# POLITECNICO DI TORINO

Corso di Laurea Magistrale

in

Renewable energy systems

## Group project report

Optimal design of a heat storage based on a Phase Change Material (PCM) for the coupling with a district heating system.


Group members:

Lavilletta Marco

Shi Yuwei

Vair Federico

Zhu Qifan


Accademic Year 2020/2021

Outline of the report:

1- Introduction

2- Steady-state optimization considering a 2D horizontal cross section

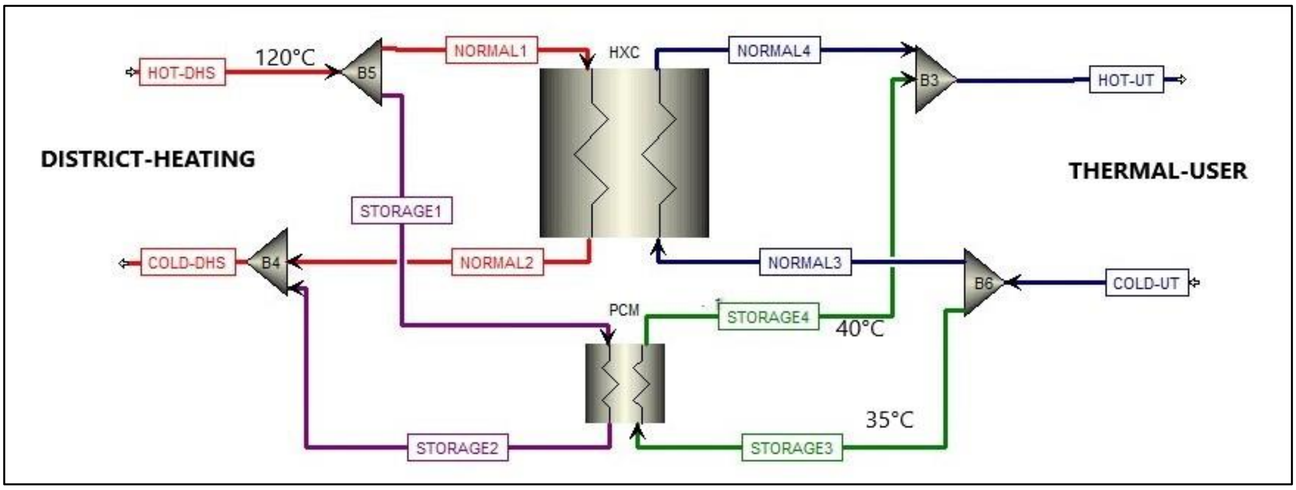
3- Transient optimization considering a 2D horizontal cross section

4- Simulation considering a 2D vertical cross section

5- Design improvement considering a 2D horizontal cross section

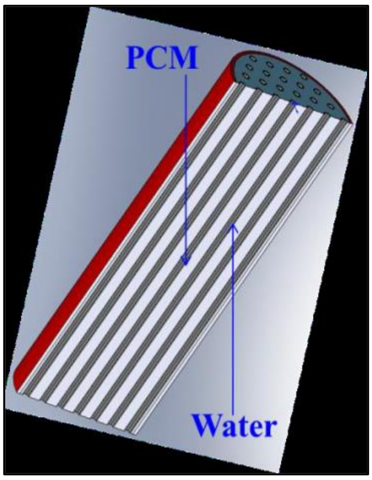
# 1- Introduction

The goal of this practice is to optimize the design of a heat storage unit characterized by a phase change material which is installed in a district heating. The idea is to store thermal energy when the user's thermal demand doesn't match with the thermal generation. The overall configuration of the system it's represented in the scheme below:



*(possible installation of the PCM heat storage unit in a district heating layout)*

The storage unit has a shell-and-tube structure: inside the unit a great number of smaller channels act as a path for the hot pressurized water coming from the district heating (T = 120°C) during the charging phase and for warm water (T = 35°C) coming back from the thermal users' radiant floor heating circuit in the discharging phase. The phase change material (from now on PCM) is in the volume outside these channels.




In order to proceed with the numerical analysis of the unit, a sub-channel analysis will be performed ("Analisi di sottocanale"): only a portion of a single channels will be considered with a defined external volume of PCM characterized by a diameter of $d_{pcm} = 0.15\ m$. The internal pipe has an external diameter of $d_{channel} = 0.01\ m$ and its thickness can be neglected.

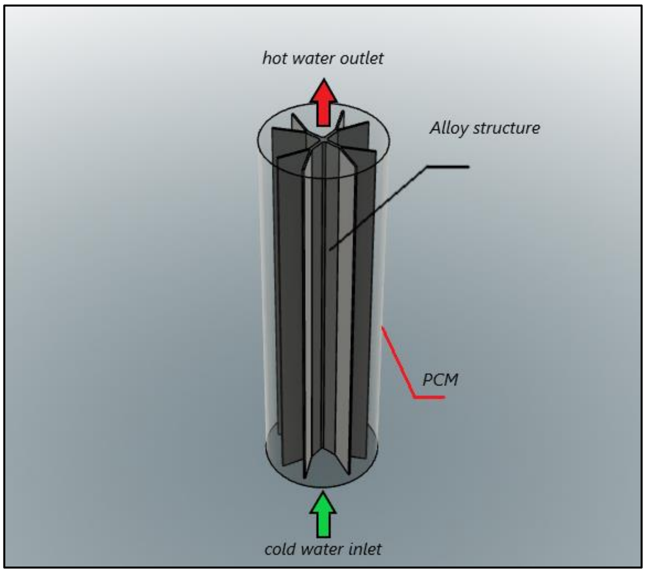
Since the PCM material is characterized by a poor thermal conductivity ($k_{PCM} = 0.15 \frac{W}{mK}$) the heat transfer is enhanced by the installation of Y-shaped fins made of aluminum alloy with the following material properties:

-density $\rho_{alloy} = 2750 \frac{kg}{m^3}$;

-thermal conductivity $k_{alloy} = 180\,\frac{W}{mK}$ ;

-specific heat $cp_{alloy} = 880\,\frac{J}{kgK}$ ;

In the following figure is presented the particular geometry of the sub-channel:




*(sub-channel scheme during the discharging phase: water is flowing in the inner tube from which the alloy fins start)*

For the analysis these properties of PCM and water will be considered:

-PCM density $\rho_{PCM} = 800\,\frac{kg}{m^3}$ ;

-Latent heat capacity (melting-solidification) of PCM $L_{PCM} = 200\,\frac{kJ}{kg}$ ;

-Melting temperature range (since it's an alloy and not a pure material) of PCM $T_{melting} = 56 - 64\,°C$ ;

-Specific heat (solid-liquid) of PCM $cp_{PCM} = 840\,\frac{J}{kgK}$ ;

-Water density $\rho_{water} = 990\,\frac{kg}{m^3}$ ;

-Specific heat of water $cp_{water} = 4185\,\frac{J}{kgK}$ ;

-Thermal conductivity of water $k_{water} = 0.6\,\frac{W}{mK}$ ;

-Dynamic viscosity of water $\mu_{water} = 0.0008\,\frac{Pa}{s}$ ;

**The optimization of the heat storage unit is performed at a component level considering the geometry of the fins.**

Since the heat transfer process occurring in the heat storage unit sub-channel is complex and requires a transient analysis, the optimization will be performed at four different levels considering additional hypothesis and simplifications.

# 2- Steady-state optimization considering a 2D horizontal cross section

An equivalent steady-state condition is considered at this level of optimization where we image to transfer all the latent heat of the PCM into the water in a defined time of $\Delta t = 3600\ s$ for a specific density.

This aspect is modeled with a volumetric heat source term in the PCM, that during the process will not experience a variation of its temperature, due to the equivalent steady-state hypothesis. The source term will be defined as:

$$q''' = \frac{L}{\Delta t}\rho_{PCM} = \frac{200000\,\frac{J}{kg}}{3600\ s}\cdot 800\,\frac{kg}{m^3} = 44444.44\ \frac{W}{m^3}$$

For what concerns the geometry of the analysis, we image to make an horizontal section of the sub-channel at half of its height. The temperature of the inner tube will be considered equal to the temperature of the water $T_{tube} = T_{water} = 37.5°C$, imposing a Dirichlet boundary condition.

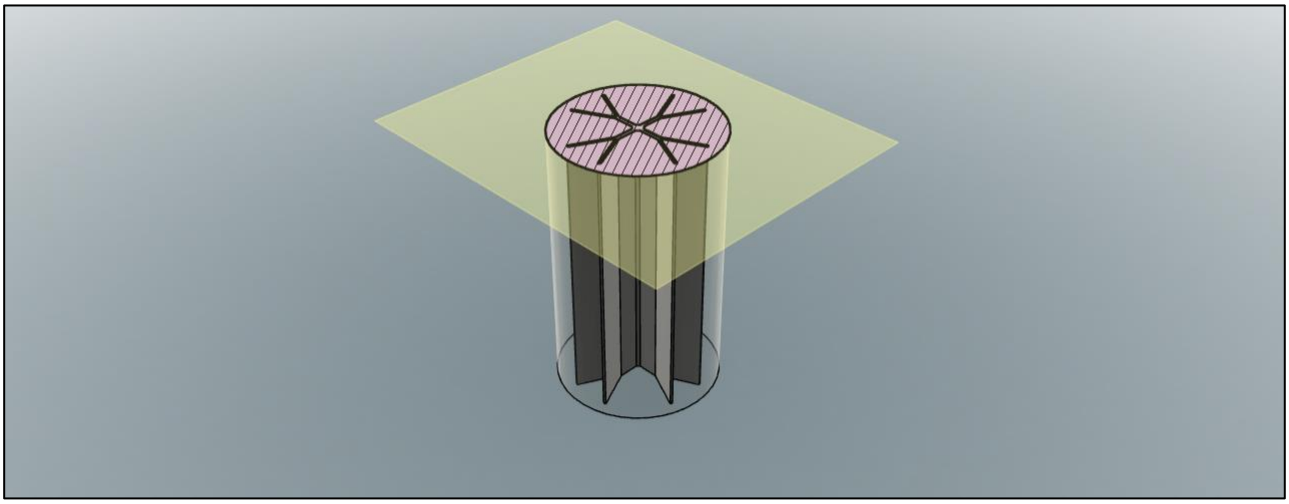
To reduce the computational cost of the numerical simulation we exploit the high symmetry of the domain that can be summarized in 1/8 of the total domain: the solution found for this portion can be repeated over the entire volume, considering the axisymmetric, to obtain the global solution.

Due to symmetry, all the other physical boundaries of the domain are considered adiabatic, it means that the isothermal lines will be perpendicular with respect to these surfaces and all the heat flux will be exchanged in the bound between water and PCM/alloy. But we do not know yet if the heat storage design will guarantee this possibility.

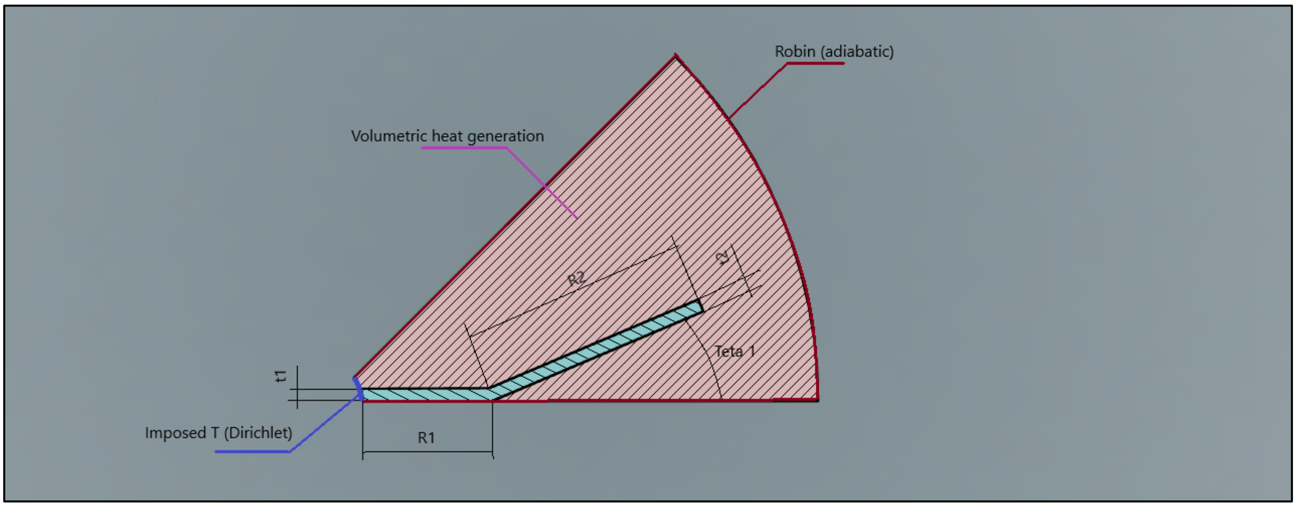
During the simulation the steady-state heat equation for heat conduction in solids will be solved:

$$-\mathrm{k}\cdot\nabla^2 T = q'''$$

Here below all these previous sentences are visualized:



(*PCM horizontal cross-section at half of the total heigth*)




(*PCM reduced domain with quotes and boundary conditions*)

As can be seen the geometry of the fins is described by five variables:

-length of the first fin $R_1 = 0.02\ m$ ;

-length of the second fin $R_2 = 0.04\ m$ ;

-thickness of the first fin $t_1 = 0.002\ m$ ;

-thickness of the second fin $t_2 = 0.002\ m$ ;

-tilt angle of the second fin $\theta_1$ ;

The optimization will be performed considering only $\boldsymbol{\theta_1}$ as design variable. The lengths and the thicknesses of the fins are kept constant.

The goal of the optimization is to maximize the heat transfer between PCM and water by optimizing the orientation of the second fin.

Since we are in an equivalent steady-state condition the heat flux will correspond to the heat source term and its expression will be a function of an equivalent global heat transfer $k_{global}$ and of the the average temperature of the PCM:

$$q' \left[\frac{W}{m}\right] = k_{global} \cdot (\overline{T_{PCM}} - T_{water})$$

To optimize the configuration, since the heat flux is constant, the equivalent global heat transfer should be maximized, but since the expression of this term and its dependence to geometry and to material variables is not known, the alternative is to **search for the value of $\boldsymbol{\theta_1}$ that minimize the average PCM temperature**.

If the minimum temperature will be near $60°C$ (the average melting temperature) it means that the design of the fins will guarantee the compatibility with the PCM taken in exam for the project, so we are able to supply the desired heat in the desired time.

If $k_{global}$ results to be too low even after optimization, the $\overline{T_{PCM}}$ must increase, so it will be necessary, in real application, to wait more time in order to exchange the desired heat, or to change the PCM material with another one with an higher melting temperature.

The optimization is performed using the GOLDEN SECTION METHOD, a direct optimization method suitable for convex objective functions with one decision variable only.

The first step is to recreate the sub channel geometry and properties and to find the bounds in which the tilt angle $\theta_1$ can vary. Considering the geometry of the problem, the minimum tilt angle allowed is 0° (fin completely in contact with the lower bound) while the maximum is 68°, where the second fin touches the upper bound of the domain (inclined line at 45°).

The method can start with the evaluation of the objective function at $\theta_1 = 68°$ and $\theta_1 = 0°$, which are the domain bounds as seen before.

Then the method consists in the evaluation of the objective function in two other points that are defined considering the golden section and the distance between the two bounds:

-*golden section,*

$$t = \frac{2}{1 + \sqrt{5}}$$

-*step,*

$$s = t \cdot (b - a)$$

*-other points,*

$$c = a + s \; ; \; d = b - s$$

Then the values of the objective function in these two additional points are compared, and it follows a reduction of the domain according the two values:

$if\ O.F(c) > O.F(d), then:$

$$a = d$$

$$b = b$$

$if\ O.F(c) < O.F(d), then:$

$$a = a$$

$$b = c$$

Once the domain where the optimum could be found is reduced, the process is repeated: with the golden section we compute the step ‘s’ of the second iteration (that will change since at least one of the two between point a and b is changed), and then new point c and d are defined.

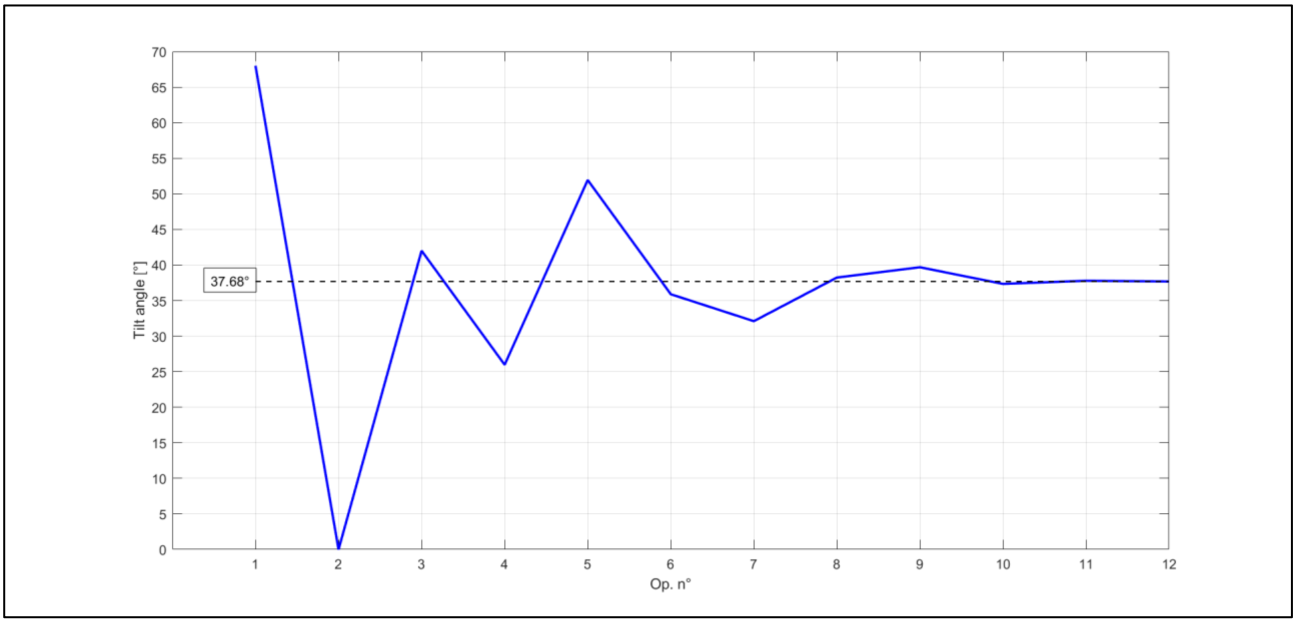
Thanks to the golden section, at every subsequent step the objective function is calculated only in one new additional point.

Here below the results of the optimization process are shown:

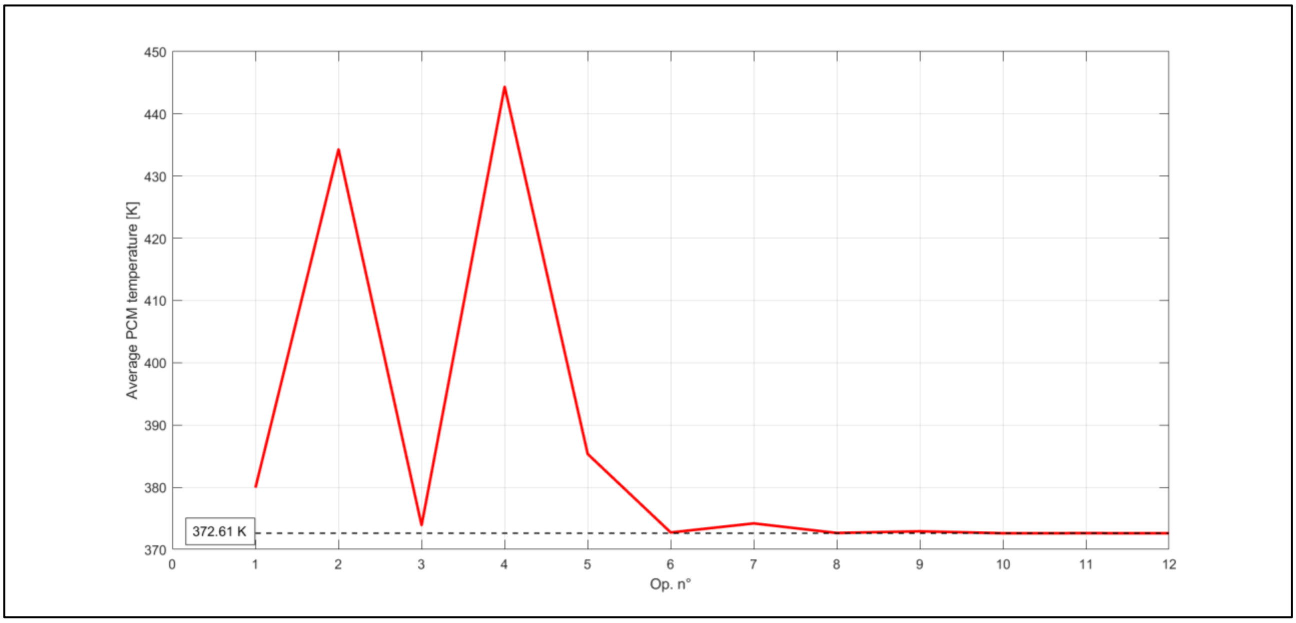
| **Iteration** | **Tilt angle $\boldsymbol{\theta_1}$** | **Average PCM temperature $\overline{T_{PCM}}$ [K]** |
|---|---|---|
| 1 | 0° | 434.29 |
| 1 | 25.97° | 444.36 |
| 1 | 42.03° | 373.89 |
| 1 | 68° | 379.93 |
| 2 | 51.95° | 385.34 |
| 3 | 35.89° | 372.73 |
| 4 | 32.10° | 374.19 |
| 5 | 38.23° | 372.65 |
| 6 | 39.68° | 372.92 |
| 7 | 37.34° | 372.61 |
| 8 | 36.79° | 372.62 |
| 9 | 37.68° | 372.61 |

The process took nine iteration to find the optimum value of $\theta_1$ with the desired precision.

In the following graphs it can be appreciated how the angle tends to the optimum value after every iteration and how, respectively, the average temperature decreases until stabilization around its minimum value:




(Tilt angle variation during the optimization process)




(Temperature evolution and its progressive stabilization during the optimization process)

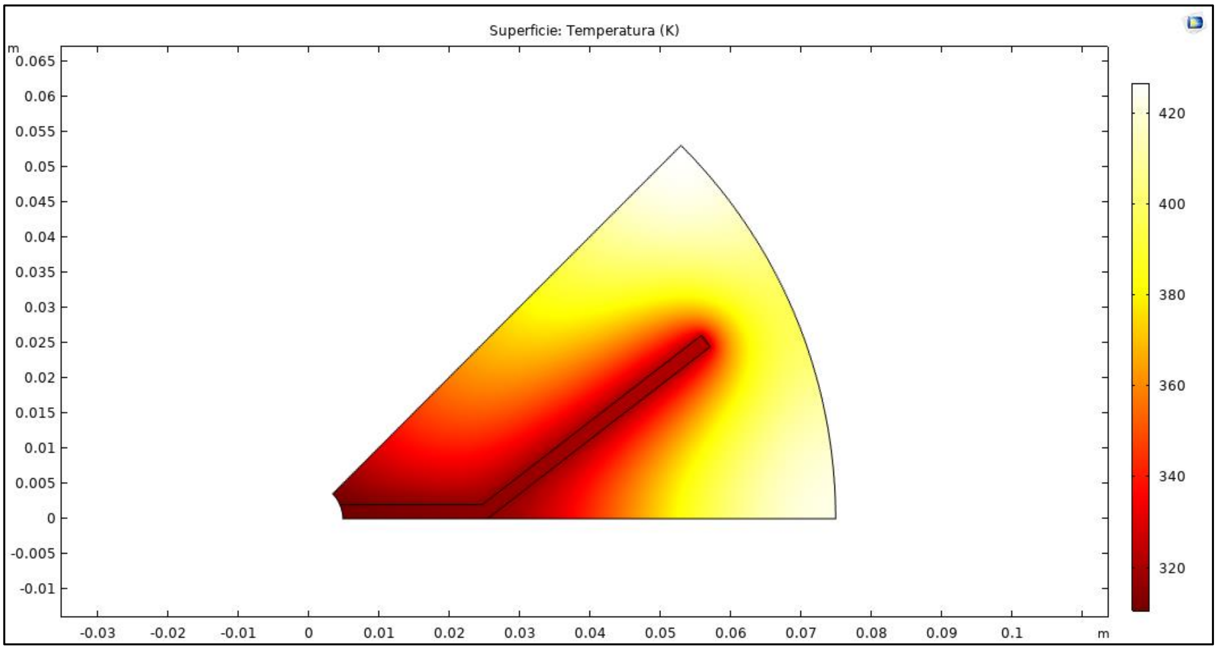
In conclusion we can say that, considering the finite precision during the manufacturing process, the optimum tilt angle for the second fin is in the range between two extremes:

$$\theta_{1,optimum} = 37 \div 38\ [°]$$

At which corresponds minimum value of the average temperature:

$$\overline{T_{PCM}}_{minimum} \approx 372.6\ [K] = 99.45\ [°C]$$

The numerical simulation gives back this steady-state temperature field of the heat storage unit for the optimum tilt angle:



(Equivalent steady-state temperature field for optimized tilt angle)

The minimum temperature though is not sufficiently low to justify the use of the selected PCM, since as seen before, the minimum temperature should have been around 60°C, that is the average melting temperature.

# 3- Transient optimization considering a 2D horizontal cross section

In this second part it is considered a transient discharge of $\Delta t = 3600\ s$ where the latent heat of solidification process is released by the PCM that starts from an initial temperature of 348.15 K (75°C).

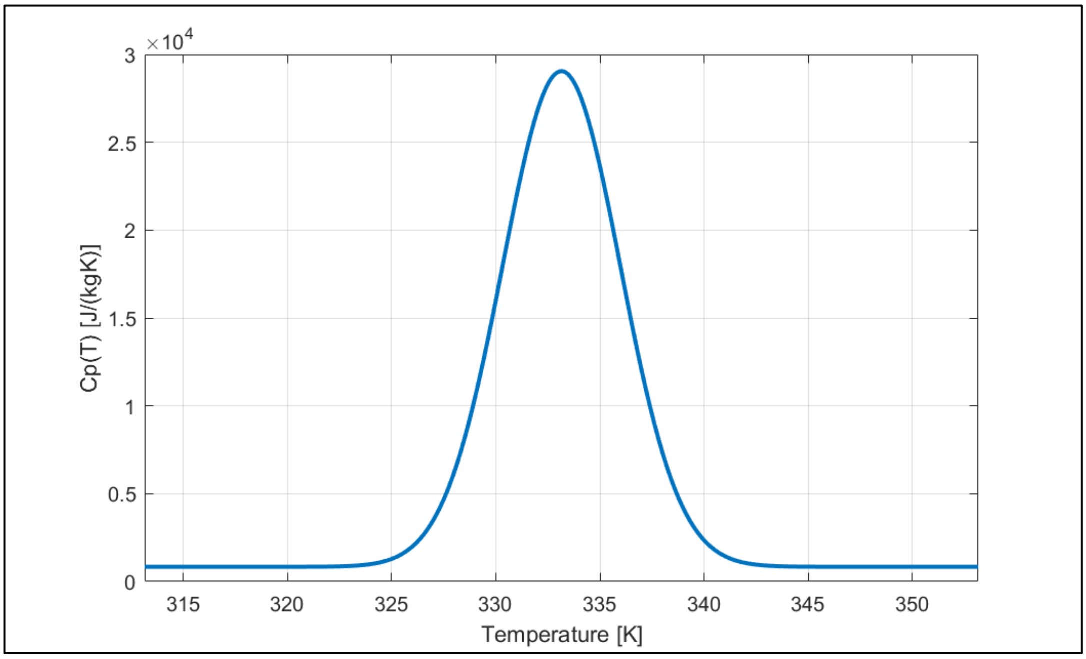
A proper setting of the PCM properties should be considered, we define the specific heat as a function of temperature, according to a Gaussian relation, considering half of the temperature range (4°C) in which the solidification process takes place:

$$c_p(T) = C_p(s,l) + \frac{L}{\Delta T_{sl} \cdot \sqrt{\pi}} \cdot e^{-\frac{(T-T_{av})^2}{\Delta T_{sl}^2}}$$

Where $C_p(s,l)$ is the specific heat of the solid or liquid PCM, $L$ is the latent heat, $\Delta T_{sl}$ is the melting temperature range and finally $T_{av}$ is the average temperature of the PCM during the phase change.

As said before the temperature range is 4°C while the average temperature is equal to 333.15 K (60°C).

By considering the phase change to start at T=337.15 K (64°C) and to finish when T=329.15 K (56°C) we can represent the specific heat in function of temperature:




(*Specific heat of PCM as a function of temperature in the melting range*)

The heat for unit of mass available from the PCM is made up by two components, a sensible heat (during the cooling phase in liquid state and in solid state) and a latent heat (during the phase change):

$$q = q_{sensible} + q_{latent} = \int_{64°C}^{T_{start}} c_p(liquid) \cdot dT + \int_{T_{end}}^{56°C} c_p(solid) \cdot dT + \int_{56°C}^{64°C} c_p(T) \cdot dT \quad \left[\frac{J}{kg}\right]$$

The idea is to maximize the thermal energy that is sent to the water during the transient. From the first law of thermodynamic for open system:

$$\Phi = \frac{\partial U}{\partial t} \quad [W] \quad ; \quad Q = \int_{T_{end}}^{T_{start}} \Phi \cdot dt \ = U_{start} - U_{end} \quad [J]$$

by doing an approximation we can write the internal energy as a function of an equivalent specific heat and of a delta temperature:

$$\Delta U = mc\Delta T = mc \cdot (T_{start} - T_{end})$$

From this two expressions is it clear that in order to maximize the thermal energy $Q$, the internal energy difference should be maximized and, since the initial temperature is fixed, **the final temperature of the PCM after 3600 s must be minimized.**

Once the objective function is defined, we perform the optimization that will be done considering two decision variables that are:

-the length of the first fin $R_1$

-the tilt angle of the second fin $\theta_1$

and in addition, we also consider an equality constrain in which the total volume of the fins must be kept constant, and since the thickness of the fins are not decision variables, the equality constrain becomes:

$$R_1 + R_2 = R_{tot} = const$$

Once all the information has been presented, the optimization process can start. For the numerical model we will consider a transient simulation where we only introduce the transient term in the heat equation for heat conduction in solids and remove the volumetric heat source:

$$\rho c \frac{\partial T}{\partial t} - \mathrm{k} \cdot \nabla^2 T = 0$$

For what concern the boundary conditions they are not modified, but an initial condition (as previously seen) must be defined:

$$T_{PCM}(t = 0s) = 348.15\ K\ (75\ °C)$$

During the optimization we will only perform modifications in the geometry of the fins, without touching other aspects of the simulation process.

The optimization is performed using the CONJUGATE DIRECTIONS METHOD, where for convex objective functions the minimum is investigated through subsequent search in perpendicular directions.

It means that for each step the optimization with respect to one single decision variable is performed, while the second one is kept fixed. Once the minimum is reached, the first decision variable is fixed and the optimization is done considering the second one.

At each step we use the QUADRATIC APPROXIMATION METHOD to perform the optimization in a single variable.

The quadratic approximation is an indirect optimization method where we search for the minimum by studying the first derivative of the objective function. Being an indirect method, the minimum could be located outside the domain defined for each variable, so at each iteration we must evaluate and check the value of the objective function at the bounds.

The optimization starts with a fixed value of the first fin's length:

$$R_1 = 0.021\ m$$

which is the value used in the first part of the practice (we could start with a fixed value of the tilt angle equal to the one found in the first section, but it won't make too much difference). At each iteration we have to pay attention since the bounds of the second decision variable change according the feasibility of the geometry itself: as seen in the first part for a length of the first fin equal to 0.002 m the maximum tilt angle is 68° and the minimum is 0°. If in the next iterations we always consider 68° and 0° as bounds of the domain we may lose a lot of possible configurations in which the minimum could be located. With all of that being said we need some additional points (for a total of at least 3 points) where to evaluate the objective function in order to approximate it with a polynomial of the second order:

$$O.F\ approx.to\ f = a\theta_1^2 + b\theta_1 + c$$

by imposing the first derivative equal to zero we find the optimum tilt angle for a defined length of the first fin:

$$\theta_{1,opt} = -\frac{b}{2a} = 34.68°$$

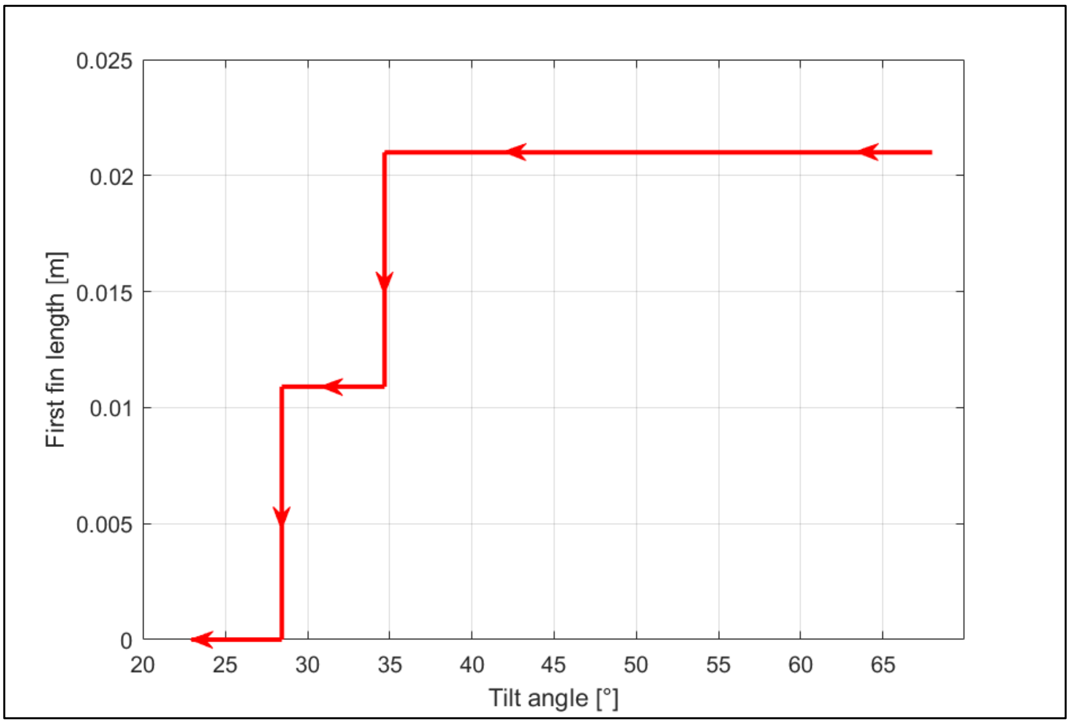
In the subsequent iteration the value of the tilt angle is fixed, and for that value, the objective function is evaluated in at least 3 points corresponding to different R1 values (always considering the bounds according to physical constrains in the manufacturing process and/or in the feasibility of the geometry itself).

In the table below, the six iterations that lead to the optimum are summarized:

| **Iteration** | $\boldsymbol{\theta_1}$ [°] | $\boldsymbol{R_1[m]}$ | $\boldsymbol{R_2[m]}$ | $\boldsymbol{\theta_{1,opt}}$[°] | $\boldsymbol{R_{1,opt}[m]}$ | $\boldsymbol{\overline{T_{PCM}}\ (t = 3600s)[K]}$ |
|---|---|---|---|---|---|---|
| 1 | 0<br>68<br>20<br>55 | 0.021 | 0.04 | 34.68 | // | 329.75 |
| 2 | 34.68 | 0.001<br>0.06<br>0.04<br>0.015 | 0.06<br>0.001<br>0.021<br>0.046 | // | 0.0109 | 328.65 |
| 3 | 5<br>53<br>12<br>40 | 0.0109 | 0.0501 | 28.43 | // | 328.78 |
| 4 | 28.43 | 0.055<br>0<br>0.002<br>0.035 | 0.006<br>0.061<br>0.059<br>0.026 | // | 0 | 328.17 |
| 5 | 44<br>2<br>14<br>31 | 0 | 0.061 | 22.93 | // | 328.11 |
| 6 | 22.93 | 0.055<br>0<br>0.014<br>0.041 | 0.006<br>0.061<br>0.047<br>0.02 | // | 0 | 328.11 |

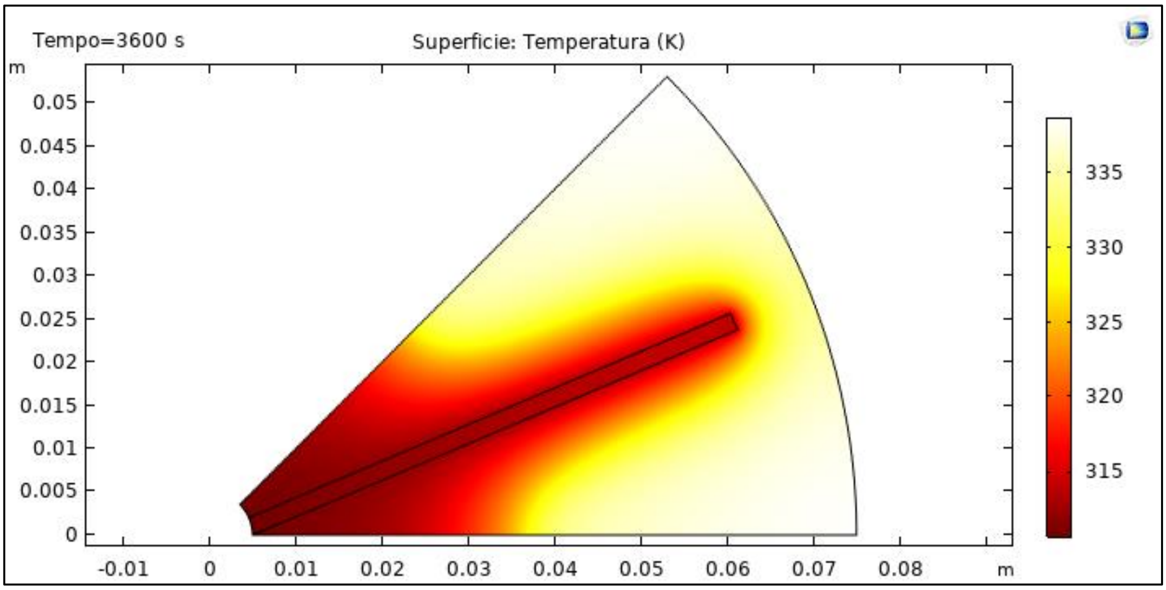
At the end of the process, we can see that the minimum final temperature of $328.11\ [K]$ is reached when the first fin doesn't exist, so for a length $R_1 = 0\ [m]$, and for a tilt angle of the second fin equal to $\theta_1 = 22.93°$.

The optimization process can be summarized in the following image, where it's possible to appreciate how the conjugate directions method works, by moving in perpendicular directions:



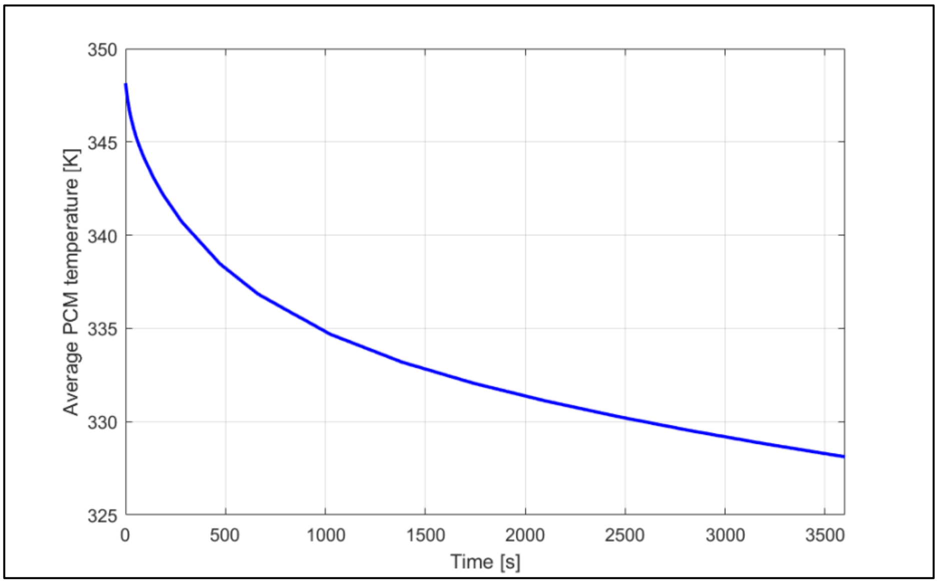

(*Conjugate directions path*)

Here it is also reported the optimized geometry configuration with the final temperature field of the sub-channel region:




(*Temperature field at t=3600 s for the optimized geometry*)

Once the optimum design has been found, the following step is to compute the average temperature evolution of the PCM during the transient. The function $\overline{T_{PCM}}(t)$ for the optimized design is shown below:



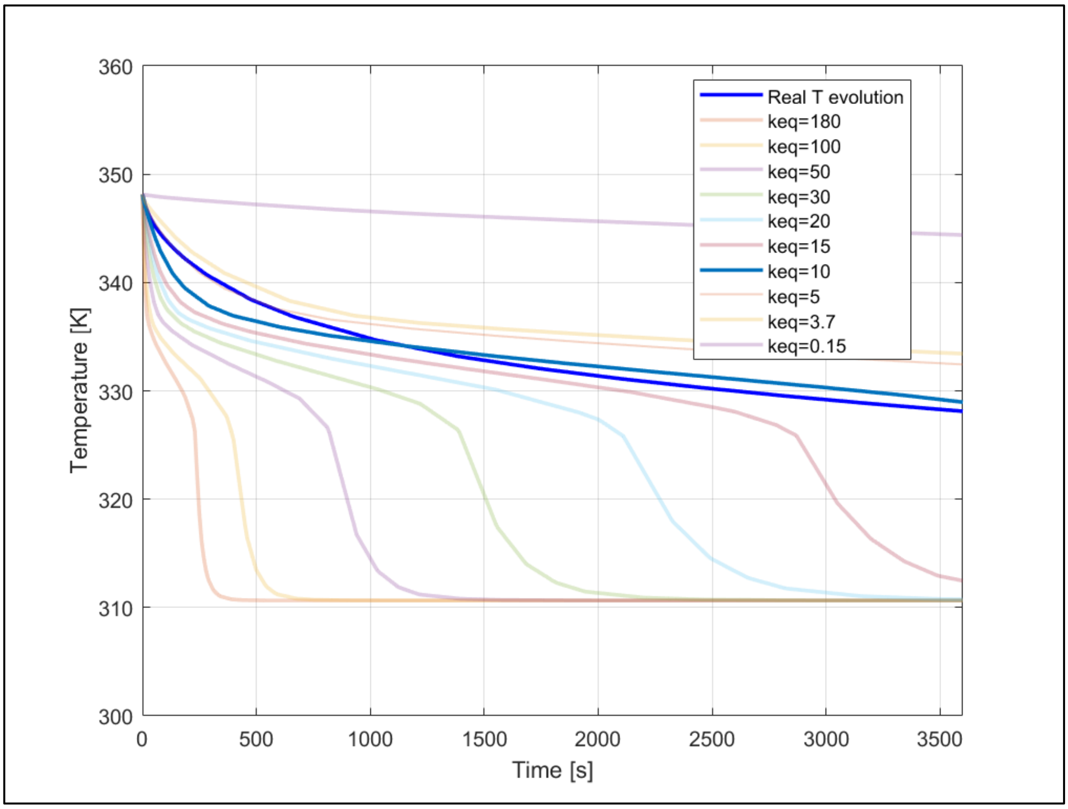
(*Transient evolution of the average PCM temperature during the discharging phase with optimized geometry*)

Starting from that, the goal is to find an **equivalent modified conductivity** $\boldsymbol{k_{eq}}$ of a fictitious homogeneous material that, without any fins but with the same expression of the PCM specific heat, will give an almost equal temperature evolution of the one found for the thermal storage unit equipped with aluminum alloy fins.

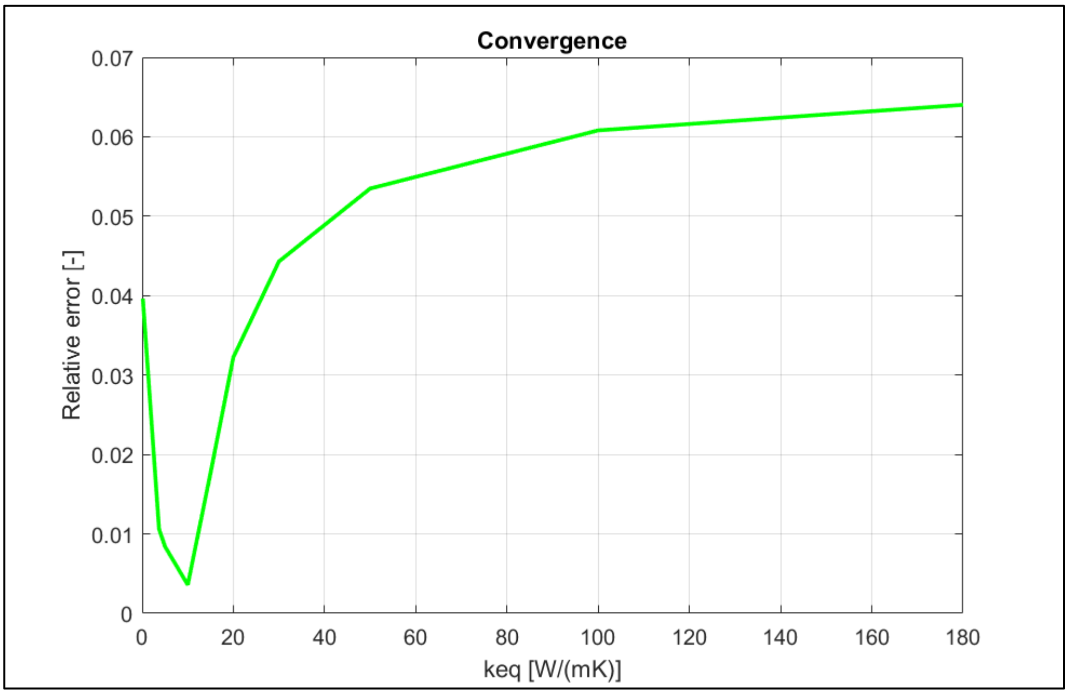
In order to do that we perform a trial and error by removing the fins in the geometry and by modifying the conductivity of the PCM until the temperature evolution converge to the one desired.

The different trials results and the evolution of the relative error computed considering the norm of the two vectors ( $\overline{T_{PCM}}, k_{real}(t)$ , $\overline{T_{PCM}}, k_{eq}(t)$ ) are presented in the table and in the graphs below:

| $\boldsymbol{k_{eq}\left[\frac{W}{mK}\right]}$ | $\boldsymbol{\overline{T_{PCM}}, k_{eq}(t = 3600s)}$ **[K]** | $\boldsymbol{rel.err}$**[-]** |
|---|---|---|
| 180 | 310.65 | 0.0640 |
| 100 | 310.65 | 0.0608 |
| 50 | 310.65 | 0.0535 |
| 30 | 310.95 | 0.0443 |
| 20 | 310.76 | 0.0322 |
| 15 | 312.47 | 0.0177 |
| 10 | 328.96 | 0.0036 |
| 5 | 332.44 | 0.0084 |
| 3.7 | 333.43 | 0.0106 |
| 0.15 | 344.39 | 0.0396 |




(*Temperature evolution according different equivalent conductivity*)




(*Relative error evolution*)

The equivalent conductivity of the fictitious material results to be something around $k_{eq} \approx 10 \left[\frac{W}{mK}\right]$; we do not properly talk about convergence since, as can be appreciated in the first graph, we are mostly interested in an equivalent conductivity that gives almost the same final temperature, while the overall temperature evolution is not perfectly reproduced.

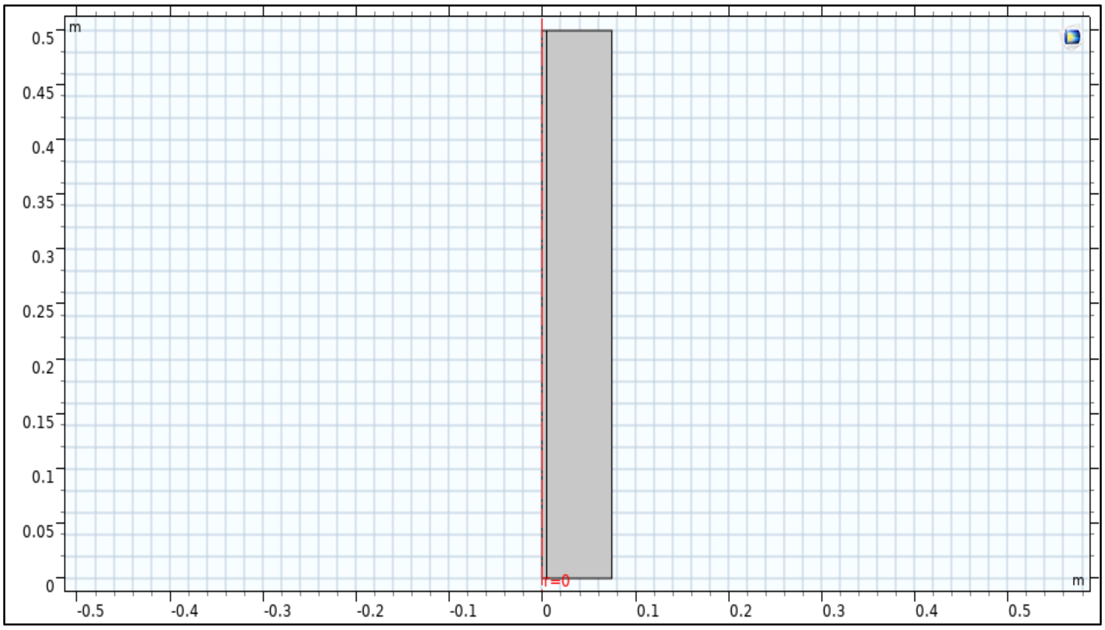
# 4- Simulation considering a 2D vertical cross section

The purpose of this section is to compute, starting from the last result found in the previous section, the temperature evolution of water at the outlet section of the heat storage unit during the discharging phase. This time a 2D vertical cross section of the unit will be considered and the complete solution of the problem will be obtained by taking advance of the axial symmetry of the system.

Is it clear that, since the simulation will consider the water flow, a multi-physics simulation is needed and a conjugate heat transfer model will be implemented;

- In the first section the velocity and pressure fields of water flowing inside the inner pipe is solved considering, as an initial hypothesis, a laminar flow condition in a steady-state configuration.
- In the second part, the results obtained in the previous one are used to simulate a transient heat transfer process. The result of this section will be the temperature field evolution in space and time during the discharging phase ($\Delta t = 3600\ s$) in the entire domain. But as said previously we will be specifically interested in the average water outlet temperature evolution.

The first step of the simulation requires to build the geometry of the system. As said previously, considering the axisymmetric of the thermal storage unit, the domain is reduced to a 2-dimensional vertical cross section, which is shown below:




*(Vertical cross section of the sub-channel)*

Concerning the dimensions, the radius of the inner pipe (where water flows) and the external radius of the sub-channel are the same presented before, while the height of the cross section is equal to the height of the thermal storage unit itself ( $h = 0.5\ [m]$).

For what regards the materials instead, the inner pipe is modeled with water thermo-fluid dynamic properties seen above, while for the PCM we won't model the PCM with aluminum alloy fins, but we will use the ensembled PCM+ fins with the equivalent fictitious thermal conductivity of $k_{eq} \approx 10 \left[\frac{W}{mK}\right]$ found at the end of the previous transient optimization considering a 2D horizontal cross section.

Density and specific heat are instead maintained equal to the ones of the PCM.

**Fluid dynamics simulation**

Once the geometry and the materials properties are defined in the model, a further step consists in the definition of boundary conditions.

We won't have initial condition since the fluid velocity and pressure fields are solved in steady-state and the transformation experienced by the fluid is isothermal.

In addition to the information which are already included in the model (as the symmetry of the system and the no-slip condition where the axial component of the velocity is $u = 0\ \left[\frac{m}{s}\right]$ at the transition in the domain from water to the ensembled PCM) at the inlet section we specify the velocity of the fluid, assuming a uniform unidimensional velocity field:

$$u = 0.1\ \left[\frac{m}{s}\right]$$

Knowing the fluid-dynamic properties of water and the internal diameter of the pipe we can compute the Reynolds' number to check if, with the selected velocity, we're actually in the laminar flow condition:

$Reynolds\ number\ condition\ for\ laminar\ fully\ developed\ flow\ in\ circular\ pipes$:

$$Re \leq 2100$$

$$Re = \frac{\rho \cdot u \cdot D}{\mu} = \frac{990 \cdot 0.1 \cdot 0.01}{0.0008} = 1237.5 \leq 2100$$

Considering the outlet section instead, we specify the relative pressure:

$$P_{rel,out} = 0\ [bar]$$

The next step consists in the creation of a proper mesh to solve the FEM simulation. The needs of a non-uniform mesh over the entire domain is due to the physics of the system: we enter the pipe with a uniform velocity field and, as soon as the water gets in contact with the pipe, viscous forces start to take a role and, layer by layer, we assist to a modification of the velocity field along the axial direction. It means that a transition region will appear in the steady-state solution, which is the region of the pipe in which the transition from an uniform flow into a parabolic completely developed laminar flow takes place.

Since the laminar flow is a relatively simple flow condition, the solution can be obtained in an **analytical** way directly from Navier-Stokes equation.

From the *Hagen-Poiseuille* solution we can obtain the actual parabolic distribution of velocity, which could be used to a convergence study of the FEM model:

$$u = u_{max}\left[1-\left(\frac{r}{R}\right)^2\right]\left[\frac{m}{s}\right]$$

And for a laminar flow we can also compute the length of the developing region considering a correlation with the Reynolds number and the pipe diameter:

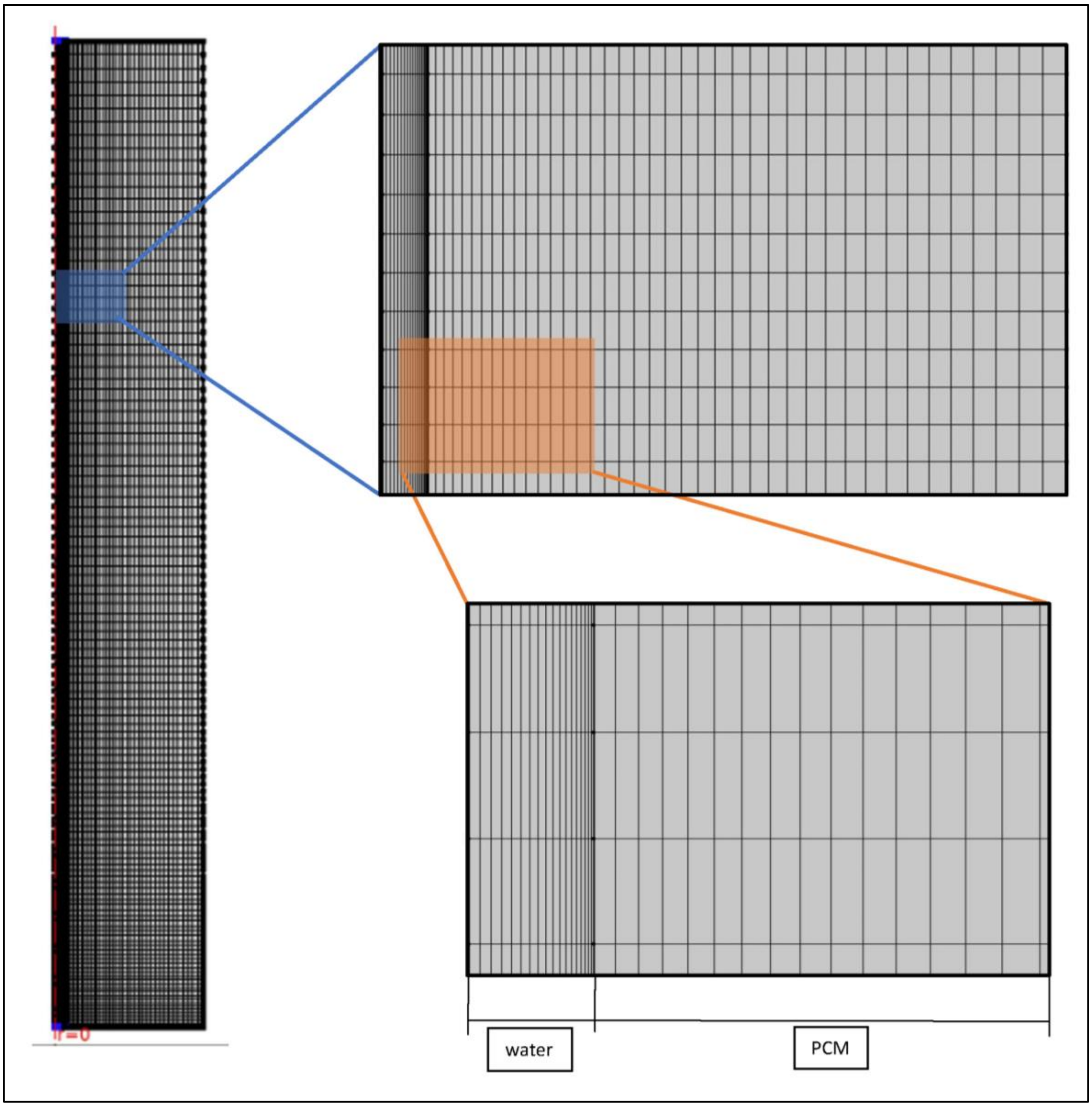
$$Z_{FD} = D \cdot 0.06 \cdot Re \approx 0.74\ [m]$$

By doing that we discover that the flow won't be fully developed inside our pipe, because the length of the pipe itself is lower than the length of the entry region. In particular, in the first part of the pipe we will have the more drastic changes of the velocity field, so in order to properly describe this evolution a high "dense" mesh is needed in the first centimeters of the pipe.

Always considering the physics of the system we must adjust the mesh also in the radial direction, since near the bounds, considering the parabolic propagation, we will have higher gradient.

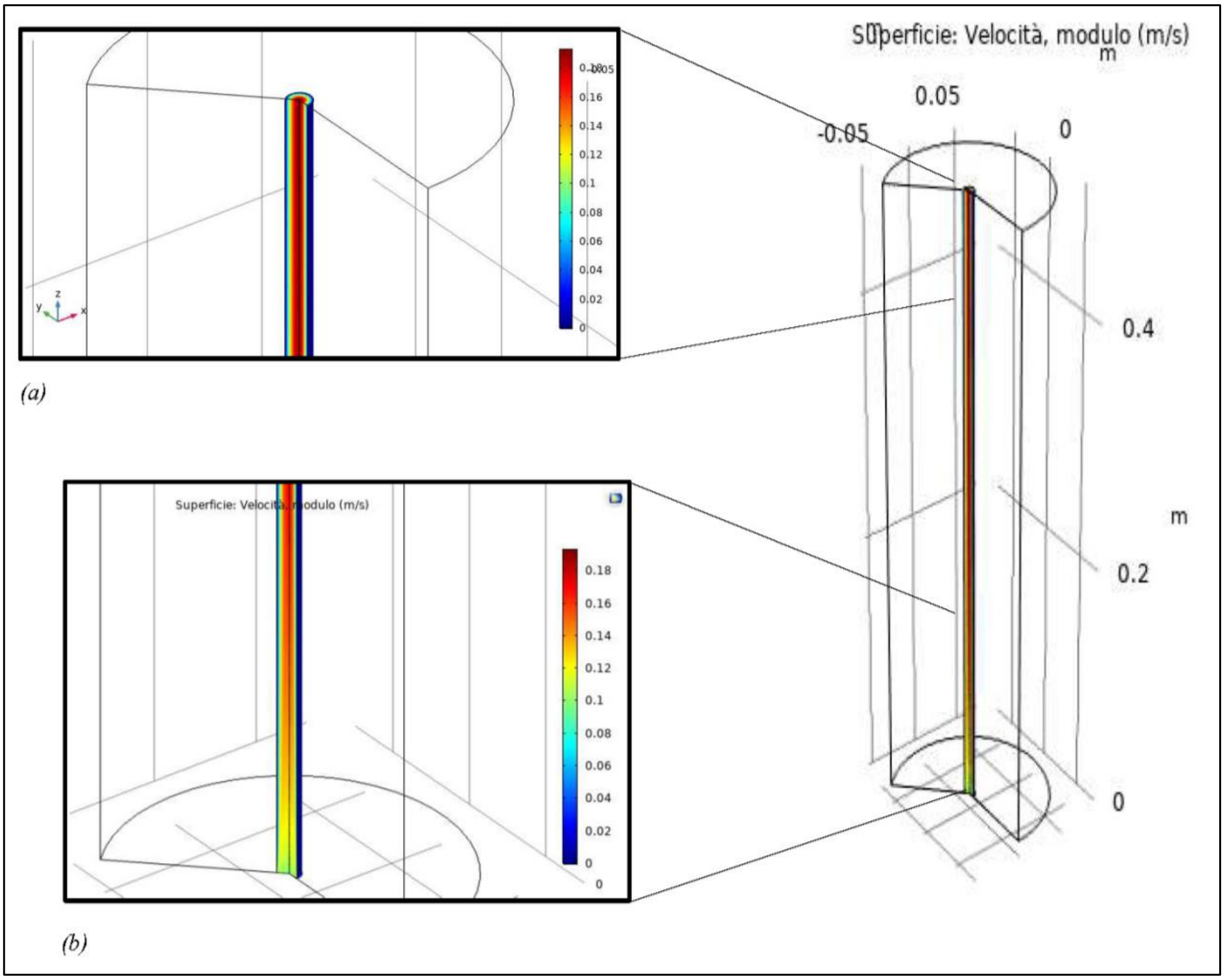
Higher gradient requires deeper discretization of the domain for a better description, so again our mesh will be characterized by non-uniform elements, which will be smaller near the bounds of the inner pipe.

Here, an example of what has been done is presented to better fix the concept:




*(details of the mapped mesh)*

Once the “mapped” mesh has been built, we can perform the simulation, which results are presented below:



*(Fluidynamic solution with details on: (a)outlet region, (b)inlet region)*

By looking at the pressure gradient we're able to see that is not constant $\frac{\partial P}{\partial z} \neq const$, sign that the laminar flow is not fully developed.

**Transient heat transfer simulation**

Once the steady state fluid-dynamic results are available, we can use them as inlet information for the subsequent step.

The goal of the thermal problem is to solve the temperature field evolution in the thermal storage unit during the transient discharging phase.

This time the heat equation will have an additional term, the "advective" one, which considers the heat removed by the water stream:

$$\rho c \frac{\partial T}{\partial t} - \mathrm{k} \cdot \nabla^2 T + \rho c \cdot u \cdot \nabla T = 0$$

The geometry and material properties are not affected; what we must do this time is to change or modify the boundary conditions and to properly set initial conditions. Starting from the initial condition itself, we impose a uniform temperature $T(t=0s)=75\ [°C]$ over the entire domain.

For what concerns the boundary conditions, for the PCM we impose the thermal insulation for each bound, except for the one in contact with water.

At the inlet section of the water pipe, we impose a fixed inlet temperature of 35°C for water.

For the outlet section instead, we select a Neumann's boundary condition stating no back propagation of heat by conduction:

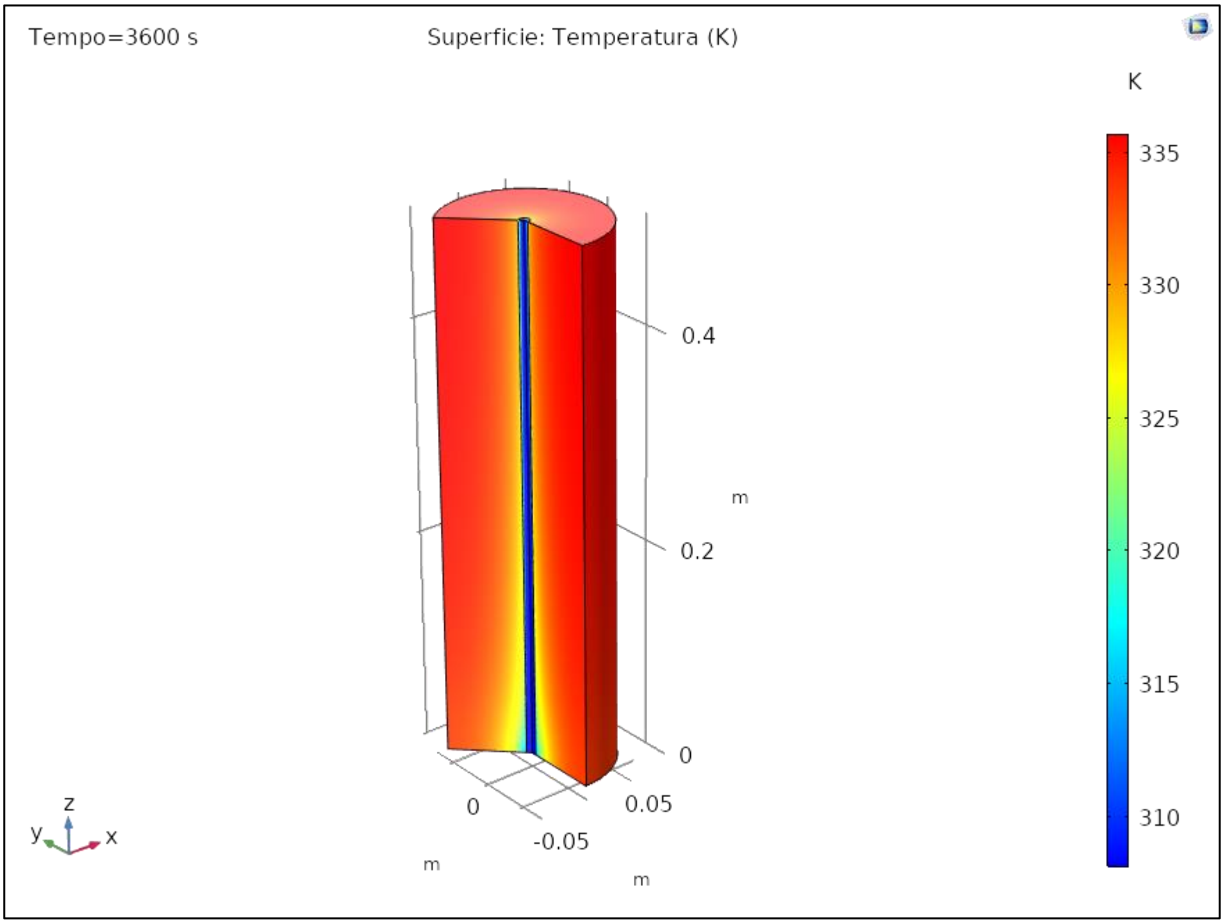
$$-k \cdot \overrightarrow{\nabla T} \cdot \vec{n} = 0$$

Is interesting to see that the no-back propagation condition has the same formulation of an adiabatic wall, but this time the meaning is completely difference because the outlet section will not be adiabatic at all since convection is possible in the direction of the flow.

The next step requires the construction of the mesh, again we apply the same concept adopted for the first simulation: the idea is to have a high dense mesh with smaller elements where the gradient of temperature is higher. That will guarantee a more accurate description of the phenomenon.

In particular, we will consider a radial and an axial variation of the elements also for the PCM, in particular with high dense mesh at the material transition surface and at axial region which corresponds to the not fully developed flow.

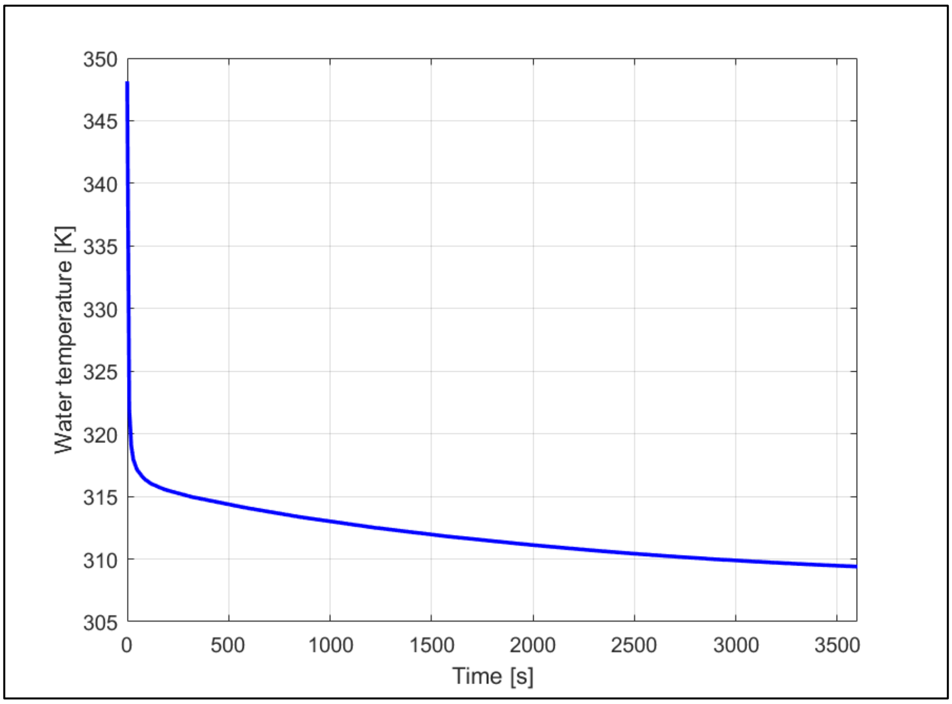
Having built the mapped mesh we can proceed with the second simulation which results are reported in the image below:



*(Temperature distribution at t=3600s)*

As said before, we are interested specifically in the evolution of the average water temperature at the outlet on the inner pipe of the thermal storage unit sub-channel.

The temperature evolution in the discharging period of $\Delta t = 3600\ s$ is shown below:



# 5- Design improvement considering a 2D horizontal cross section

In the last part of the report the attention goes to the design optimization of the same geometry proposed in “Steady-state optimization considering a 2D horizontal cross section” part.

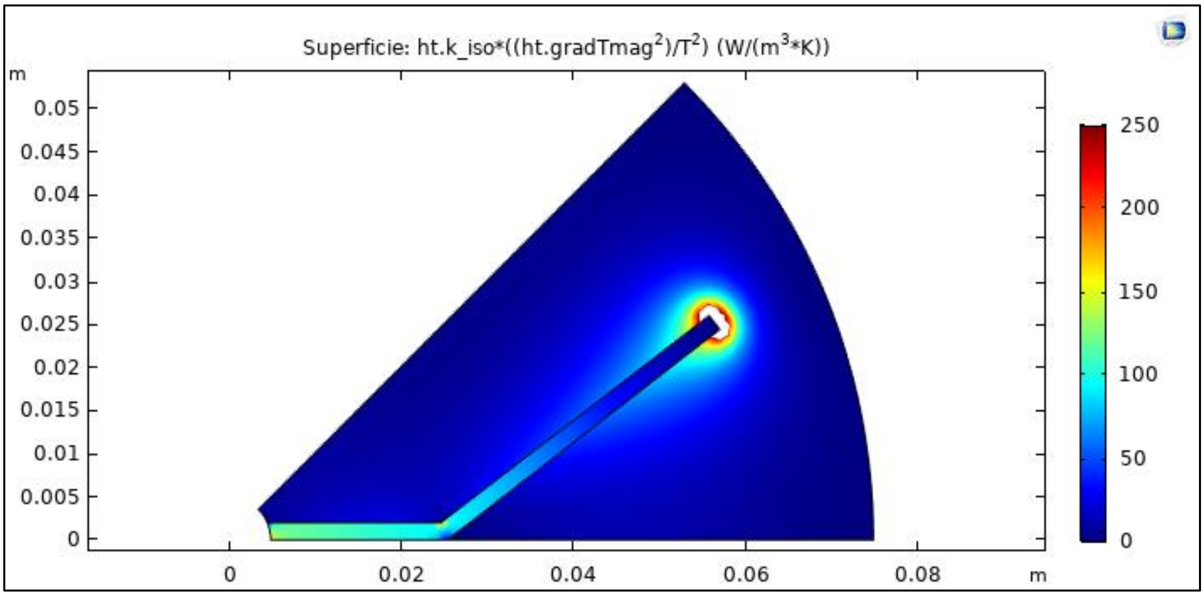
This time though, the optimization is performed according to the analysis of the entropy generation map inside the domain, with the goal of making it as homogeneous as possible, without necessary search for a smaller integral value, but while maintaining almost the same average equilibrium temperature of 372 K.

The process starts with the simulation of the one variable optimized design, the one found as a result in the first part, where for a length of the first fin equal to $R_1 = 0.02\ m$ the optimized tilt angle of the second fin is $\theta_{1,optimum} = 37$ [°] .

With that specific geometry the entropy generation map can be easily defined thanks to the continuum level definition of the entropy generation term related to the heat transfer process (which is the only one occurring at this level of characterization of the phenomenon):

$$\Sigma_{irr} = k\left(\frac{\nabla T^2}{T^2}\right)$$

Here it is presented the entropy generation map for the one variable optimized geometry:




*(Entropy generation map for the optimized geometry found in task 1)*

As can be seen in the first fin, where the heat exchange flux per unit surface increases, the temperature decreases and the thermal conductivity is high, the entropy generation is higher.

We are neglecting the hot-spots located at the top of the second fin since they're mostly a "numerical scar" due to the presence of sharp edges and high temperature gradient.

The purpose now is to modify the geometry of the fins, while keeping the total volume constant as an equality constrain, to increase the homogeneity of the entropy generation map.

In order to do that we should think about the three elements that make up the $\Sigma_{irr}$ term in order to find a geometry that balances the thermal conductivity and the square of the ratio between the thermal gradient and the temperature itself: when the temperature is high, also the thermal gradient should be high, if the ratio $\left(\frac{\nabla T^2}{T^2}\right)$ is high, we should counteract the entropy generation by putting material with low thermal conductivity.

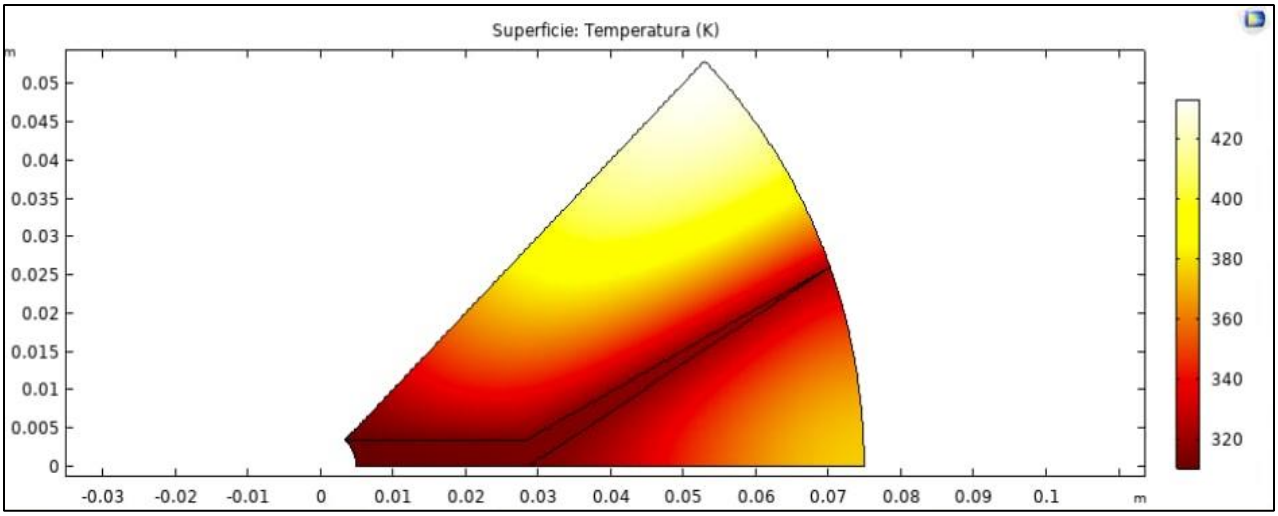
As a first move, we increase the thickness of the first fin to have a bigger region characterized by lower temperatures and high thermal conductivity.

The amount of material which is transferred to the first fin is taken from the second fin, which will result to be shorter. That is a problem because a shorter second fin will not be able to properly cover the PCM, and the average temperature will be higher.

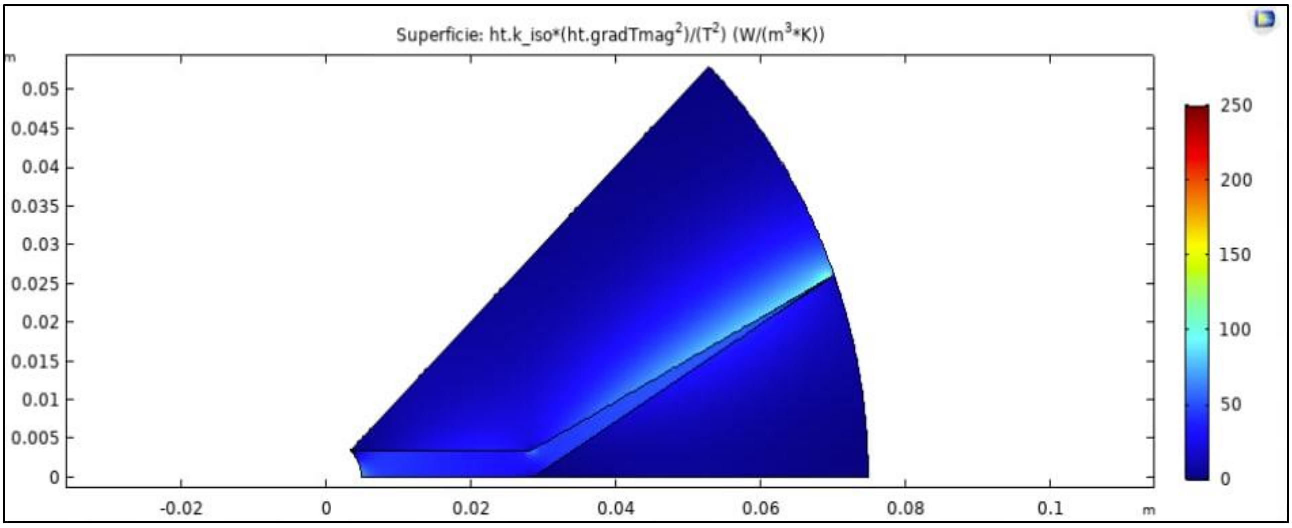
To overcome that, we use a different geometry for the second fin: a triangular shape.

By using a triangle instead of a rectangle, saving half of the material and properly distributed increasing the length of the second fin, finding, after the simulation, an average temperature $\overline{T_{PCM}}_{minimum} = 364K = 90.85°C$, which results to be even better than the previous one.

Here below the results of the optimized geometry are presented:



(Equivalent stedy-state temperature distribution)




(New entropy generation map with the optimized geometry)