in the reservoir:

The first and the most important thing is the amount of energy storable in such a reservoir, we aim at maximizing the amount of energy described as:

$$E = V.\rho.Cp.\Delta T \quad (3)$$

Here is the amount of energy stored, V is the volume of the reservoir, ρ is the mass per volume unit of the fluid, Cp is the thermal capacity of the fluid, ΔT is the difference between the highest and the lowest temperature of the fluid

From the expression for E, we see, that in order to maximize E, all four parameters V, ρ, Cp and ΔT have to be maximized

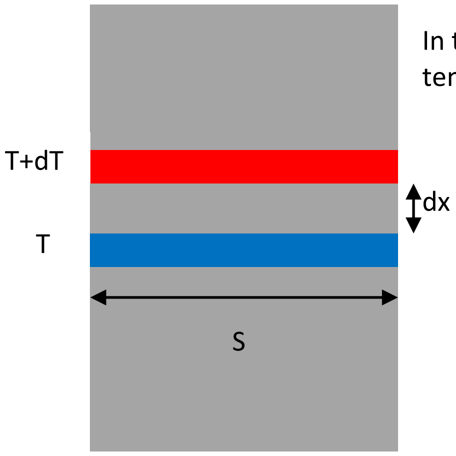
The maximum value of ΔT is the difference between the vaporization temperature and the liquefaction temperature. Therefore, we have to choose a heavy fluid, with a big thermal capacity, which remains liquid in a big range of temperature and in a big reservoir.

Further, we want the hot exchanger (which absorbs energy from the fluid) to be in contact with the hottest fluid and the cold exchanger (which brings heat to the fluid) to be in contact with the coolest part of the fluid for the exchangers to be more efficient. By natural laws, the hottest fluid “floats” above the coolest. This is induced by the dilatation of the fluid, the hottest fluid expands and thus, a given volume of hot fluid is lighter than the same volume of cold fluid.

As we have this configuration (the hot fluid floating above the cold one), we want to preserve it as long as possible. Indeed, the conduction phenomena in the fluid will lead the temperature to become homogeneous. This means that the time for temperature to become the same at the top and at the bottom must be as big as possible.

Below, we estimate the time needed for the temperature to become homogeneous in the reservoir without heat exchanger.

We represent the cylinder in 2D with two sections S separated by a distance of length dx



In the upper section (in red) temperature is higher by dT than the temperature of the lower section (in blue)

The heat flux exchange by the two sections is

$$\phi = \frac{\lambda S}{dx} dT$$

The energy exchanged during time dt is thus

$$dE = \phi dt$$
$$dE = \frac{\lambda S}{dx} dT.dt$$
$$dE.dx = \lambda S dT.dt$$

After integrating along the length of the cylinder, we obtain

$$\Delta E.L = \lambda S \Delta T t \quad (4)$$

Where t is the time needed for the exchange of energy ΔE to have occurred between the top and the bottom of the reservoir.

If we evaluate the time needed for the temperature to be independent of the altitude, we consider ΔE to be the difference of temperatures at the top and the bottom of the reservoir, namely,

$$\Delta E = mCp\Delta T \quad (5)$$

With m being the mass of the fluid in the reservoir, Cp the thermal capacity and ΔT the temperature difference of the liquid between the top and the bottom. Combining (4) and (5), we obtain

$$t = \frac{mCpL\Delta T}{\lambda S \Delta T}$$
$$m = \rho.V = \rho SL$$
$$t = \frac{\rho L^2 Cp}{\lambda} \quad (6)$$

This formula finally shows that we have to maximize ρ, Cp, the altitude of the reservoir L and to minimize λ. This formula can be used to estimate a length δ characteristic of the conduction during a period t.

$$\delta = \sqrt{\frac{t.\lambda}{\rho.Cp}} \quad (7)$$

Another phenomenon we have to take into account is the Rayleigh-Benard's instability. The theory gives us a condition for the hottest fluid situated under the cold fluid to go up. We obtain a mixing flow. This condition is given by Ra>1706 where Ra is the Rayleigh number, defined as

$$Ra = \frac{Archimede\ force}{viscous\ force} = \frac{g\beta.\Delta T.L^3}{\nu\alpha}$$

Where g is the gravity, β is the thermal dilatation of the fluid, ΔT is the temperature difference between the top and the bottom of the reservoir, L is the altitude of the reservoir, ν is the kinematic viscosity, α is the thermal diffusivity.

The condition Ra>1706 comes from an empirical formulae determined by experimental results.

In addition to the previous analysis, we have to make sure that Ra>1706 (the higher the Rayleigh number, the higher the velocity of the flow induced).

Considering this analysis, we can now choose a fluid with rather good parameters for the simulations. Among the fluid, the so called "melted salts" present the best properties.

For example, two candidates are:

- Fluid 1: $NaNO_2$-$NaNo_3$-KNO_3 (7%-40%-53%)

  $\rho=2000 kg/m^3$
  $Cp=1506\ Jkg^{-1}K^{-1}$
  $\lambda=0,3\ Wm^{-1}K^{-1}$
  $\nu=2,5.\ 10^{-6}\ m^2s^{-1}$
  $\beta=10^{-6}$
  $\alpha=1.10^{-7}$
  $T_{min}=250°C$

  $$\frac{E}{V} = 1,66.10^9 J.m^{-3}$$
  $$\delta = 9,3cm$$
  $$Ra = 2,17.10^{13}$$

- Fluid 2: LiF-NaF-ZrF_4 (42%-29%-29%)

  $\rho=2800 kg/m^3$
  $Cp=1460\ Jkg^{-1}K^{-1}$
  $\lambda=0,4\ Wm^{-1}K^{-1}$
  $\nu=4,5.\ 10^{-6}\ m^2s^{-1}$
  $\beta=10^{-6}$
  $\alpha=9,8.10^{-8}$
  $T_{min}=460°C$

  $$\frac{E}{V} = 1,39.10^9 J.m^{-3}$$
  $$\delta = 9,2cm$$
  $$Ra = 1,03.10^{13}$$

The comparison with the properties of water for exemple shows that they are both good candidates

Water:

ρ=1000kg/m^3
Cp=4185 Jkg^{-1}K^{-1}
λ=0,6 Wm^{-1}K^{-1}
ν=1. 10^{-6} m^{2}s^{-1}
β=2,4.10^{-4}
α=1,44.10^{-7}
T_{min}=0°C
T_{max}=100°C

$$\frac{E}{V} = 4{,}19.\,10^8 J.m^{-3}$$
$$\delta = 11{,}1cm$$
$$Ra = 1{,}78.\,10^{15}$$

This proves that melted salts are better than water for this application. According to the studies already done for previous experimental systems, it is actually the best candidate among a very large quantity of fluids.

Both fluids 1 and 2 give Ra>>1706. Fluid 1 is better for energy per volume E/V and they give a similar result for δ.

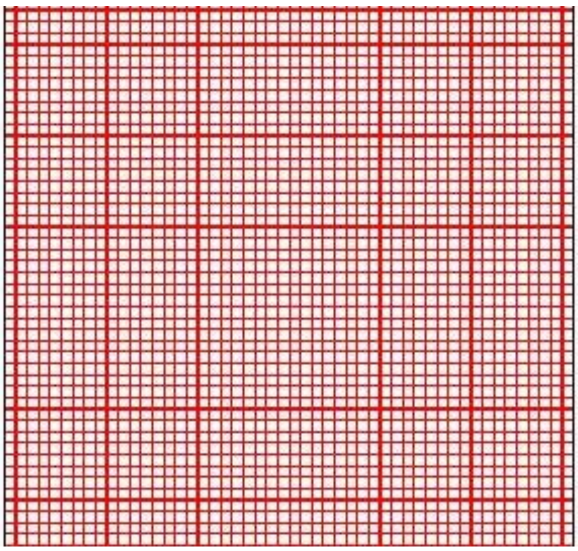
We thus choose fluid 1.

b. Code validation

Before trying to have results simulation, we must be sure the simulation tools are reliable.

To do this, we create a very simple geometry (a square) with orthogonal regular mesh.

Figure 2: orthogonal mesh in a square domain



To make sure the fluid is moving (the hottest "floating" on the coolest), we impose the following boundary conditions: The left wall is at T=450K and the right wall at T=460K

Figure 3 shows the initial temperature field

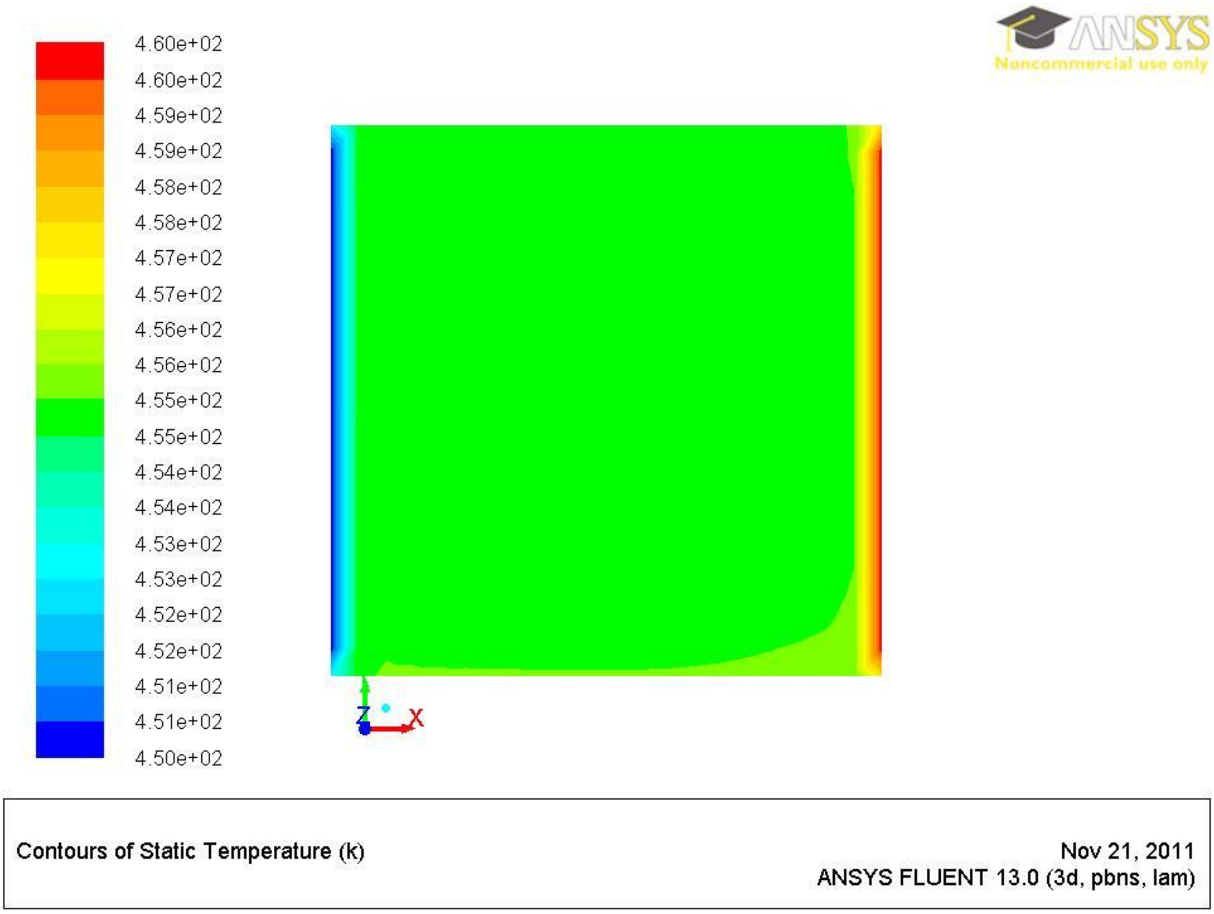
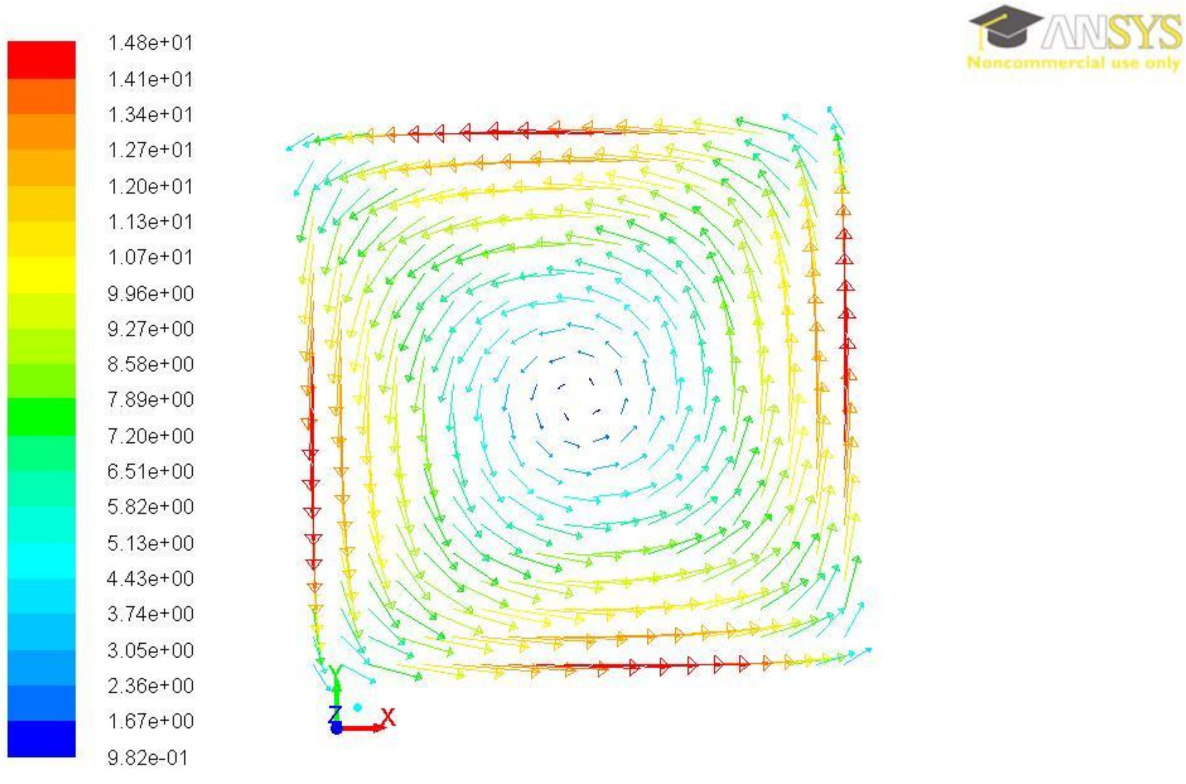


Figure 4 shows the velocity field



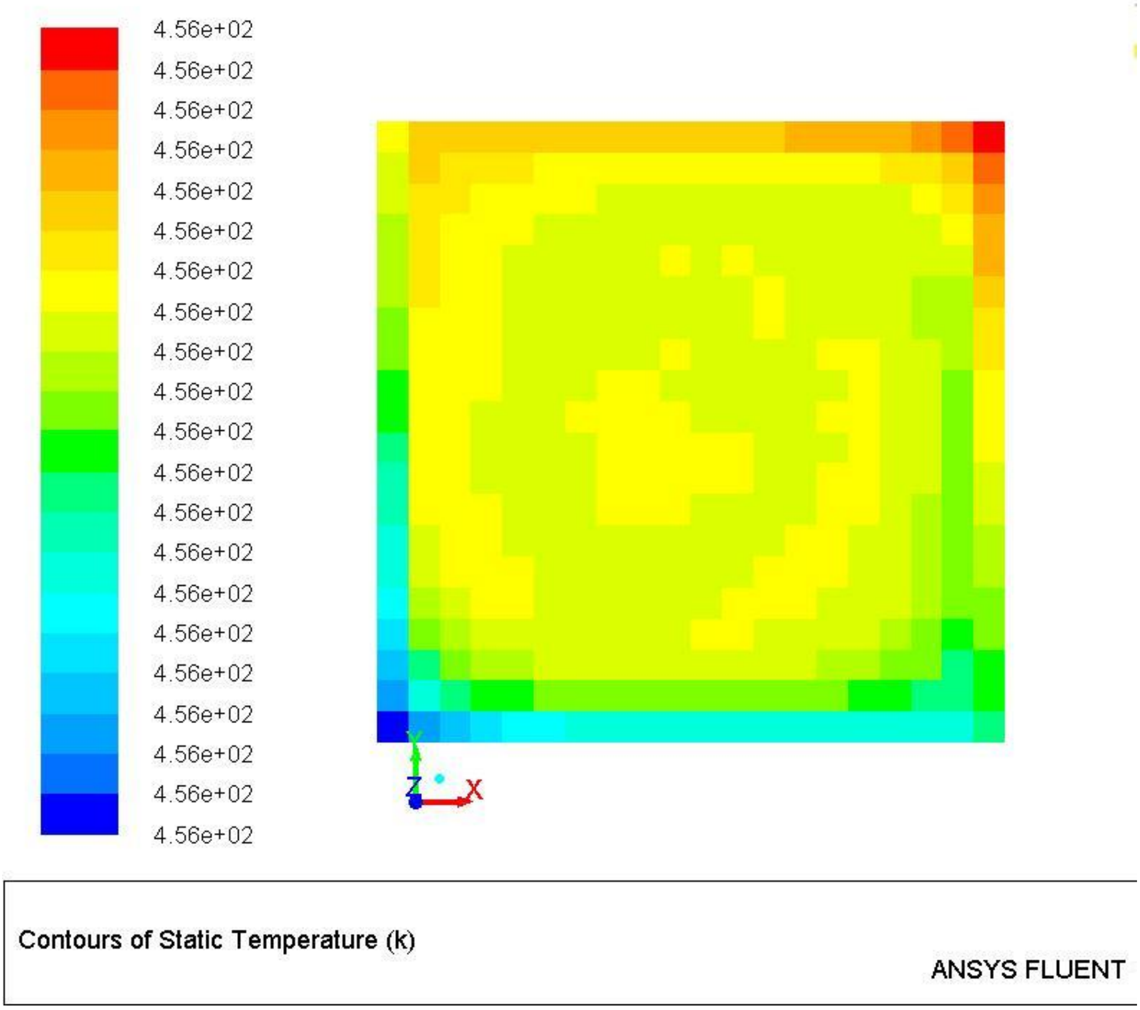
Velocity Vectors Colored By Velocity Magnitude (m/s) Nov 28, 2011
ANSYS FLUENT 13.0 (3d, pbns, lam)

According to the predictions, the cold fluid is going under the hot one.

This is the first simulation result in the report and it is the occasion to remind how the discrete model has been generated.

Fluent uses FDM (Finite Difference Method) and the time discretisation is an implicit scheme

Figure 5 shows the temperature field



These figures provide evidence that the solving tools used are compatible with the problem.

We can now check if the boundary conditions agree with theory.

Figure 6: x velocity on a vertical line

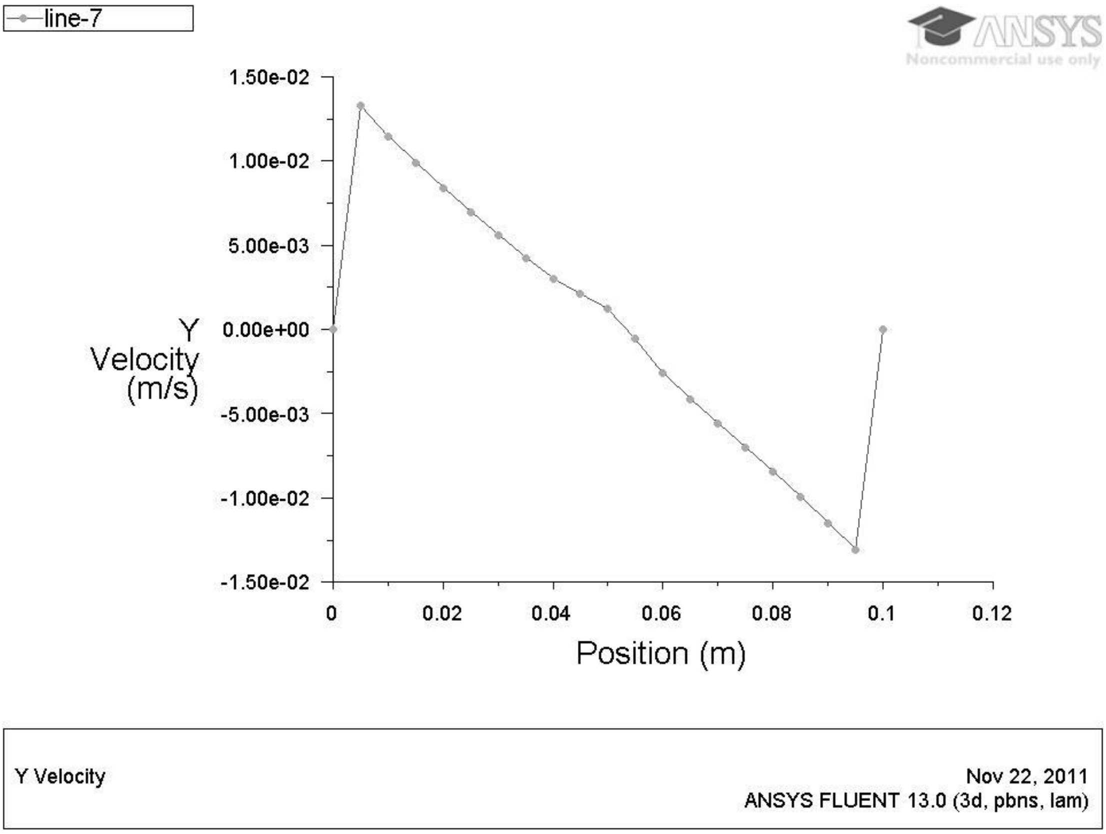
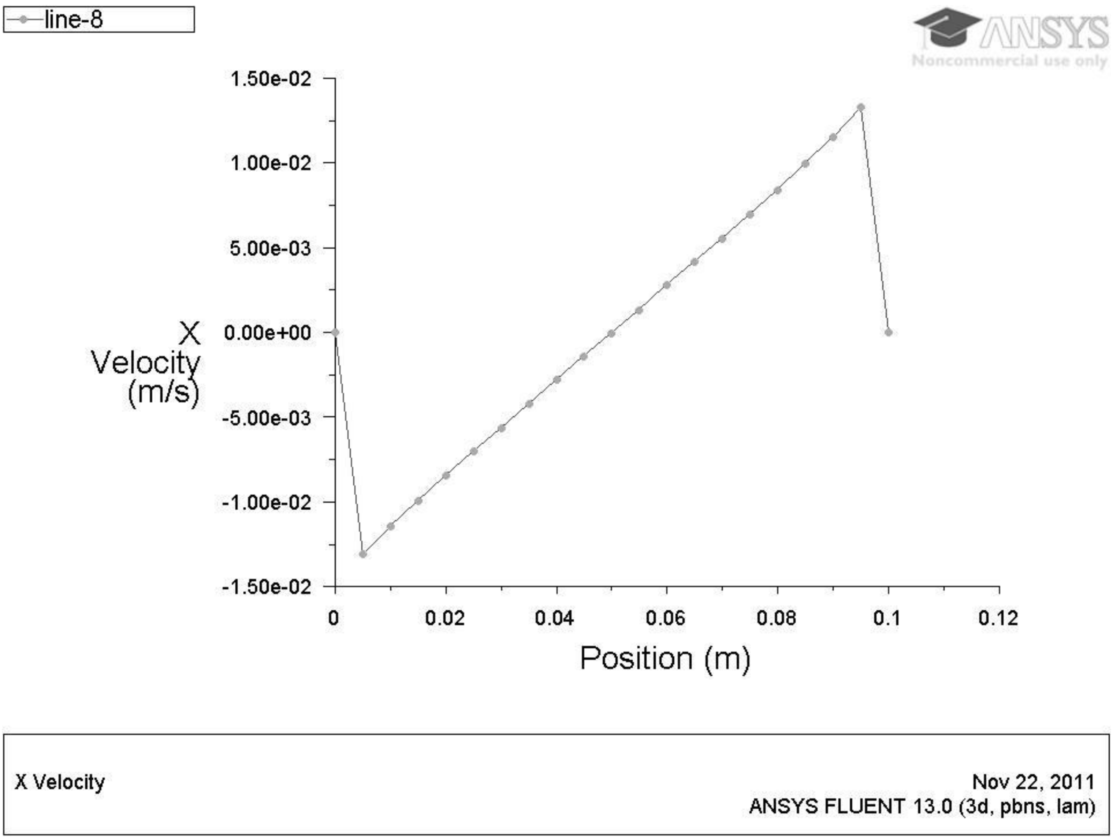


Figure 7: y velocity on a vertical line



The average velocity on both lines is zero and the “no slippery conditions” on the walls is respected as the velocity is zero at the nodes corresponding to boundaries.

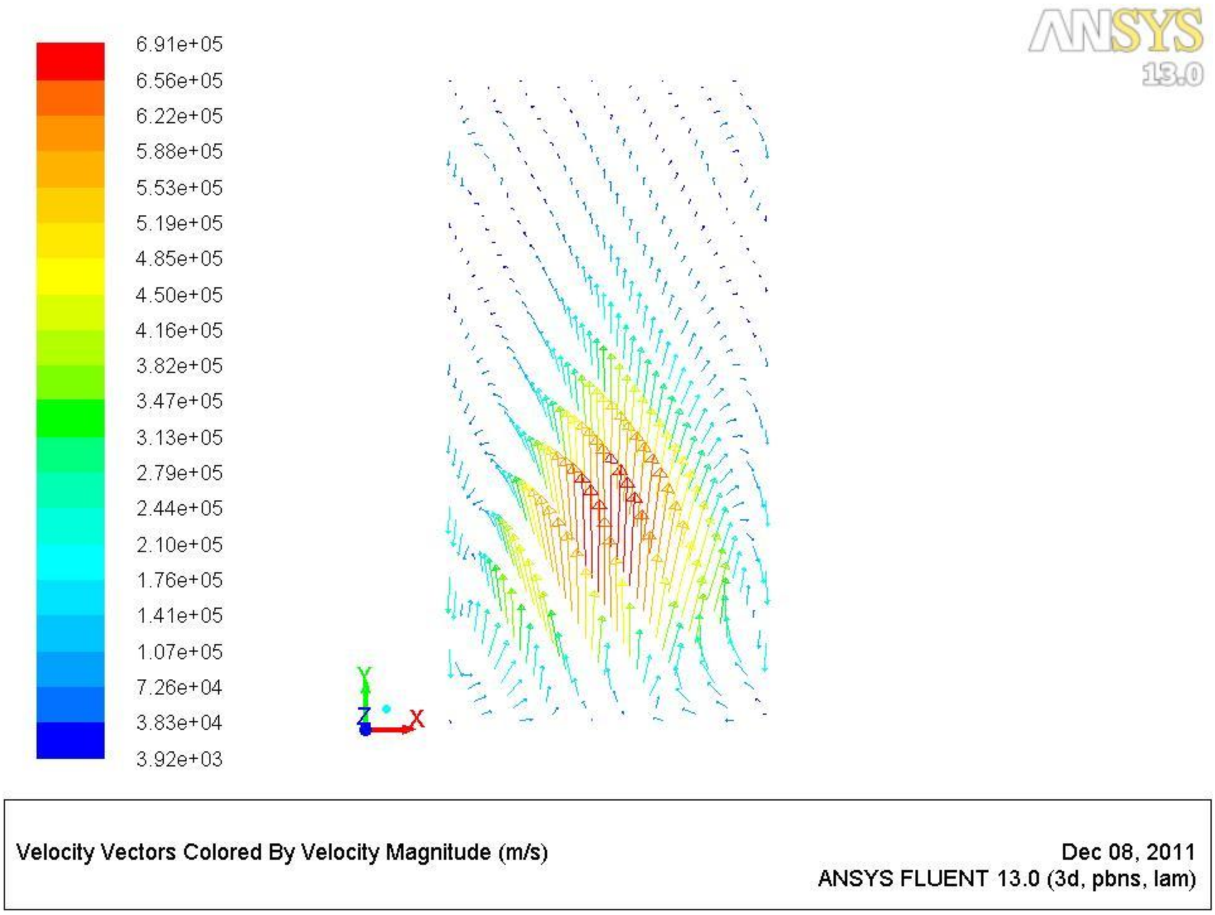
c. Validation of Choices

We now simulate the flow, considering that the bottom and the top are the heat exchangers and the walls are supposed to be adiabatic.

In this section I verify that the different assumptions and choices made previously are realistic.

Figure 8: the mesh used

Figure 9: velocity field in the reservoir



The software gives us a velocity average of 70000m/s which is really high and is not realistic. It obviously comes from a problem in the solving tools and has nothing to do with the reality. One of the assumptions made in order to choose the tools might be wrong.

We estimate the Reynolds number defined as

$$Re = \frac{Ud}{\nu}$$

U is the average velocity; $U=70000ms^{-1}$ from the result given by Fluent

d is a characteristic length of the system; d=0,1m (length of the reservoir in the chosen mesh)

ν the kinematic velocity; $\nu=2{,}5.10^{-6}$ in the simulating system from the fluid chosen previously

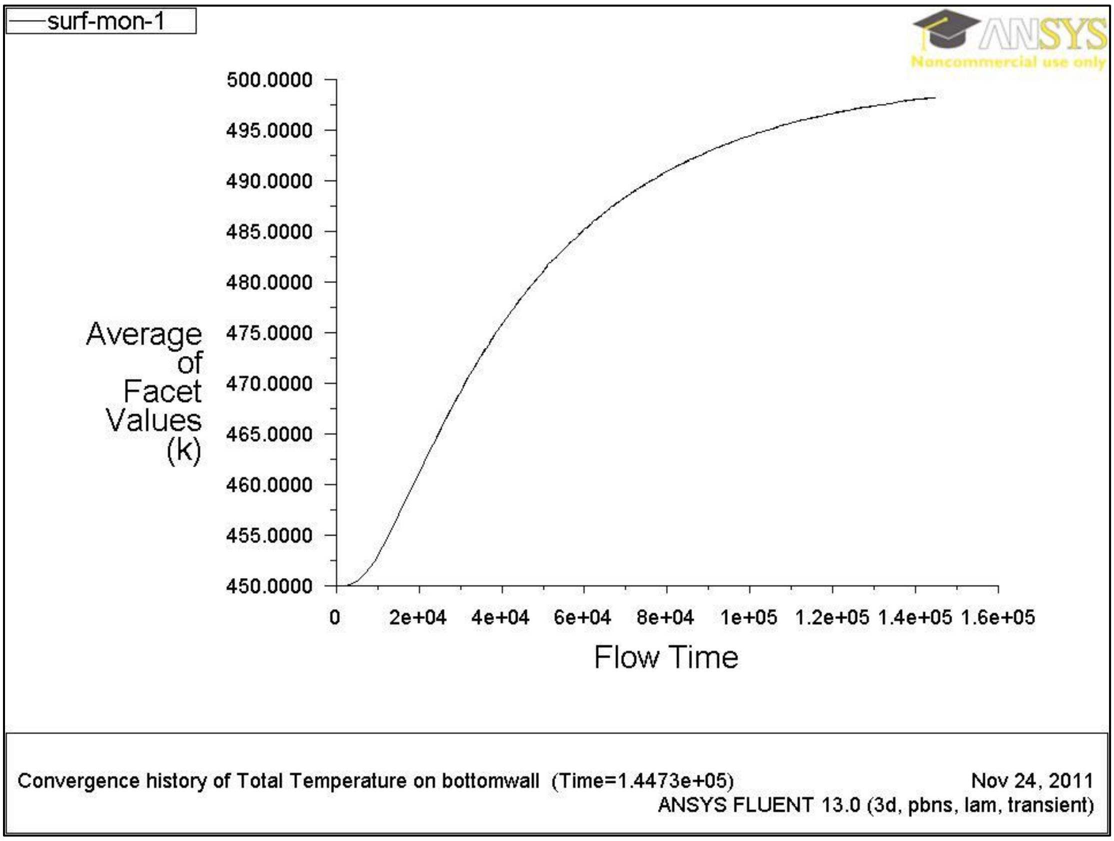
We obtain $Re=3{,}20.10^{7}>>2000$

Re>2000 , the flow is not laminar, 3D effects are involved and thus, the simulation cannot be done in 2D.

Then, we check if the time needed to have a constant temperature in the reservoir match with theory.

To do it, we create a virtual line (no wall) in the middle of the domain and observe how long it takes for the temperature to be constant at this line.

Figure 10 : temperature in function of time in the middle of the domain



The previous formula gives us:

$$\delta = \sqrt{\frac{t.\lambda}{\rho.Cp}}$$

$$t = \frac{\rho.Cp.\delta^2}{\lambda}$$

According to the theory and the formula (6), t=1,1.10^5s

On the previous graph, Fluent gives us t=1.4.10^5s

This two results are close enough to consider the model to be right.

III. Results

a. Sizing the reservoir

We consider two cases: a system with one reservoir and another system with two reservoirs

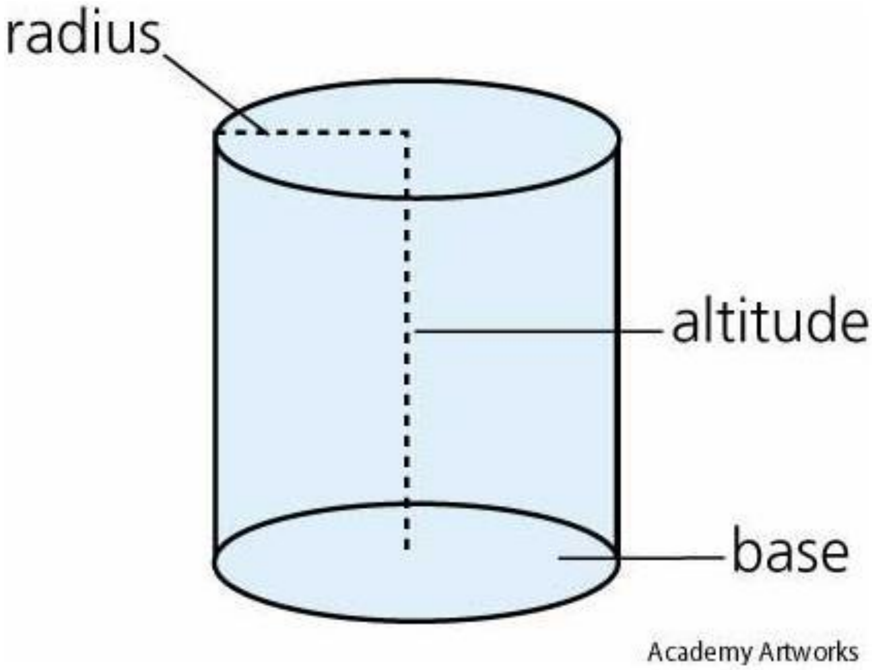
We now estimate the difference in efficiency between these two cases.

First of all, we need to have the exchange surface in both cases.

- **Case 1: one reservoir is used to store the energy**

We denote the radius and the altitude of the cylindrical reservoir by R and H, correspondingly.

Figure 11: Scheme of the reservoir



The most efficient system is such as for a given volume, the surface of exchange is the smallest.

We choose a volume V_1 for the reservoir depending on which amount of energy we need to store (see below for the calculation of this volume):

$$V_1 = \pi R^2 H$$

$$S_1 = 2\pi RH + \pi R^2 + \pi R^2 = 2\pi R(R + H)$$

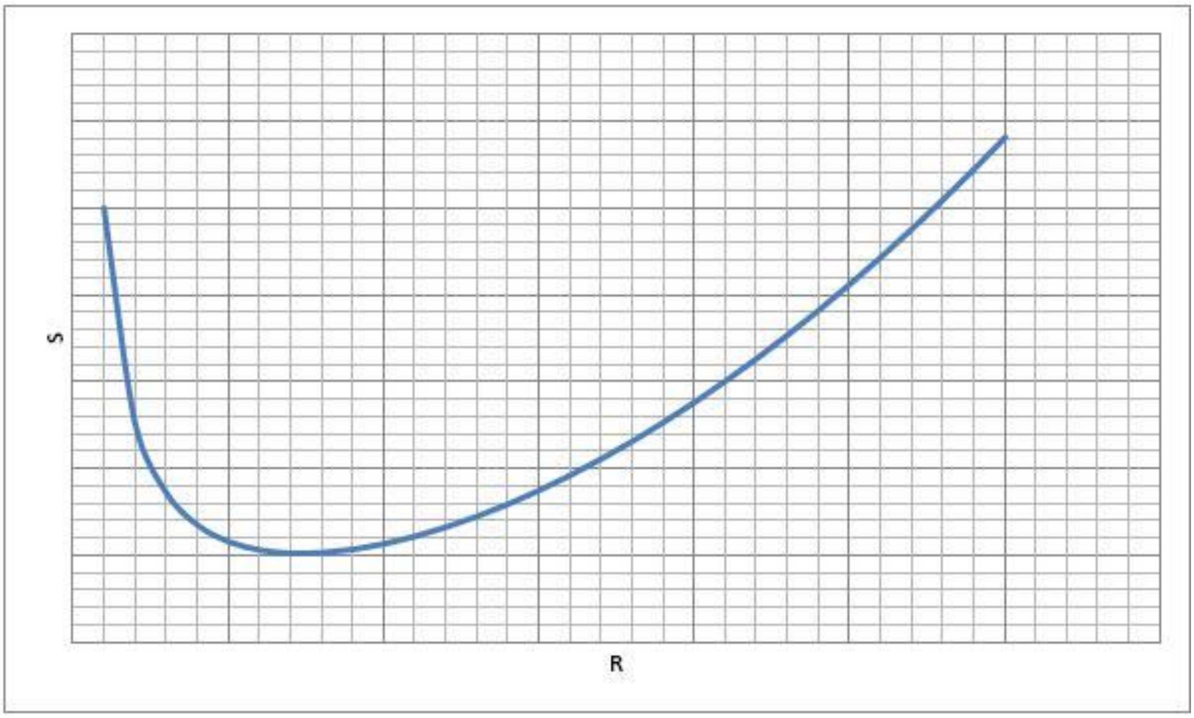
Where S_1 is the exchange surface.

We minimize S_1 for a given volume V_1

$$H = \frac{V_1}{\pi R^2}$$
$$S_1 = 2\pi R\left(R + \frac{V_1}{\pi R^2}\right)$$
$$\frac{dS_1}{dR} = 2\pi\left(2R - \frac{V_1}{\pi R^2}\right)$$
$$\frac{dS_1}{dR} > 0 \quad if \quad 2R > \frac{V_1}{\pi R^2} \quad i.e. if \quad R > \left(\frac{V_1}{2\pi}\right)^{1/3}$$
$$\frac{dS_1}{dR} < 0 \quad if \quad 2R < \frac{V_1}{\pi R^2} \quad i.e. if \quad R < \left(\frac{V_1}{2\pi}\right)^{1/3}$$

We have a plot like following:

Figure 12: plot of the section in function of the radius for a given value



The minimum is reached for $R = \left(\frac{V_1}{2\pi}\right)^{1/3}$.

For this configuration, we have $H = \frac{V_1}{\pi R^2} = 2^{2/3}.\left(\frac{V_1}{\pi}\right)^{1/3}$.

Thus, the optimum configuration is reached for:

$$\frac{R}{H} = \frac{1}{2}$$

We then have:

$$V_1 = \pi R^2 H = 2\pi R^3$$
$$S_1 = 2\pi R(R + H) = 3\pi R^2$$

- **Case 2: two reservoirs are used to store the energy**

Now, we are in the second case, we chose a different notation and we note now V_2 , the volume of one reservoir and S_2 the surface of one reservoir.

Both tanks must have the same capacity, we choose the best configuration for each one, and therefore, they have the same dimension: r for the radius and h for the altitude.

$$V_2 = \pi r^2 h$$
$$S_2 = (2\pi rh + \pi r + \pi r^2) = 2\pi r(r + h)$$

As we still want the exchange with the environment to be as little as possible, we still have: $\frac{r}{h} = \frac{1}{2}$

In the same way as previously, we obtain:

$$V_2 = 4\pi r^3$$
$$S_2 = 6\pi r^2$$

b. Efficiency

The results below are given when one reservoir

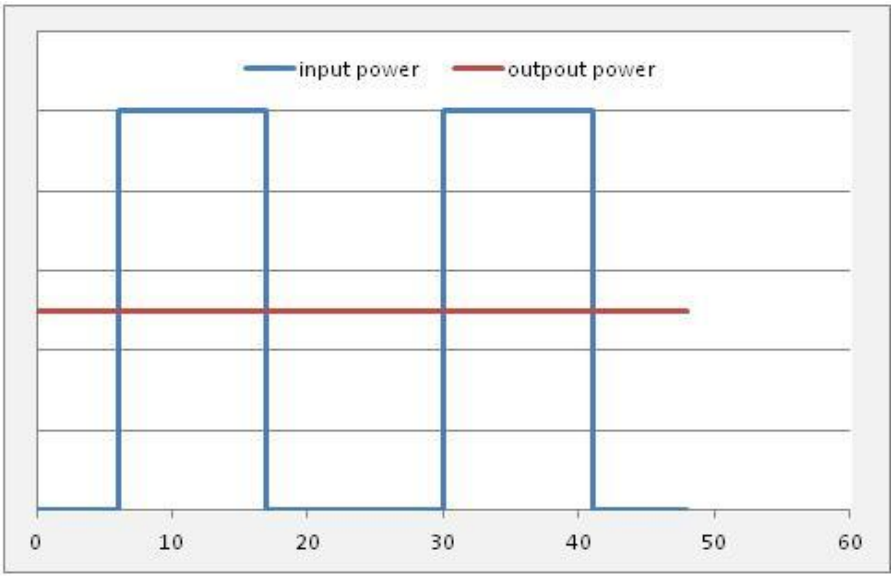
As shown in II.c (en of page 17), the simulation cannot be done in two dimensions, we thus do it in three dimensions.

The reservoir is a cylinder with the following properties:

$$\Delta T = 300°C$$
$$V = 1000 m^3$$
$$H = 10{,}70 m$$
$$R = 5{,}35 m$$

We then have the same storage capacity as Solar II, with a storage capacity of 10 MW during one day.

Figure 13: condition simulated

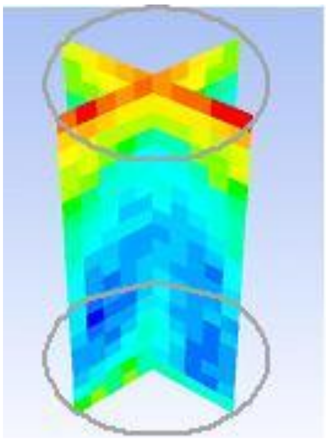


This figure shows in blue the power given to the fluid by the sunlight and in red the output power (the power given by the fluid)

In this case, we assume that the production is at the maximum during 12 hours and straightly at the minimum during the next 12 hours. The goal is to have a constant output power.

In the simulation, this means a flow of -10MW on the top and alternatively 0/20 MW at the bottom.

Figure 14: Temperature field in the cylinder



A cut of the cylinder shows in red the hottest fluid (at the top) and the coolest fluid in blue.

The flux report on Fluent gives us the losses: 500kW. This is really low compared with the 10 MW average output power on a whole day.

The system is thus very efficient.

Conclusion

The system studied has a high efficiency, it is a safe technology as the salt is chemically inactive and moreover, it overcomes the main difficulty linked to the use of solar power. And the use of only one reservoir is more efficient than using two.

Finally, the goal of this project is partially achieved as we have an estimation of the losses calculated by Fluent and these are very small. But seen the amount of work done, the results obtained seem to be really few and this is the most disappointing part… I would have liked to compare the influence of different parameters with computing science but I only could choose analytically good properties and check with Fluent that it worked. This is mainly because of the problems encountered with the mesher and then for the 2D simulation.

Even if the results are partially satisfying because I expected to have more time to study more parameters such as the material of the reservoir, this was a very good experience as it was my first experience in such a project. It forced me to take decisions concerning the coming up issues. It can be a good transition to the professional world. It was the first time I worked without precise directives from a teacher and this brought me a lot.

TABLE OF FIGURES

BIBLIOGRAPHY

Documentation of softwares:

- Fluent:
  http://www.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/ANSYS+Fluent

- OpenFoam:
  http://www.openfoam.com/

Webography:

https://www.sharcnet.ca/Software/Fluent13/help/flu_th/flu_th_sec_vof_eq.html
http://www.energy.ca.gov/sitingcases/solartwo/
http://lisas.de/projects/alt_energy/sol_thermal/powertower.html
http://webbook.nist.gov/chemistry/fluid/
http://home.earthlink.net/~bhoglund/whatsMoltenSal.html7
http://www.ardi-rhonealpes.fr/c/document_library/get_file?uuid=043f4037-0598-4dfb-a6c5-a05a39da2a70&groupId=10136