UNIVERSIDAD SAN FRANCISCO DE QUITO USFQ

Colegio de Ciencias e Ingeniería

Diseño del sistema de refrigeración para la caja lectora del detector de partículas tipo hadrón

.
Proyecto de Investigación

David Alejandro Jaramillo Bazurto

Ingeniería Mecánica

Trabajo de titulación presentado como requisito
para la obtención del título de
Ingeniero Mecánico

Quito, 10 de mayo 2019

UNIVERSIDAD SAN FRANCISCO DE QUITO USFQ

Colegio de Ciencias e Ingenierías

HOJA DE CALIFICACIÓN

DE TRABAJO DE TITULACIÓN

Diseño del sistema de refrigeración para la caja lectora del detector de partículas tipo hadrón

David Alejandro Jaramillo Bazurto

Calificación:

Nombre del profesor, Título académico: David Escudero Ph.D.

Firma del profesor: ______________________________

Quito, 10 de mayo 2019

Derechos de Autor

Firma del estudiante: ____

Nombres y apellidos: David Alejandro Jaramillo Bazurto

Código: 00117002

Cédula de Identidad: 1716358492

Lugar y fecha: Quito, 10 de mayo de 2019

DEDICATORIA

Me gustaría agradecer en estas líneas la ayuda que muchas personas y colegas me han prestado durante el proceso de investigación y redacción de este trabajo. En primer lugar, quisiera agradecer a mi familia, especialmente a mis padres que me han ayudado y apoyado en todo mi producto, a mi tutor, David Escudero, por haberme orientado en todos los momentos que necesité sus consejos.

Así mismo, deseo expresar mi reconocimiento a la Universidad y a sus profesores por todas las atenciones e información brindada a lo largo de esta indagación.

Finalmente, a mis amigos. Con todos los que compartí dentro y fuera de las aulas. Aquellos amigos del cole, que se convierten en amigos de vida y aquellos que formaron parte de la carrera universitaria y quienes serán mis colegas, gracias por todo su apoyo y diversión.

ABSTRACT

This project presents a development upgrade for the cooling system of the Readout Module Box (RBx) of the Hadron Calorimeter Detector (HCAL) from the LHC collider. The Compact Muon Solenoid (CMS) experiment will have a better performance if the HCAL electronics are working at the right conditions. One of the suitable options for doing this is by removing the heat from the front end electronics (FEE) located in the RBx compartment using water as a cooling fluid through cooper lines, however the current design is not cooling the electronics effectively, and thus a new design is proposed to cool the system. First, an analytical analysis with the fundamentals of heat transfer mechanism, thermodynamics and fluid mechanics is performed to determine the heat transfer area and then the length of the cooper tube. Then a fluid mechanics analysis to overview the pressure losses in the tube and momentum conservation effect by bends. Also, computational simulations were carried out using COMSOL Multiphysics software to observe the thermal behavior of the RBX using three types of arrangements of cooper lines on the floor of the RBx. The new distribution designs of the copper lines of the refrigeration system reduce the temperature of the RBx in approximately 10°C, improving the performance of the electronics.

Key words: RBx, cooling system, heat dissipated

RESUMEN

El presente proyecto es un desarrollo de mejora para el sistema de refrigeración de la caja lectora del detector de partículas tipo hadrón del largo colisionado de partículas. El experimento del solenoide compacto de muones tendrá un mejor rendimiento si el detector de partículas tipo hadrón trabaja en las condiciones correctas. Una de las opciones adecuadas es remover el calor de la caja electrónica frontal ubicada en el compartimento de la caja lectora, mediante el uso de agua como fluido de refrigeración por medio de tuberías de cobre. Sin embargo, el diseño actúa no está enfriando la electrónica eficientemente, por lo que un nuevo diseño es propuesto para enfriar el sistema. En primera instancia, un análisis analítico con los fundamentos de transferencia de calor, termodinámica y mecánica de fluidos es realizado para determinar el área de transferencia de calor y así la longitud de la tubería de cobre. Así también mediante el análisis de mecánica de fluidos se determina las pérdidas de presión en la tubería y la conservación del momento por efecto de los dobleces. Además, simulaciones computacionales fueron llevadas a cabo usando del software COMSOL Multiphysics para observar el comportamiento térmico de la caja lectora usando tres tipos de arreglos de tubería en el piso de la caja lectora. Como resultado, por el método de criterio de ponderados se seleccionó los diseños 1 & 2 para la fase preliminar de este proyecto. La nueva distribución de los diseños de las tuberías de cobre del sistema de refrigeración reduce la temperatura de la caja lectora en aproximadamente 10°C, mejorando el rendimiento de la electrónica.

Palabras claves: Caja lectora, sistema de refrigeración, calor disipado

INDEX

Dedicatoria 3
1. Introduction 13
2. Methodology 22
2.1 Analytical Design 22
2.1.1 Fundamentals of heat transfer. 22
2.1.2 Fundamentals of fluid mechanics. 27
2.1.3 Conjugate Heat Transfer: Internal Flow. 29
2.1.4 Fundamentals applied to the system. 32
2.2 CFD Analysis 38
2.2.1 Comsol Multiphysics Overview. 38
2.2.2 System design and analysis 45
3. RESULTS AND DISCUSSION 60
3.1 Mathematical Results 60
3.1.1 Temperature distribution. 60
3.1.2 Pipe friction losses. 62
3.2 Tube layout results 63
3.3 Simulation Results. 65
3.3.1 Qualitative results. 65
3.3.2 Quantitative Results. 72
4. CONCLUSION AND RECOMMENDATIONS 76
5. FUTURE WORK 77
6. REFERENCES 79
7. APPENDIX 81
7.1 APPENDIX A 81
7.2 Appendix B 83
7.3 Appendix C 84
Script C.1. Octave script for design proposal 1 84
Script C.2. Octave script for design proposal 2 85
Script C.3. Octave script for design proposal 3 86
7.4 Appendix D 87

Acronyms

LHC	Large Hadron Collider
CMS	Compact Muon Solenoid
HCAL	Hadron Calorimeter
HB	Hadron Barrel
HF	Hadron Forward
HE	Hadron Endcap
HO	Hadron Outward
RBx	Readout box
FEE	Front-End Electronics
RM	Readout Module
QIE	Charge Integrator Encoder card
ODU	Optical Decoder Unit
HPD	Hybrid Photodiode
SiPM	Silicon Photomultiplier
LS2	Long shutdown 2
OD	Outside diameter
ID	Inside diameter
TDR	Technical Design Report
BC	Boundary Condition

Nomenclature

q	heat dissipated
q_x	heat dissipated in the x direction
T	Temperature
T_m	mean temperature
T_{in}	temperature at the inlet
T_{out}	temperature at the outlet
Dt	differential difference of temperature
UA	Overall heat transfer coefficient
T_1	Temperature at point 1
T_2	Temperature at point 2
q''_x	Heat flux
ΔT_{cv}	Delta convection temperature
ΔT_{UA}	Delta overall heat transfer coefficient
T_s	Surface temperature
T_∞	Fluid temperature
q''_x	Heat flux
E_{in}	Inside energy
E_{out}	Outside energy
E_{gen}	Energy generated
E_{st}	Storage energy
ΔT_{IE}	Temperature difference internal energy
T_{QIE}	QIE card temperature
T_M	Module temperature
T_{RBX}	RBx outside wall temperature
$\dot{m}$	mass flow rate
h_L	Pressure losses
h_f	Major losses
h_m	Minor losses
u	Velocity of fluid
F	Friction factor
V	Fluid velocity
v_1	Velocity specified at point 1
v_2	Velocity specified at point 2
P_1	Pressure specified at point 1
P_2	Pressure specified at point 2
Z_1	Height specified at point 1
Z_2	Height specified at point 2
c_p	specific heat
ρ	density of fluid
μ	kinematic viscosity
K	thermal conductivity
ε	Rugosity material
H	Convection heat transfer coefficient
$\underline{h}$	Mean heat transfer coefficient

Nu	Nusselt number
$\underline{Nu}$	Mean Nusselt number
Pr	Prandlt number
Re	Reynolds number
Re_D	Reynolds number based on the diameter of the tube
A	heat transfer area
D	Diameter of tube
ID (D)	Inner diameter
OD	Outside diameter
Dx	Differential of a x segment
Δx	Delta x
L	Length of tube
A_t	Tube area
A_p	Plain area
ΔV	Voltage difference
I	Current
R	Resistance
R_t	Thermal resistance
$\sum R_t$	Sum of resistances
R_{cd}	Conduction resistance
R_{cv}	Convection resistance
R_{tot}	Total resistance
R_{tot}	Sum of total resistances
d	Differential

LIST OF FIGURES

Figure 1. CMS Overview (INSTITUTE OF PHYSICS PUBLISHING AND SISSA, 2008).13
Figure 2. HCAL Sections Overview (Ruchti, 2001)...14
Figure 3. Isometric view of a complete half barrel of the HB HCAL (Barney, 2003)............14
Figure 4. Isometric view of the HB wedges, showing the hermetic design of the scintillator tiles (INSTITUTE OF PHYSICS PUBLISHING AND SISSA, 2008)..................................15
Figure 5. An r-z schematic drawing of quarter of the HCAL section showing the RBx location (CERN Geneva The LHC Experiments Committee, 1997).....................................16
Figure 6. Isometric view of a wedge mockup showing the RBx location (Ruchti, 2001).......16
Figure 7. Drawing of a Readout box and its components (Schmidt, 2018)............................17
Figure 8. Drawing of a RM without covers (Schmidt, 2018)...18
Figure 9. Detail of copper tubes on the shell of the RBx (Schmidt, 2018)..............................20
Figure 10. Temperature distribution of electronics in the RBx (Schmidt, 2018)....................20
Figure 11. Illustration conduction mode...23
Figure 12. Illustration convection mode...24
Figure 13. Thermal resistance analogy...26
Figure 14. Boundary layers. a) Velocity boundary layer. b) Thermal boundary layer [Incropera, 2011]..30
Figure 15. Thermal energy ilustration..32
Figure 16. Heat load from electronics in the RBx (Schmidt, 2018)..33
Figure 17. Graphical model for the mathematical analysis...34
Figure 18. Conservation of heat in the system..35
Figure 19. Temperature vs position system graph...36
Figure 20. Comsol Multiphysics Overview (Wollblad, 2017)..38
Figure 21. Comsol steps setup...39
Figure 22. Pattern proposals for tube floor. a) Proposal 1. b) Proposal 2 (CAE ASSOCIATES, 2009). c) Proposal 3 (Yang, 2009)...46
Figure 23. Tube variables description..46
Figure 24. Layout proposal design 1..50
Figure 25. Layout proposal design 2..51
Figure 26. Layout proposal design 3..51
Figure 27. Comsol model...52
Figure 28. Geometry setup for Comsol model..53
Figure 29. Material setup for Comsol model...54
Figure 30. Heat transfer in solids module setup for Comsol model.......................................55
Figure 31. Non isothermal module fluid flow setup for Comsol model.................................56
Figure 32. Non isothermal module heat transfer setup for Comsol model.............................57
Figure 33. Mesh setup for Comsol model...58
Figure 34. Study setup for Comsol model...59
Figure 35. Temperature function graph...61
Figure 36. Relationship between cooling area and temperature...62
Figure 37. Pressure change in function of length and number of lengths...............................63
Figure 38. Proposal design 1 result...64
Figure 39. Proposal design 2 result...64
Figure 40. Proposal design 3 result...64
Figure 41. Initial temperature distribution of the system at t = 30 [min]................................65
Figure 42. Temperature distributon system of the present design at t = 35 [min].................66

Figure 43. Temperature distribution system at t = 35 [min] a). Design 1................................67
b) Design 2. c) Design 3. 67
Figure 44. Heat flux distribution at t = 35[min]. a). Design 1 68
b) Design 2. c) Design 3. 68
Figure 45. Floor temperature distribution at t = 35[min]. a). Design 1 70
b) Design 2. c) Design 3. 70
Figure 46. Module 1 temperature distribution at t = 35[min]. a). Design 1 71
b) Design 2. c) Design 3. 71
Figure 47. Transient study of water at t = 35[min]. 72
Figure 48. Transient study of module 1 from t = 30 [min] to t = 35 [min]..............................73
Figure 49. Transient study of floor from t = 30 [min] to t = 35 [min]. 73
Figure 50. Experimental setup of the prototype cooling system. 77

LIST OF TABLES

Table 1. Known parameters of the RBx (Schmidt, 2018)..33
Table 2. Heat transfer analysis assumptions..34
Table 3. Comsol steps description 39
Table 4. Common nodes or features in each Comsol step...40
Table 5. Common nodes or features for heat transfer in solids module 41
Table 6. Common nodes or features for non-isothermal flow module in heat transfer...........43
Table 7. Common nodes or features for non-isothermal flow module in fluid flow...............44
Table 8. Specification and values for the general variables of the design proposals...............49
Table 9. Detail of the fill and modified variables of the design proposals 49
Table 10. Fluid properties results...60
Table 11. Length per design..65
Table 12. Centralized weight method 74
Table 13. Experimental Setup Proposed Budget 78

1. INTRODUCTION

The curiosity of humanity to understand the origin of matter has deliver results such as the Higgs Boson or the God´s particle. This investigation was concluded by using the LHC from CERN laboratories. The LHC particle accelerator is located at Geneva, Switzerland and has a 27 km diameter tunnel, installed 100 meters underground. The main motivation for the development of the LHC project, is to probe the structure of matter and the forces that control it by yielding collision of two protons. Furthermore, the observation of these particles could only be done in special detectors as the CMS experiment (Figure 1).

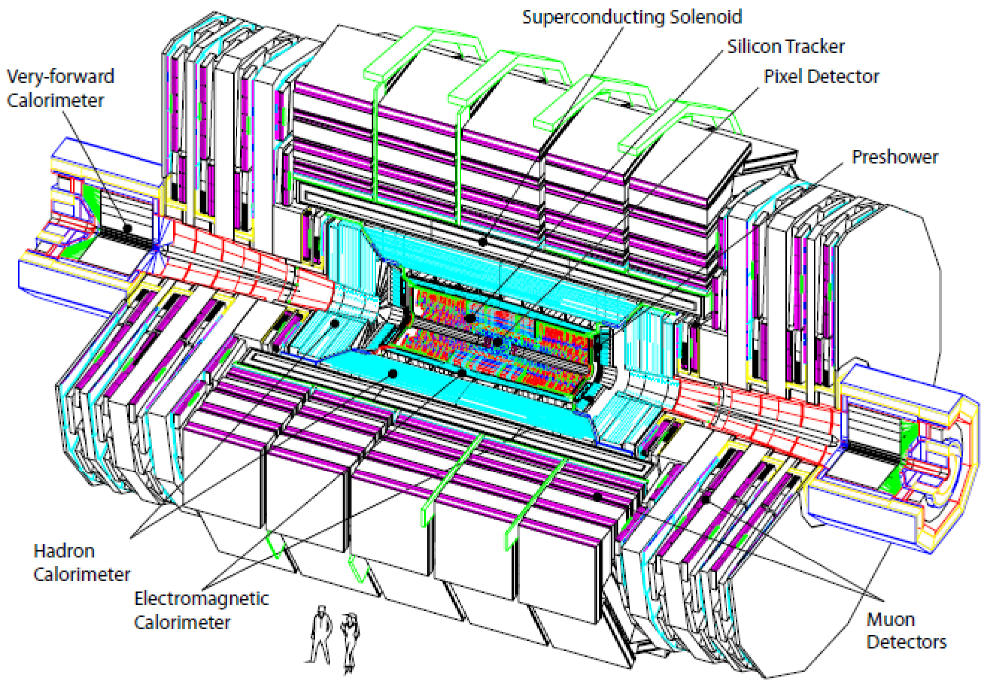


Figure 1. CMS Overview (INSTITUTE OF PHYSICS PUBLISHING AND SISSA, 2008)

The CMS is an assembly of different subsystems that its configuration defines the measure of momentum, energy and identification of a particle. One of these subsystems is the HCAL, it has the function to identify and measure jets, b-quarks, taus particles and transverse energy; specially hadrons which gives its name (CERN Geneva The LHC Experiments Committee ,1997). The HCAL is divided in three sections: HB, HF, HE (figure 2); corresponding to each part specific features to distinguish the type of particles they can detect. The HB is composed of two half barrels of 18 wedges per barrel, sums up to 36 wedges (figure 3). A wedge is a compartment which structure is configure as a sandwich of brass plates in the inside and stainless-steel plates on the outside faces (figure 4). Moreover, the HB houses the front-end electronics and the back-end electronics which are responsible for the data transmission of the proton-proton collisions events in the CMS (The Phase-2 Upgrade of the CMS Barrel Calorimeters TDR, 2017).

Figure 2. HCAL Sections Overview (Ruchti, 2001)

Figure 3. Isometric view of a complete half barrel of the HB HCAL (Barney, 2003)

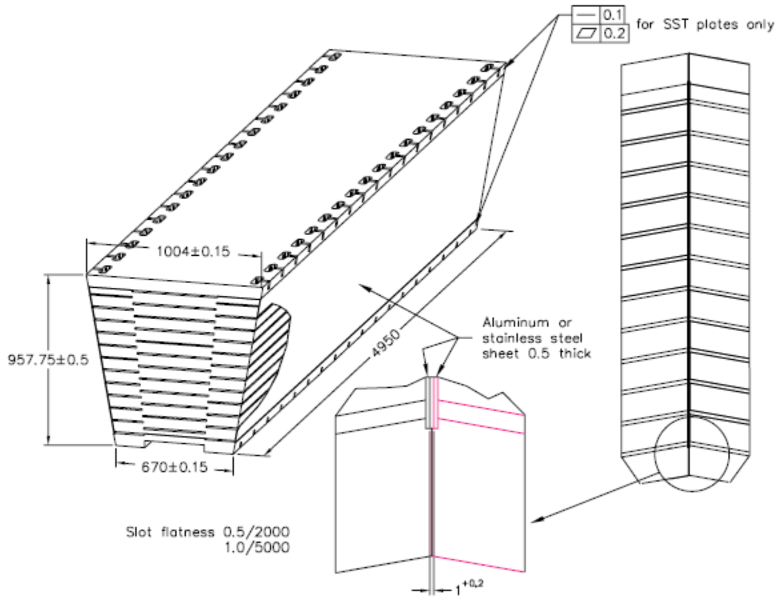


Figure 4. Isometric view of the HB wedges, showing the hermetic design of the scintillator tiles (INSTITUTE OF PHYSICS PUBLISHING AND SISSA, 2008)

Furthermore, the HCAL integrates an optical system that converts the energy from the collisions of the proton-proton particles to a digital signal. The optical system started with the collection and storage of the energy in the brass plates from the collisions. Then, the light is emitted by the scintillator tiles which are special thin plates that its material reacts to energy and convert it to light; they are visible by using wavelength shifting fibers which are embedded in the tiles. The energy is further transported in special fibers called optical fibers to the front-end electronics. Here, the light is detected by a photodetector which account to convert the optical signal (light) to an analog signal. At last, all the data process in the optical system is sent to a room outside from the CMS.

The FEE lies in each wedge, specifically in a component called RBx; that seems as a rectangular box. Each wedge has a cutoff in its back part to house the RBx.; Figures 5 and 6 detail the location of the RBx in the HCAL.

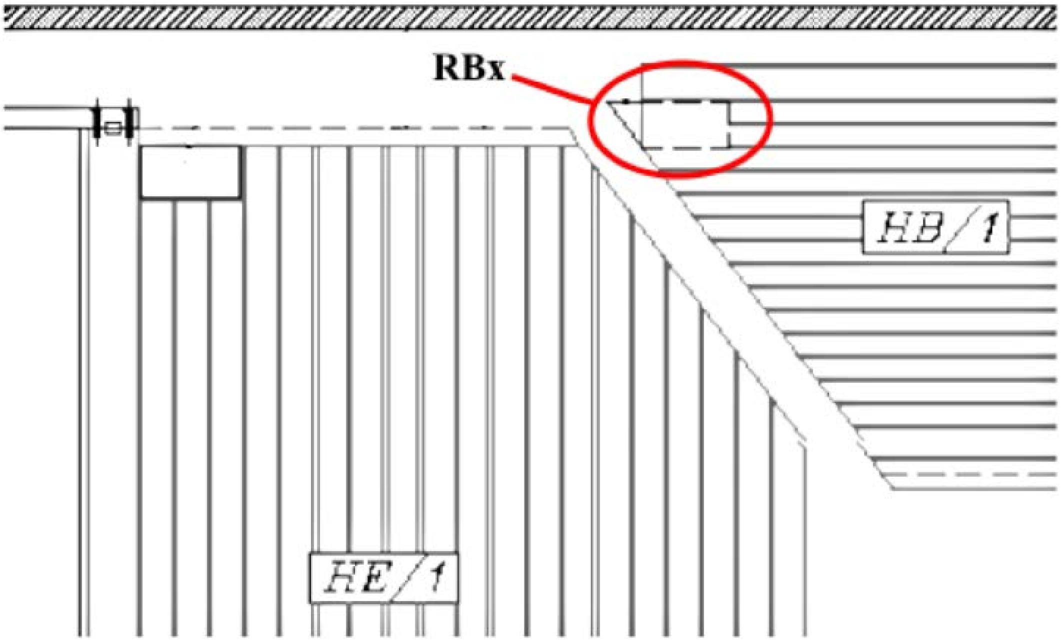


Figure 5. An r-z schematic drawing of quarter of the HCAL section showing the RBx location (CERN Geneva The LHC Experiments Committee, 1997)

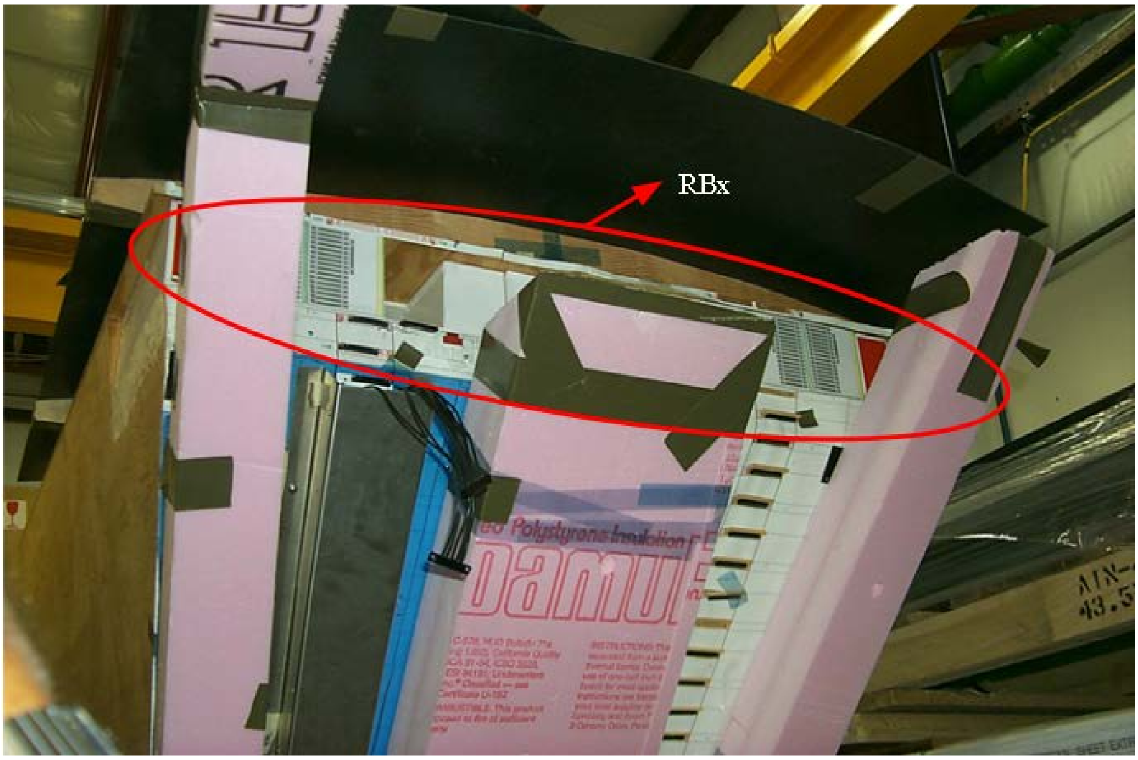


Figure 6. Isometric view of a wedge mockup showing the RBx location (Ruchti, 2001)

The RBx is a compartment that houses components that oversee converting, digitalizing, and transmitting the information readout by the HCAL detector. The components of the compartment (Figure 7) and the description of their particular function is listed as follows :

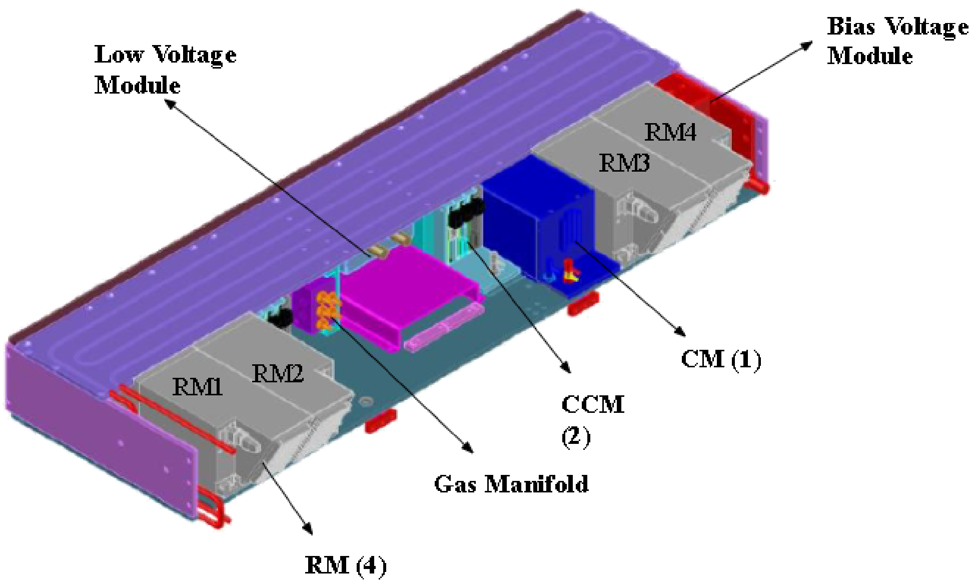


Figure 7. Drawing of a Readout box and its components (Schmidt, 2018)

Main components

- 4 Readout Modules (RMs): which have the function of signal conditioning and data transmission of the energy deposit of hadrons. Each RM (Figure 8) contains:
  - ***Electronic boards:*** codify the signal from optical to electrical with the use of and HPD, and then convert the analog signal to digital signal by using the QIE cards. Each RM contains 4 QIE cards.
  - ***A photodetector:*** it acts as an image intensifier, because it converts the optical signal into an image. Before, HPDs where the photodetectors, but in the LS2 they were replaced by the SiPMs, due to its benefits to better function in those conditions (Mans, 2012).
  - ***A Peltier Cooler:*** is the cooling system for the photodetector, implemented in the LS2 (Mans, 2012).

- ***An ODU:*** is the receiver of the optical fibers that comes from the scintillator tiles.

- 1 Calibration Module (CM): it generates a led pulse through a laser pulse for calibration.
- 1 Clock and Control Module (CCM): oversees controlling the other modules.

Secondary components

- Bias voltage module: regulates the required voltage to the photodetector.
- Low voltage module: regulates the required voltage to the FEE.

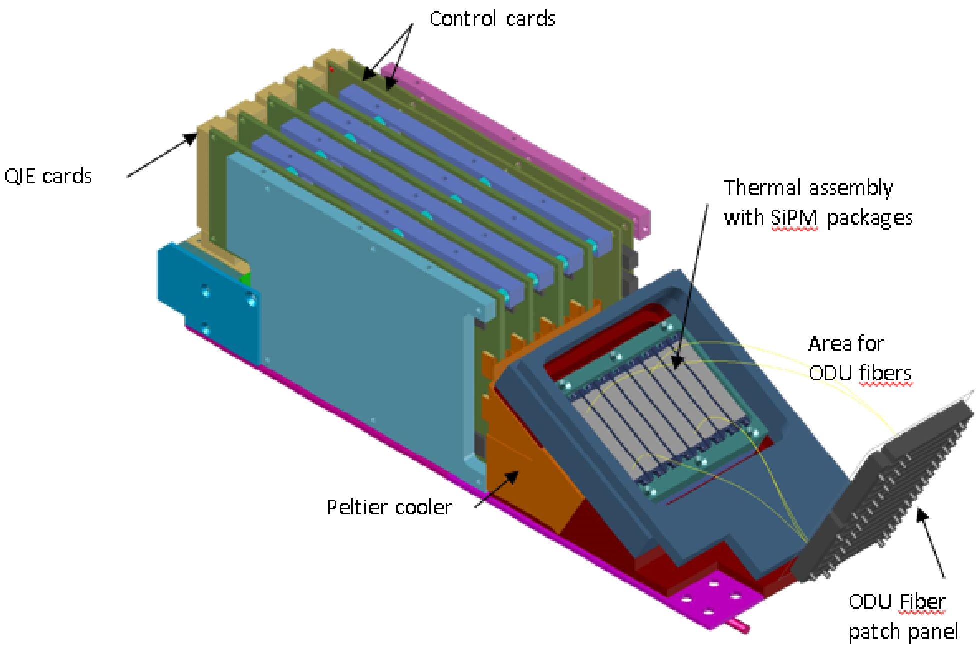


Figure 8. Drawing of a RM without covers (Schmidt, 2018)

The LHC is undergoing over a series of upgrades to optimize the detection and storage of the collisions. In the phase 2 such changes as the integration of the peliter cooler and modifications of the modules of the RBx has be done. Nowadays, to a better performance the cooling system has to be upgrade as well, which is the main objective of this senior thesis (Butler, 2011).

The objective of the project is to design the cooling system for the RBx, to dissipate the heat generated from the modules. It has a previous design which consisted of copper pipes with water, that are embedded over the roof and floor of the RBx (figure 9). In the past, when the RM didn't integrate the SiPMs and the Peltier cooler, the cooling system could remove 237 W of heat approximately (Karmgard, 2001). An upgrade process was implemented to include the SiPMs and the Peltier cooler, this implementation supposes and increase the amount of heat that was needed to be removed, the upgrade process established 255 W of heat to be removed without the use of the Peltier cooler and 387 W with the Peltier cooler (Schmidt, 2018). However, nowadays the RBx is working in conditions that are affecting its performance, because the present design of the cooling system is not dissipating enough heat from the electronics, which creates increases in the temperature of the electronics component, lowering their efficiency. Indeed, as Schmidt shows in a graphic of Temperature vs Peltier voltage (figure 10), the Peltier cooler is not functioning properly. This statement is justified by looking up how the temperature of the QIE cards increases as the voltage of the Peltier cooler rises. In view of this, the results suggest that the Peltier cooler is removing the heat towards the FEE. For this reason, it is necessary to design a better RBx cooling system, that allows to decrease the temperature of the electronic components in the RM's by 10ºC, which is a better working temperature for the QIE cards.

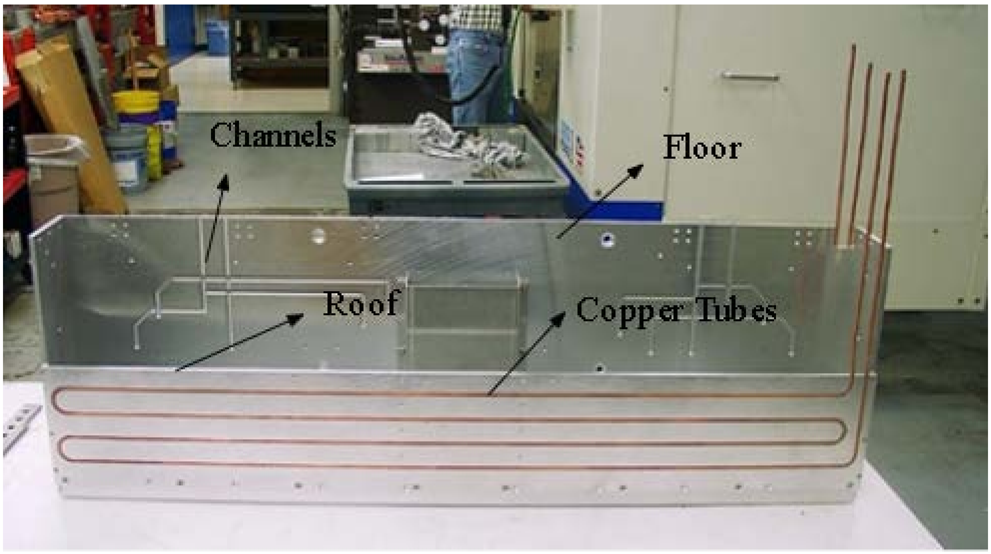


Figure 9. Detail of copper tubes on the shell of the RBx (Schmidt, 2018)

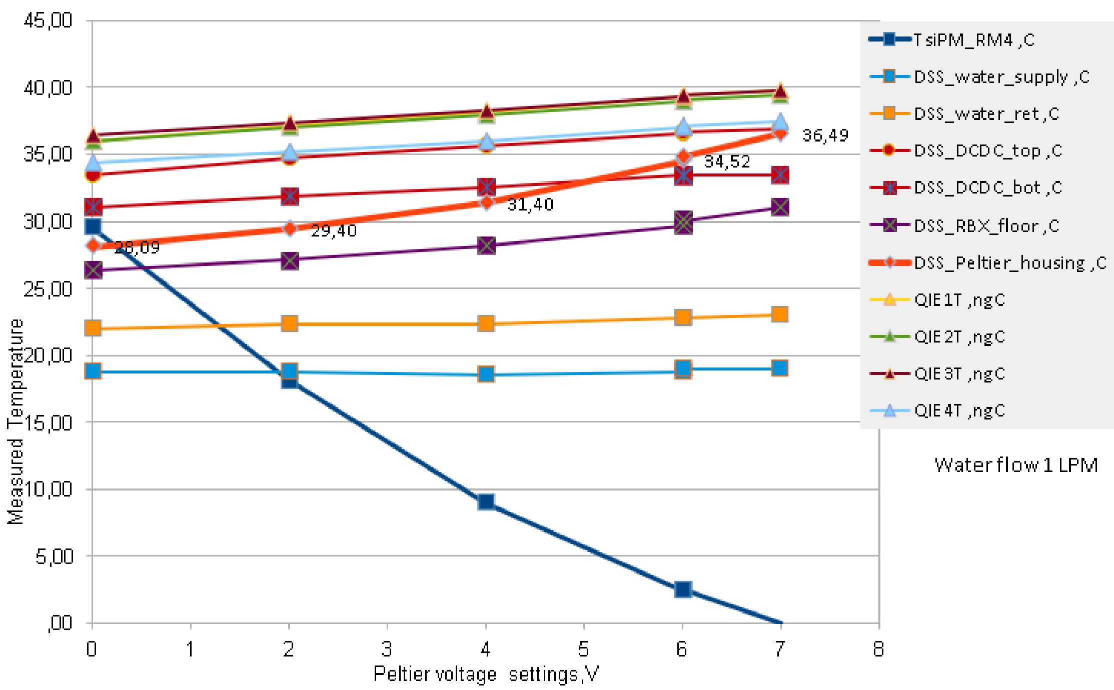


Figure 10. Temperature distribution of electronics in the RBx (Schmidt, 2018)

Where:

- TsiPM_RM4: is the temperature of SiPM phototedector in RM # 4.
- DSS_water_supply: is the temperature of the water supply to the cooling system.
- DSS_water_ret: is the temperature of the retainer that holds the water.
- DSS_DCDC_top: is the temperature in the top part of the DC component of the RBx.

- DSS_DCDC_bot: is the temperature in the bottom part of the DC component of the RBx.
- DSS_RBx_floor: is the temperature of the floor of the RBx
- DSS_Peltier housing: is the temperature of the box that contains inside the peltier.
- QIE: is the temperature of the QIE card.

The goal of this project is to design a new distribution layout for the copper tubes on the floor of the Rbx. Therefore, it will help dissipate the heat produced by the electronics in the RBx and reduce the temperature of these components.

2. METHODOLOGY

2.1 Analytical Design

In order to design an analytical model that will reduce the temperature of the QIE cards, a review of the fundamentals of the physics involved in the system must be done.

2.1.1 Fundamentals of heat transfer.

Heat transfer phenomenon exist when there is a difference in temperature between two sides. The heat is exchange from the hot side to the cold side, until a steady state of the system is reached, and a temperature distribution is set. Giving so, this physics studies how the heat process is developed, meanwhile a thermodynamics approach of know the initial and final state of the system. Heat transfer processes or modes such as: conduction and convection are described below.

Conduction

This process features heat transfer due to a temperature difference within a solid or physical media. Examples as: heat generation of a chip pc circuit causes the circuit to be hotter than other physical parts, so the heat transport from the chip to other sides, thus, they get hot. It is described analytically by the Fourier´s law, that in 1D is:

$$q_x = -kA\frac{dT}{dx} = -kA\frac{T_1 - T_2}{\Delta x} \qquad (1)$$

Next, is a figure to understand this heat transfer process.

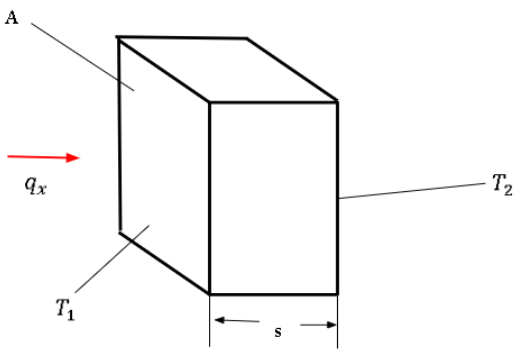


Figure 11. Illustration conduction mode

In addition, sometimes is useful an expression to indicate the direction of heat, as a vector. This expression namely heat flux is function of the area and heat rate. Its expression in the x direction is:

$$q''_x = \frac{q_x}{A} \tag{2}$$

Convection

This process features heat transfer between the temperature difference of a solid and a fluid. Typical examples as the experiment to take out the hand of a car in movement, gives the sensation that the hand feel cold due to the convection process that acts around it. As the example, sometimes the solid could be in movement or the opposite could also occur, when the fluid is in movement. This mode is described by Newton´s law of cooling, which is:

$$q = hA\Delta T_{cv} = hA(T_S - T_\infty) \tag{3}$$

Where the $A = \pi(ID)L$.

Like in conduction, a figure is used to understand better the convection process.

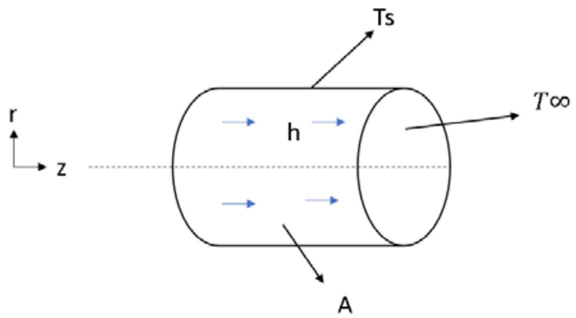


Figure 12. Illustration convection mode

Temperature distribution: Heat Diffusion equation

Furthermore, is desire a temperature function in terms of position, that gives the temperature distribution in the system. For this, establish a differential analysis in a control volume, where applies the first law of thermodynamics. From basic knowledge of a thermodynamic course it is know that.

$$E_{in} - E_{out} + E_g = E_{st} \tag{4}$$

This equation associate with the heat transfer phenomena and heat conduction, uses Newtow´s law for the inlet and outlet energy exchange. In addition, it considers the heat generation that can be produced in the system. At last, the storage term considers the energy variation of temperature in time. Then, it can be established an equation that from it the temperature is obtained as function of time and position. However, is necessary to define the coordinate system which will be used. For a cartesian coordinate system, the equation is:

$$\frac{d}{dx}\left(k\frac{dT}{dx}\right) + \frac{d}{dy}\left(k\frac{dT}{dy}\right) + \frac{d}{dz}\left(k\frac{dT}{dz}\right) + \dot{q} = \rho C_p \frac{dT}{dt} \tag{5}$$

This equation is known as the heat diffusion equation. Several assumptions can be done, to define an adequate expression for the system to analyzed. Under steady state condition, 1D dimension, it can be found that the equation reduces to.

$$\frac{d}{dx}\left(k\frac{dT}{dx}\right) = 0 \qquad (6)$$

The solution of the equation can be determined using the appropriate boundary conditions, which can be constant heat flux, constant temperature, adiabatic surface or convection-conduction boundary condition. It is critical to select the correct boundary, because the results for each one is different. Understanding how the system interacts within or with the limits, helps to figure out the type of boundary condition.

Composite walls

In more complicated systems composed of different walls either in parallel or series, heat transfer processes mention earlier are applied to it. Further, is define an expression that overview the system and makes the analyze easier, which is:

$$q = UA\,\Delta T_{UA} \qquad (7)$$

This is known as the overall heat transfer coefficient equation. To emphasize, the temperature difference in the equation refers to two selected temperatures, so, it could be the temperatures in the sides of a layer, or it could be the temperatures of a system limits. Whenever this equation is use in 1D, there is the assumption that the parallel walls to the heat flux are adiabatic.

The composite walls use the thermal resistance concept and make an analogy to heat transfer. From Ohm’s law it is known that in circuits:

$$\Delta V = IR \qquad (8)$$

Therefore, the voltage difference is analogy to the temperature difference, due that both variables cause a flow of something to go through the system. In the case of Ohm´s law is the current, while in heat transfer is the heat rate. The flow in both cases experiences a resistance to flow, in heat transfer it is called thermal resistance. Considering this, it can be established the following equation:

$$R_t = \frac{\Delta T_{UA}}{q} \tag{9}$$

A visual representation of this analogy is shown below.

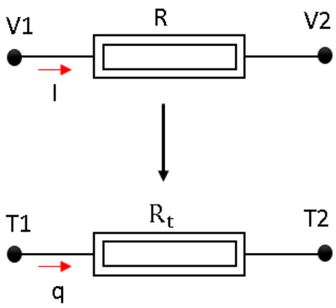


Figure 13. Thermal resistance analogy

In addition, the relationship between the thermal resistance and the overall heat transfer coefficient, is described by:

$$UA = \frac{1}{R_{tot}} \tag{10}$$

If more than one resistance is present in the system, then:

$$R_{tot} = \sum R_t \tag{11}$$

The expression for the conduction and convection modes thermal resistances are:

$$R_{cd} = \frac{\Delta x}{k * A} \tag{12}$$

$$R_{cv} = \frac{1}{h * A} \tag{13}$$

2.1.2 Fundamentals of fluid mechanics.

This physics overview the development, forces, conditions that the fluid experiences in a continuum movement. To analyzed properly a fluid, the flow condition and the fluid type must be determined first. There are two types of fluids: laminar or turbulent, which depends on the fluid velocity. A laminar fluid characterizes to be smooth with streamlines parallel to each other. On the other hand, a turbulent flow is a chaotic fluid which drown to vortices inside the fluid, to mix it. The velocity in the second type is higher than the first type. Also, there is a transition zone from the laminar to the turbulent flow; in this zone the fluid shares a mix of both flow types features.

Flow conditions

To determine the type of flow a dimensionless number know as Reynolds. It helps to identify the type of flow by a numerical range, which separate the laminar from the turbulent region. Also, it depends where the fluid goes through, outside or inside the system. For example, the cooling of a chip using a blowing fan is an external flow and a heating ventilation system using cooper lines is an internal flow. The Re definition is:

$$Re = \frac{\rho u D}{\mu} \qquad (14)$$

However, in a pipe fluid analysis the Re can be computed using the mass flow rate[1].

$$Re_D = \frac{4\dot{m}}{\pi D \mu} \qquad (15)$$

In such cases, a transition number know as critical Re (Re_{cr}) is established.

[1] It is a more common used variable in situations with pipes

Losses by friction in pipes

Furthermore, the fluid experiences forces on it when it goes through a pipe. Such forces like: body and contact. Body forces correspond to the gravity. A contact force is the friction force that creates between the walls and the fluid. It depends on the rugosity of the material of the pipe, identify it by smooth or a rough pipe. Also, the friction force has influence on the velocity of the pipe, due that it causes changes in the direction of the fluid at the solid-fluid interface. The equation that describes the effect of friction is the Darcy-Weisbach equation, which is:

$$h_f = \frac{fLv^2}{D2g} \tag{16}$$

The friction factor (f) is obtained from the Colebrook equation[2], that is applied either for any range of Reynolds number.

$$\frac{1}{\sqrt{f}} = -2\ log\ log\ (\frac{\frac{\varepsilon}{ID}}{3.7} + \frac{2.51}{(Re)f^{0.5}} \tag{17}$$

The velocity change also affects the pressure in the system when its experience forces, because they are directly proportional by the Bernoulli equation.

$$\frac{v_1^2}{2g} + \frac{p_1}{\rho g} + z_1 = \frac{v_2^2}{2g} + \frac{p_2}{\rho g} + z_2 + h_l \tag{18}$$

Where:

h_l accounts for major losses (h_f) and minor losses (h_m) in the pipe. Major losses are associate with the friction effects and minor losses to bends, fittings, contractions, expansion, etc. Thus:

$$h_l = h_f + h_m \tag{19}$$

[2] This equation is use instead of the Moody chart

Minor losses are determine with the following expression:

$$h_m = \frac{kv^2}{D2g} \qquad (20)$$

Fluids momentum conservation

Moreover, an essential part of fluid mechanics is the conservation of momentum. This overview how the fluid interacts with the solid interface. The layout of the pipe is an essential factor to consider when is analyzed. Especially when the pipe has bends, contractions or expansions. Bends causes the fluid to change direction and its momentum. The Navier-Stokes equation describes the change in momentum a fluid experiences in a fluid domain with its boundaries, which is:

$$\sum F = \frac{d}{dt}\int_{cv} \rho v d\underline{v} + \sum_{cs} \rho v(v * A) \qquad (21)$$

2.1.3 Conjugate Heat Transfer: Internal Flow.

For more complicated problems that deal with the coupling of heat transfer and fluid mechanics, the conjugate heat transfer physics is use and if is inside a pipe, is name internal flow.

Boundary layers

Due to the solid-fluid interaction, a boundary layer developed for both velocity and temperature variables. The boundary layer describes the graphical behavior of the profile of the variable in study, as follows. As a fluid is confined and a volumetric flow rate is set to it:

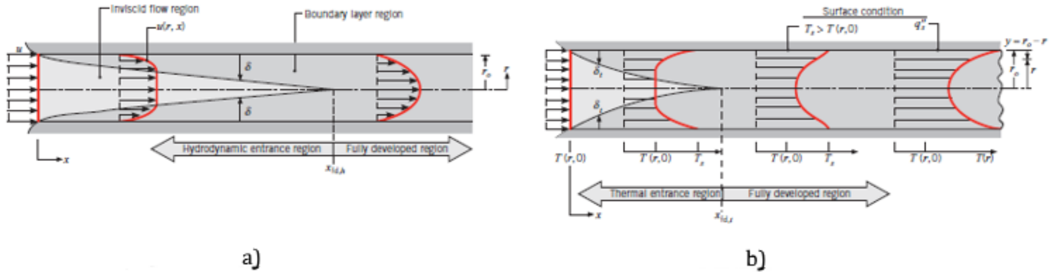


Figure 14. Boundary layers. a) Velocity boundary layer. b) Thermal boundary layer [Incropera, 2011]

As seen, the fluid starts from an entry region until it reaches a fully developed behavior. From the entrance to some point, a critical point (x_c) the fluid is unsteady than a constant behavior in the fully developed region. In problems where the relationship L/D is higher, it is considered that the fluid is fully developed. Several assumptions are taken under this consideration, one of them is that the velocity along the axis of the tube doesn't change. A second one is that the radial velocity is zero. A third one is that the temperature difference $(T_s - T_m)$ is independent of x. The last one is that the heat rate is only in the radial direction.

Mean Temperature

Further, is define a temperature known as mean temperature. It is used as a reference value to look up the thermo physical properties of the fluid.

$$T_m = \frac{T_{out} + T_{in}}{2} \tag{22}$$

It is applied in Newtons law of cooling equation, where it substitutes the temperature specified for the fluid (T_∞). Also, such dimensionless numbers as the Re used it to obtain the correct properties values.

Heat Transfer coefficient

Is possible to find a problem where the heat transfer coefficient is not defined, in those situations empirical correlations are used to find the convection coefficient. Firstly, to apply them another dimensionless number named Nusselt (Nu) has to be determined. This number evaluated the capacity of a system to dissipate heat by convection.

For most cases of internal flow in tubes with a small diameter, it is assumed a fluid turbulent behavior, thus, is suggested to use the empirical correlations for turbulent flow inside a tube. Among all the empirical correlations, the Gnielisnki correlation is suitable for the analysis, which is:

$$Nu = \frac{\left(\frac{f}{8}\right)(Re_D - 1000)Pr}{1 + 12.7\left(\frac{f}{8}\right)^{\frac{1}{2}}\left(Pr^{\frac{2}{3}} - 1\right)} \qquad (23)$$

The equation[3] uses variables well known from previous equations, however only Pr (Prandlt) is a unknow variable. Prandlt is an extra dimensionless number that features the properties of the fluid. Its value can be determined from its definition, but most of the books computed it for a list of temperatures. The Nu is related with the heat transfer coefficient by its definition:

$$Nu = \frac{hD}{k} \qquad (24)$$

In addition, in the case of a fully developed fluid it is assume that the $\underline{Nu} = Nu$ of the system, thus, $\underline{h} = h$.

[3] The Gnielisnki correlation is valid to apply for the range 0.5 < Pr < 2000 and 3000 < Re < 3E5.

Internal energy

In a heat transfer exchange between a solid and fluid, it can occur a change in the temperature of the fluid, meaning its internal energy increases. Regarding an incompressible fluid and a conservation of energy analysis, the following expression is defined.

$$q = \dot{m}Cp\Delta T_{IE} = \dot{m}Cp(T_{out} - T_{in}) \tag{25}$$

The equation known as thermal energy comes out under a sensible heat simplification and assumptions such as: no latent heat changes, no generation, negligible potential and kinetic energy, steady flow.

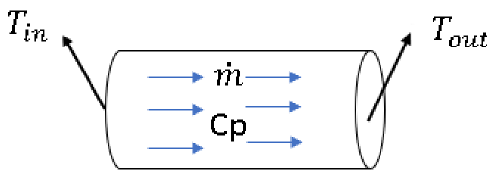


Figure 15. Thermal energy ilustration[4]

2.1.4 Fundamentals applied to the system.

Once review the fundamentals of fluid mechanics and heat transfer, they are applied to the problem of study. Then, it must be understood what is known from it to proceed to the correct analysis.

System overview

It is known: the geometry properties, that include the diameters of the tube to use and the dimensions of the RBx and its modules; the materials of the system, which are water for the fluid and Aluminum 6061 for the solid parts; the heat load specified by module from the RBx; the volumetric flow rate (Q) and the inlet temperature of the fluid. This information is

---

[4] The temperatures are taken as average temperatures computed over the cross section specified.

known from CERN repository cads and figure 16 shown below. Heat load is shown in figure 16 and parameters are summarized in table 1.

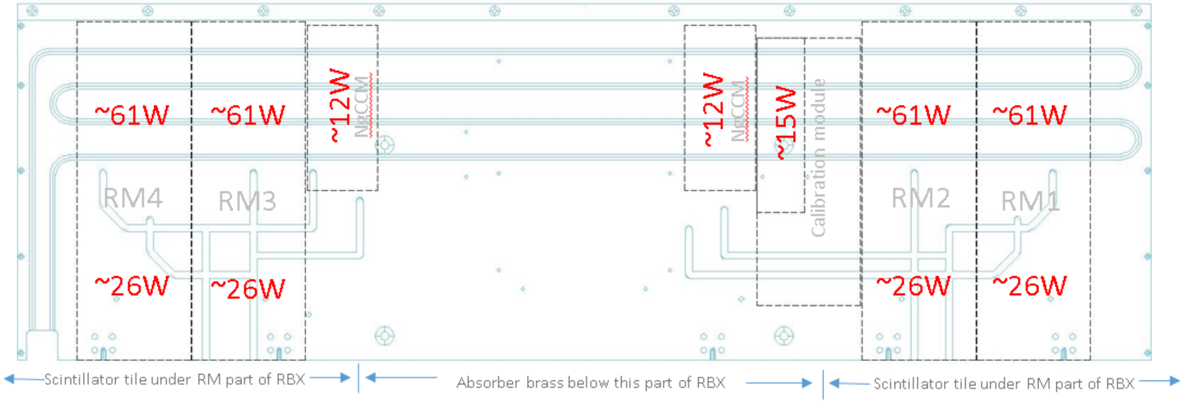


Figure 16. Heat load from electronics in the RBx (Schmidt, 2018)

Table 1. Known parameters of the RBx (Schmidt, 2018)

Variable	Value (unit)
Outside Diameter of the tubes (OD)	3/16 (in)
Internal Diameter of the tubes (ID)	1/8 (in)
Water Flow rate (Q)	1 (lpm)
Water Inlet temperature	20 (°C)
Tube length (L)	4.138 (m)

Considering the known parameters, is necessary to determine an expression that is function of the area or more specified the length of the tube, thus, with it compute the heat rate loss from the RBx to the tube and obtain the temperature distribution in the system. With the purpose to determine this expression, a series of assumptions and considerations that make easier to compute results are established, these assumptions are the following:

Table 2. Heat transfer analysis assumptions

One dimensional analysis	Cooper resistance is negligible
Steady state system	No fouling consider
Fully developed flow	No corrosion consider
Incompressible fluid without phase changes	Cooper resistance is negligible
No conduction in the fluid, just convection	No radiation consider
Isotropic materials	No contact resistances consider
Bousiness approximation taken for the fluid[5]	

A mathematical model is proposed for the system. The RBx is a system which is composed of two main parts: the tubes and the components (modules and walls of the RBx). The tubes stands for the cooling system, which are embedded in the roof and floor of the RBx. However, there is a difficulty in the system to perform easily an analysis, because of the mismatch of the coordinates system. The tubes fundamental theory works in cylindrical coordinates, while the components works in cartesian coordinates. Therefore, the objective is to analyze each part separated, and make a relationship between them by the heat transfer effect. The graphical model set up for the mathematical model is:

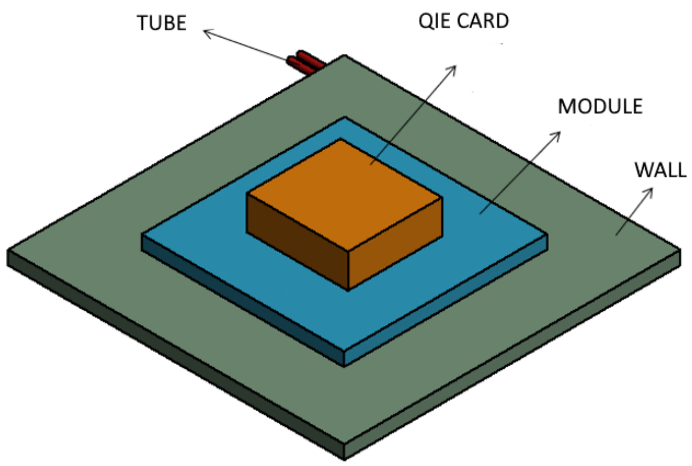


Figure 17. Graphical model for the mathematical analysis[6]

[5] The density of the fluid is constant independent of time

[6] The model can be applied to the lower part of the system or for the top, just change the type of floor and dimensions

Where:

QIE card: stands for the whole bunch of QIE cards in the modules.

Module: stands for the total wall thickness of the modules in the RBx (CM, CCM, RM).

Wall: stands for the external wall thickness of the RBx.

Tube: stands for the tube that belongs to the selected wall.

Then, a simplification for the components of the system, where they are considered a whole of the same type and not by the individual approach, looking for an easier way to compute the results

The approach given for the mathematical model considers the conservation of heat in the system concept, thus, to said that the same amount of heat released from the QIE cards is the same that goes through the external walls, as shown in the figure.

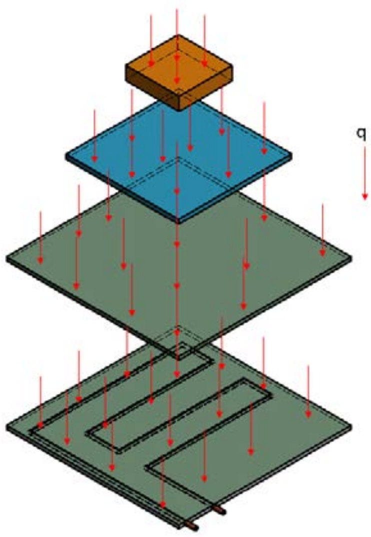


Figure 18. Conservation of heat in the system [7]

Moreover, to couple the systems together is consider the effect of the cooling system established by Newton´s law of cooling. A common way to increase the heat transfer

[7] From previous assumptions, it was set an 1D analysis, so then it explains why the arrows are draw just in one direction.

convection is by increasing the h, however, is expensive. Therefore, the suitable option is to increase the tube length and then the area of the tube occupied

inally, using equation an expression of temperature as function of length of the tube is obtained.

In addition, once the thermal circuit has been set, the expected temperature distribution in the system could be approximate to the following graph, if the two co-systems are joined together.

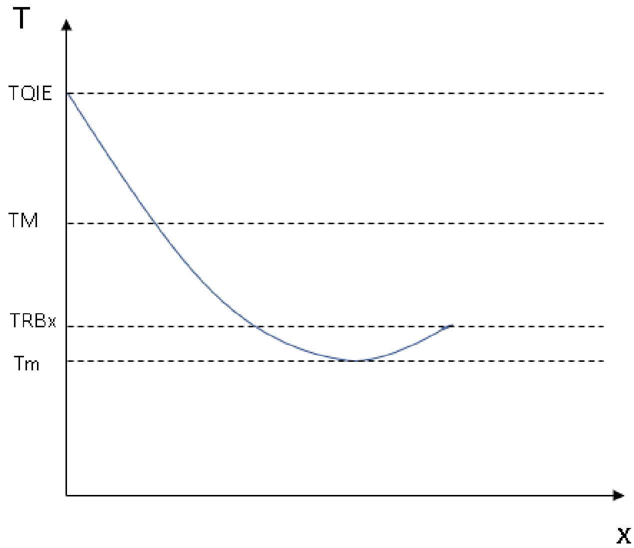


Figure 19. Temperature vs position system graph

Figure 19 lists the temperatures that exist in the system: temperature of the QIE card, temperature of the module, temperature of the RBx and the temperature of the fluid. It is observed from the QIE temperature to the temperature of the RBx wall (floor/roof) developed a linear relationship, that match with the Fourier’s law equation. However, after it a parabola profile developed due to the behavior of the temperature inside the tube, justify by the temperature profile in the boundary layer section.

Under this graphical model and mathematical model, it has been proposed a guide with a series of steps in order to obtain an expression which relates the length of the tube and temperature. As detail below:

1.- Determine the T_{out} of the fluid by using: q $= \dot{m}Cp\Delta T$

2.- Obtain T_m and use it to compute the fluid properties

3.- Determine the $\dot{m}$ by using the continuity equation and then compute the Re by using equation 15. Once the Re is obtained, the friction factor can be determined from the Colebrook equation. Using the results of the f and Re, the Nu can be computed from the Gnielisnki correlation. Then, h is obtained from the Nu definition.

4.- Set the thermal circuit for the system and determine the resistances and its expressions by using equations 12 & 13. Further, compute the total resistance by using equation 11 in terms of the length of the tube.

5. – Using equation 10 the UA can be obtained, and at last using equation 7 the function of the temperature difference in terms of the length can be obtained.

2.2 CFD Analysis

2.2.1 Comsol Multiphysics Overview.

An easy way to prove the function of a design proposal is to perform a simulation with a CAE software that allows to do it. One of this software´s, is Comsol Multiphysics that uses the finite element analysis approach to solve problems. The overview of Comsol Multiphysics is:

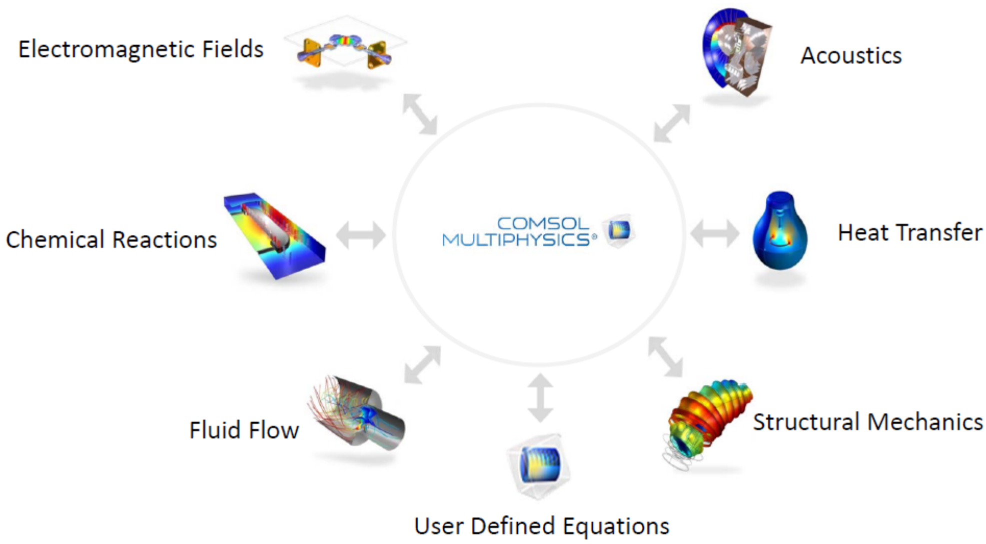


Figure 20. Comsol Multiphysics Overview (Wollblad, 2017)

This figure shows that the software is linked to different physics and they can be added individually. To each physic Comsol name it as module and each one has submodules. For example: Fluid Flow module has submodules like single-phase flow, non-isothermal flow, thin-film flow, etc. In addition, it might exist linked to each submodule a variety of types of the submodule study. Due to the great capacity of the software to couple physics one with another, it is given the name Multiphysics.

The setup of the steps that Comsol Multiphysics follows to solve a problem, is specified below.

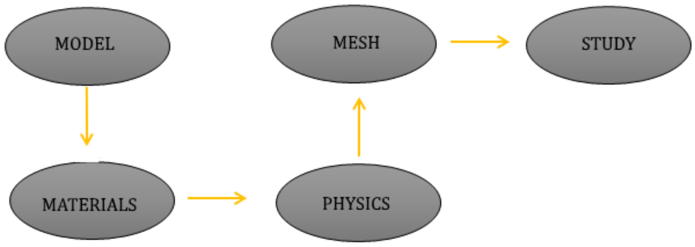


Figure 21. Comsol steps setup

Where:

Table 3. Comsol steps description

Step	Description
Model	a geometry is imported from a CAD software or is draw
Materials	add the materials to the parts of the system
Physics	add the physics (module) to use for the problem
Mesh	set the mesh to divide the system in finite elements[9]
Study	set the type of study for the problem. For example: steady-state or transient

Comsol Multiphysics has a special tool that when two physics are closely related, name Multiphysics coupling. For example: flow with heat transfer. The function of this tool is to add the coupling to relate both physics together when the software is computing. It gives the user the opportunity to set special conditions to consider in the solution.

Furthermore, as other CAD-CAE software's Comsol has the structure of a tree. To understand, recall the steps that need to be set up to perform a simulation analysis. Each of the steps has its own features, name nodes and in each node set up features with parameters or variables. In short, it is common to find in these nodes features as:

[9] Small details or high level of detail expected a finer mesh for more resolution of the elements.

Table 4. Common nodes or features in each Comsol step

Step	Nodes/Features
Model	Import, draw functions for 1D, 2D and 3D, etc
Materials	Browse materials, add new material, material link, etc
Physics	Depends on the physics selected. If for example heat transfer is selected, the features are: Temperature, Inflow, outflow, heat source, etc
Mesh	There are two types, the default one or the user defined. In the default Comsol use the most convenient way for it to do a mesh. However, in the user defined is specified the type of element in 3D, 2D or 1D.
Study	is specified the physics to analyzed and the variables. For example, in the transient study: the time range is set. Also, it gives the opportunity to link one study to another with the values of dependent variables option

In general, Comsol integrates a database where all the equations are specified for each physic. It uses then the corresponding equations, to compute and solve the problem. Indeed, each physic defines a list of variables to solve for, so, when it solves the problem; in the result section these variables can be view.

Considering the fundamentals review and related to the problem of study, it is selected the two more appropriate modules: heat transfer and fluid flow. The submodules selected are heat transfer in solids and non-isothermal pipe flow. Their detail is shown below.

2.2.1.1 Heat transfer in Solids module.

This module analyzes heat transfer modes, such as: conduction, convection and radiation in a solid. However, it can specify a fluid domain in the system and this field is recognize for the software as fluid. The variable solved in the module is the temperature. The most common nodes or conditions that are set up in this module are:

Table 5. Common nodes or features for heat transfer in solids module

Node/Feature	Description
Temperature	B.C to set a boundary with a specified temperature
Heat flux	Set a boundary with the ability to generate or exchange heat. It can be set a general inward heat flux condition, a convective heat flux condition and a heat rate condition
Thermal insulation	By default, those boundaries that are not specified by a condition, are set up in this condition. It defines an adiabatic wall condition, meaning that there is not heat transfer interface due that it lacks a temperature difference
Heat source	Set a boundary or a domain a heat generated
Initial values	Set the initial values for a transient analysis regarding the variable solve for in the module.
Solid	Associate the domain selected to a material

The equation that is used in the heat transfer in solids module is the following:

$$\rho C_p \frac{dT}{dt} - \nabla(k\nabla T) = Q \tag{29}$$

As noticed the equation comes from the first law of thermodynamics. Is like the heat diffusion equation but displayed in a different way. The first term accounts for the energy change in time; the second accounts for the conduction term; and at last the term in the right accounts for the heat generation in the system. For steady-state problems, the first term is zero (COMSOL, 2014).

2.2.1.2 Non isothermal pipe flow module

This module model and simulate fluid flow, heat, and mass transfer in pipes and channels. Model 3D pipes as curves in 2D or 3D. The properties are cross section averaged quantities. The variables solved for in this module are: velocity, pressure and velocity. It includes automatic coupling to the surroundings, so it can exchange properties with other modules, for example: heat transfer. Includes correlations for forced and natural convection. Further, it integrates tools as: surface roughness, friction factors, wall layers, pipe geometry, water hammer, acoustic in pipes to consider for the analysis. This module resolves and compute in less time due that it uses 1D edges. Moreover, for possible pipe connections such elements as: manifolds, fittings can be added to it.

Heat transfer in pipes is computed by an energy balance in the pipe. Then, solves a temperature equation for a fluid transported in a pipe. Model heat transfer for an incompressible fluid in 1D. The most common nodes that features are:

Table 6. Common nodes or features for non-isothermal flow module in heat transfer

Node/Feature	Description
Heat outflow	B.C to set when an inlet condition at the entrance of the tube is set. Also, it is specified to establish a convection dominate heat transfer condition in the tube. Heat Source: set a part of the tube which generate heat
Heat source	Set a part of the tube which generate heat
Heat transfer	Define heat convection and conduction properties
Initial values	Set for a transient analysis, considering the values solved for in this module
Temperature	Set a temperature for a point in the system
Wall heat transfer	Set up the heat exchange across the pipe wall. When this condition is set up, it is needed to set up an external temperature and an internal or external film resistance
Pressure	Set up this condition in the opposite end point of the pipe, when a temperature node is set
Wall layer	set up a layer to the pipe with its properties

Wall layer: set up a layer to the pipe with its properties.

The equation that this module uses to solve the variables is:

$$\rho A C_p \frac{dT}{dt} + \rho A C_p u \nabla T = \nabla * Ak\nabla T + f_D \frac{\rho A}{2d_h} \ |u^3| + Q + Q_{wall} \qquad (30)$$

This equation is known as the heat balance equation. Is like the heat equation for heat transfer in solids module, but the difference is that it considers other source energy terms as consequence of the velocity. The second term in the left side accounts for the transient state of the fluid system, like the first term in the left side. Then, the second term of the right-side accounts for friction heat dissipated due to viscous shear. The term Q corresponds to a general heat source and the right-side term accounts for an exchange heat term between the pipe wall and the surroundings. Its expression is defined as:

$$Q_{wall} = (hZ)_{eff}(T_{ext} - T)\ (\frac{W}{m}) \qquad (31)$$
[10]

Q_{wall} is a mix of the overall heat transfer equation and Newtown's law of cooling. It has the same approach of the second equation and its compute like the first equation. The expression $(hZ)_{eff}$ accounts the total thermal resistances between the pipe inside and the external temperature, where effective is name because it uses an effective heat transfer coefficient and Z is the wall perimeter of the pipe (COMSOL,2012).

Moreover, the non-isothermal pipe flow includes the fluid interface. As the heat transfer physics, is detail the most common nodes encounter for this physic.

Table 7. Common nodes or features for non-isothermal flow module in fluid flow

Node/Feature	Description
Inlet	set an inlet condition for the tube. A volumetric flow rate, a velocity or a mass flow rate.
Initial values	set initial conditions for a transient analysis.
Pipe properties	set the geometry type and type of friction model with its roughness. Comsol hold out for a variety of friction models: Churchill, Haaland, Wood, Colebrook and others.
Pressure	set condition if inlet condition is set.

Also, it can be added special components or tools as: valves, bends, pumps, etc. Such components are only added when the pressure conditions depend on it.

The equations used for the fluid flow branch in the non-isothermal module are:

$$\rho \frac{du}{dt} = -\nabla p - f_D \frac{\rho}{2d_h} u\,|u| + F \qquad (32)$$

[10] Text can be user defined set or given by another computed physics field

This equation is the momentum equation and is like the Navier-Stockes equation set before, with an additional term and displayed in a different way. Some terms are common from both equations, however, the second term in the right-side accounts for the pressure drop due to the viscous shear. The F expression is a volume force term.

And the second equation is:

$$\frac{dA_{\rho}}{dt} + \nabla * (A\rho u) = 0 \qquad (33)$$

Which is the continuity equation.

2.2.2 System design and analysis

Since there are some limitations in the modeling due to the mesh, it is selected an approximated model to the real one that represents mostly the system. The details neglected for this model are: holes, filets and chamfers. Also, elements irrelevant to the physics analyzed as: wire, extra modules. However, firstly what is consider is the tube design in the floor wall, because the system depends on it.

2.2.2.1 Overview of tube design

In general, a cooling system main task is to exchange heat from a source to a destination. Thus, the system lowers its temperature from its initial condition. It can change in its design and application, but it always searches to accomplish to reduce the temperature. The selected one in this case is by using cooper tubes as media to reduce the temperature of the QIE cards. Applications examples such as: ceiling floors, geothermal heating loops, oil pipelines insulation, cold plates, heat pipes for desktop show the versatility of the topic.

It is looking forward to setting a new layout or pattern of the tube on the floor wall of the RBx, in order to optimize the heat dissipation. Then, three types of patterns proposals review from cooling systems used in similar applications are established:

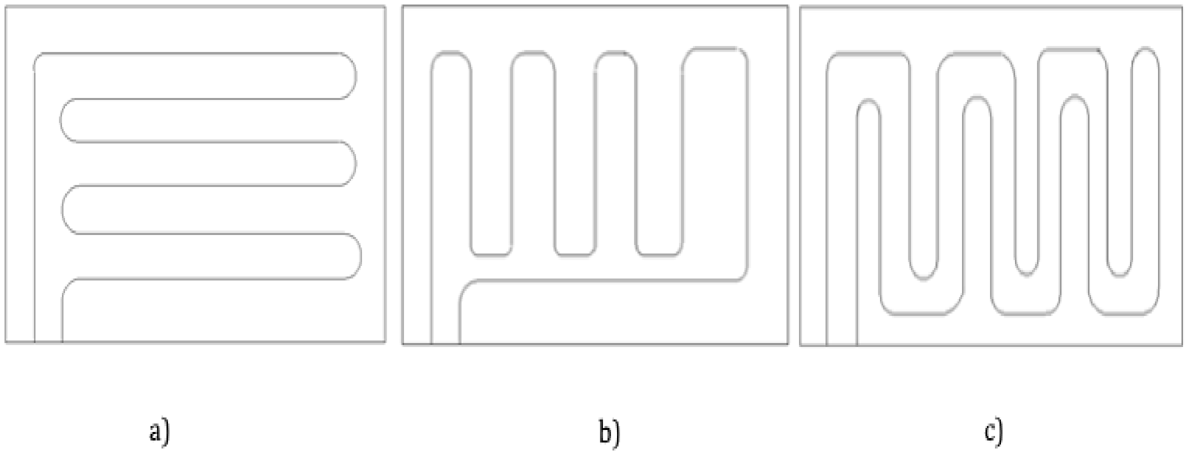


Figure 22. Pattern proposals for tube floor. a) Proposal 1. b) Proposal 2 (CAE ASSOCIATES, 2009). c) Proposal 3 (Yang, 2009)

It is establish from most of the pumbling handbooks [11]and technical cooper guides [12]that the variables for the design are: outside diameter, bending radius (BR), degree of bend (DEB), machine tooling, material, distance between bend tangent and end of tube (DBE) and distance between bends (DBB). For a better comprehension of the physical representation of the variables, an illustrative figure is show below.

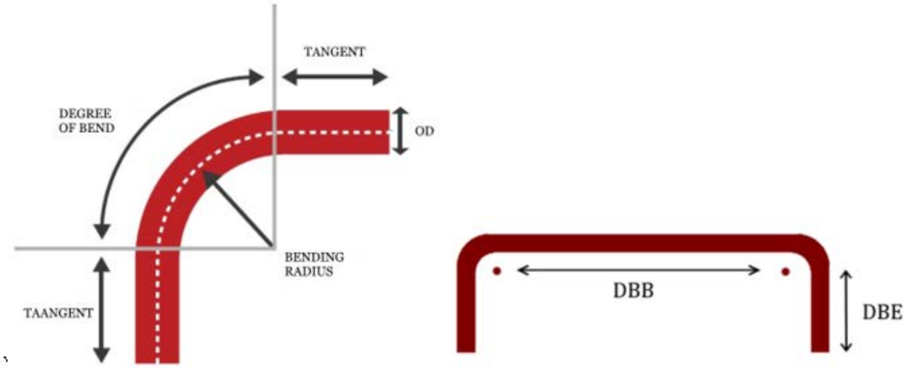


Figure 23. Tube variables description

[11] Copper tube handbook (Copper Development Association, 2017)
[12] Technical Guide (YORKSHIRE, 2014)

Most of the variables are detail in the figure, except for the machine tooling and the material. The material defines the properties of the tube and the easiness to bend it, therefore the machine tooling use for the operation. There are different bending processes, but the more often use for bending cooper are: compression bending, draw bending and roll bending. Either the process applied, the machine tooling defines the bend radius for the tube. From plumbing and mechanical codes, the bend radius must be a multiple of the OD. Further, a hard material establishes good resistance properties; however, it encounters difficulty to bend and is more costly than a soft material. A typical soft material use for tubes is cooper. Cooper is a good material due that is ductile and flexible, which makes it easier to bend; then less spring back appear when it bends. Nevertheless, because of its flexibility sometimes it is made tight bends for tubes below ¾ in of the OD. In bending field, there is a factor that is used to indicate how well the tube reacts to the bend, known as the wall factor. Is simply computed as: OD/t. If is higher than 15, considerations must be done to the tube diameter. Then, it is looking for not to use small bending diameters less than 2*OD, due that it causes wrinkling's which may lead to collapse either on outside or inside of the tube. An important point to review is that when a tube is bend, the outside of it tends to stretch and the inside to compress, meaning less material waste on the outside.

For an optimum design, the following variables are specified. The material selected is copper and works with a small diameter tube, so then, a suitable machine tooling is a lever bender. The level bender defines the bending radius as said before, and its specification is detail in the Appendix C for a 3/16 in of OD. However, to be consistent with a nominal standard size it is defined a tube bending range from 2.5 to 3.5 of the OD. Also, for U bends the DBB is not specified due to the machine tooling use for the operation and the DBE must be equal to one

OD. In the case of the layout design, the degree of bend for all the bends is 90 degrees. Possible modifications be done if the wall factor goes beyond the limits

A methodology is implemented to obtain the optimum tube layout. It considers the space to occupy and the design variables. For the first pattern proposal is detail explicitly, but for the rest of proposals just the design layout is shown.

Design

In the design, it is wanted to increase the number of loops, so the length tube increases as well. The methodology is to setup parameters for the segments of the layout, then establish a program that uses the variables to compute and find out the optimum design. The optimum design must fulfil all the design variables and is the longest one.

It has been established three types of variables: general, fill and modified. The general features are measurements obtained from CERN repository cads and they are preserved for all the designs. The modified variables are those that are used to modify the number of loops (n). And finally the fill variables are used to fill measures of the layout. In each proposal the modified and fill variables are different, due that each layout is unique. In addition, in order to obtain the real length of the tube layout, extra variables are defined. Its only function is to help determining the total length (L_T) of the tube.

The specification of the three types of variables is listed below.

Table 8. Specification and values for the general variables of the design proposals

General variables	Specification	Value (m)
H	accounts for the max height of the design space for the layout	0.304
W	accounts for the max width of the design space for the layout	0.965
W	separation between the starting point and the end point of the layout	0.018
O	separation between the limit and the starting point of the layout	0.0127
H	Tube length fill end	h = OD 0.0047625
A	separation between the layout and the limits	It is established that a = O 0.0127
C	separation between the layout and the back-side end. Is the sum of parameter b and a. The parameter b accounts for a geometry interference.	b = 0.0144 c = b+a

Table 9. Detail of the fill and modified variables of the design proposals

Design proposal	Type of variable		
	Fill	Modified	Extra
1	L1, L2	L3, L4	C1[13]
2	L1, L4, L5	L2, L3	
3	L1, L5, L6	L2, L3, L4	

Due to the complexity of not knowing the tube bending optimum diameter exactly. It is established a user-friendly program by Octave Online Editor, which use the variables of the figure and just vary the number of loops with the modified parameters. This optimum bending

[13] defines a single pipe bend (curve), not all the U shape

diameter is looking forward for not to be the min or max limits of the bending diameter degree, due that a less diameter involves higher effects of corrosion and dirtiness, and a larger diameter is not ideal for optimize the use of available space. Therefore, this diameter in addition has to be one which satisfies a long length, not too long or short, besides the conditions that has to fulfil. Then, it is correct to select the bending radius that the machine tooling specified. The purpose is then to obtain maximum number of loops for it. The scripts can be check out in Appendix C. It is established then for each proposal layout. Figures 24-26 show the different distribution of the cooper pipes obtained.

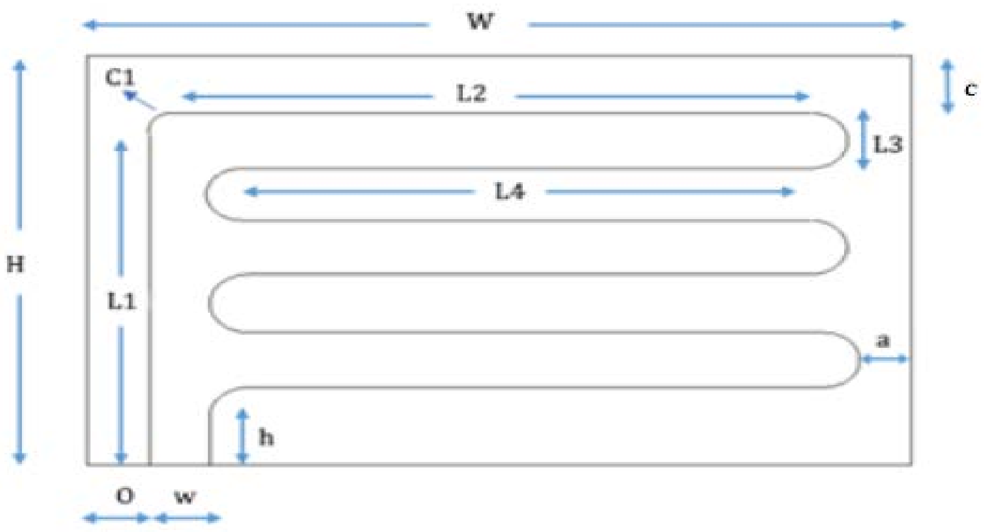


Figure 24. Layout proposal design 1

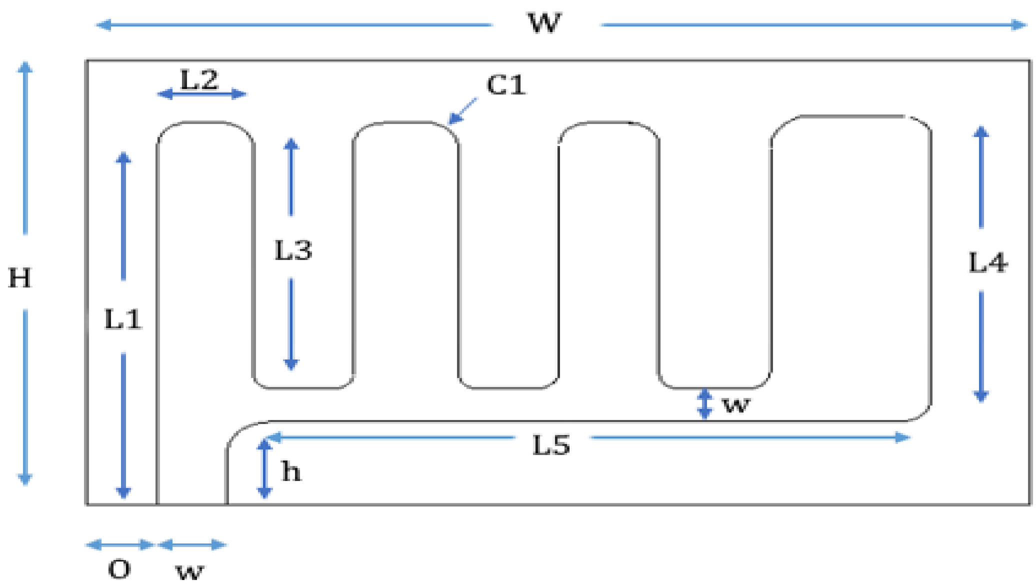


Figure 25. Layout proposal design 2

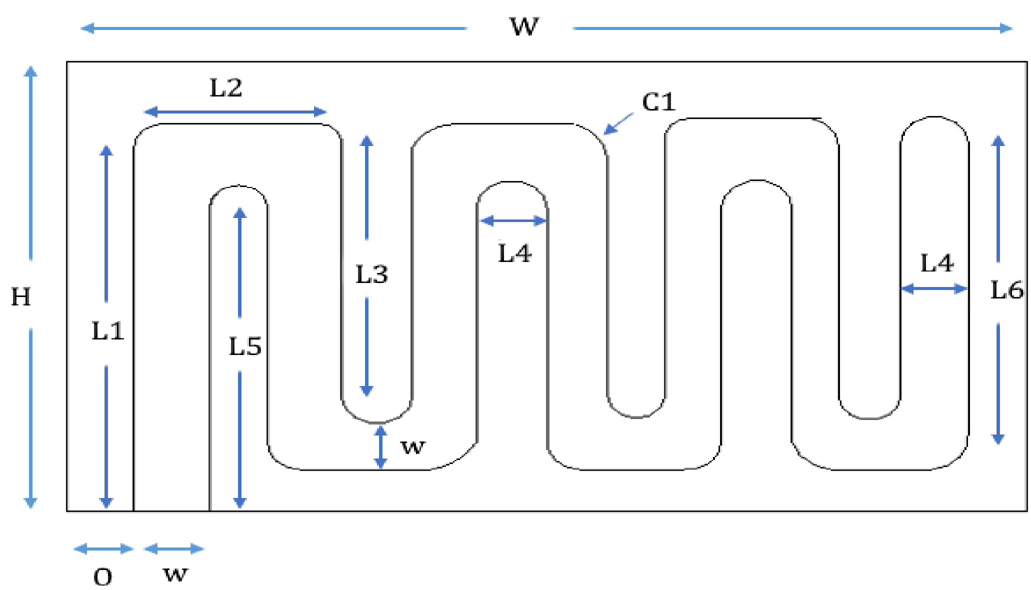


Figure 26. Layout proposal design 3

2.2.2.2 Comsol Multiphysics System Setup

Considering the tube design layout proposals, the following model for the RBx is draw for the Comsol simulations.

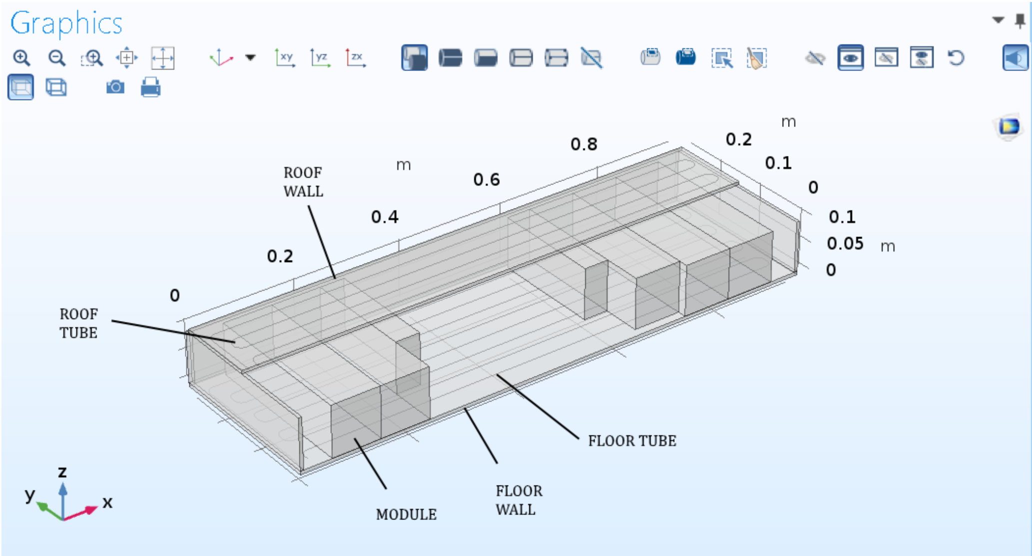


Figure 27. Comsol model

Which is mainly composed by:

Walls: corresponds to the roof, floor, sides and back wall of the RBx.

Modules: they are represented as boxes. This simplification is done, due to the lack of the present design of the CADs of the modules.

Tubes: corresponds to the roof and floor tubes. In the floor part is inserted the design layout proposal tube.

Some details of the present RBx have been removed to avoid warning mesh. For example: the floor wall has a cutoff in the entrance of the tube and also in the back part of it that contact the back wall has a cutoff to link it up to the back wall, in order to fulfil the height measurement the RBx has. However, the simplifications done are not relevant to the physics study.

The approach that lies towards the simulation is to recreate a similar environment as the real one. Therefore, an approximation of the environment is done by defining a simulation process arranged in two parts. The first one is to heat the system from time 0 [s] to 30[min] setting the initial value as the ambient temperature. Then, the temperature at 30 [min] is use as the initial condition for the following part. The second part cools down the system for 5 [min] and in consequence another distribution temperature exists in the system. Both parts in Comsol are set as transient studies. Further, the selection of the most appropriate cooling proposal tube layout is the one which lowers the system temperature more.

The detail of the steps set up in Comsol Multiphysics for the model are show below. In the physics and follow steps, it is established a sequence with numbers to provide a guide of how each node is define.

Geometry

The model is draw in Comsol using the following tools.

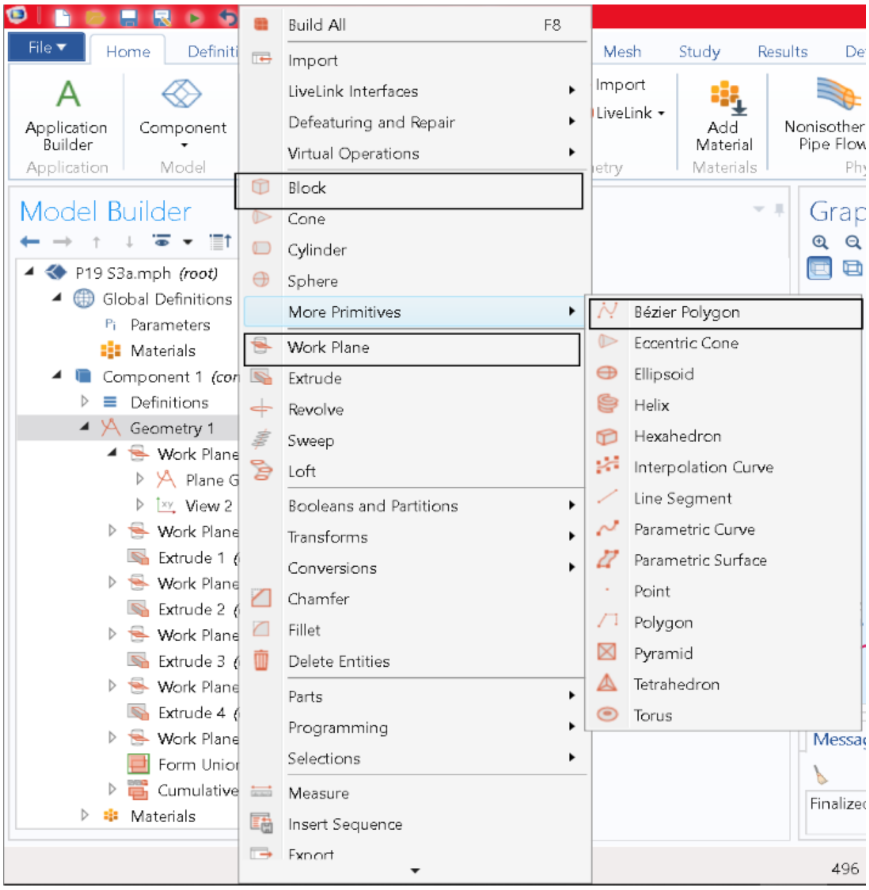


Figure 28. Geometry setup for Comsol model

Block: is use to create the walls and the modules.

Work plane: is use to create independent parts, but are related to a common plane.

Bezier polygon: is use to create the 1D curves for the tubes.

The model is design to only insert a new tube layout proposal and keep everything the same. So as not to have draw every time the layout change.

Material

Once the model is constructed, the next step is to specify the materials. To do this, the material is created or selected from the material´s library. In this case there are two materials, water for the tubes and Aluminium 6061 for the rest of the model. The Aluminium 6061 is not a material defined in Comsol, then is created using the blank material option. The setup of the materials is show in the figure.

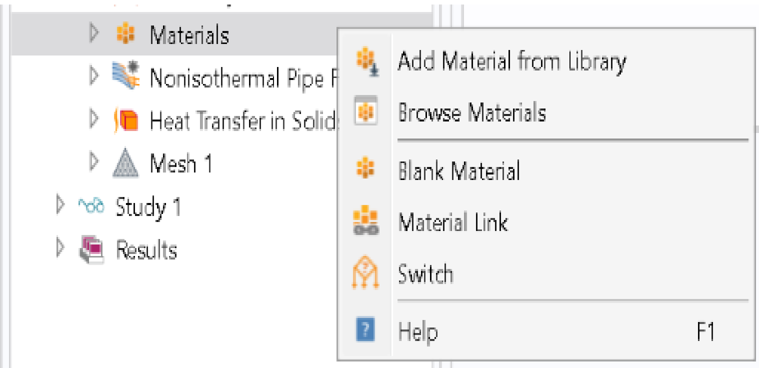


Figure 29. Material setup for Comsol model

Physics

The features for the nodes and properties either for the heat transfer in solids and the non isothermal pipe flow module are:

Heat Transfer in solids module setup

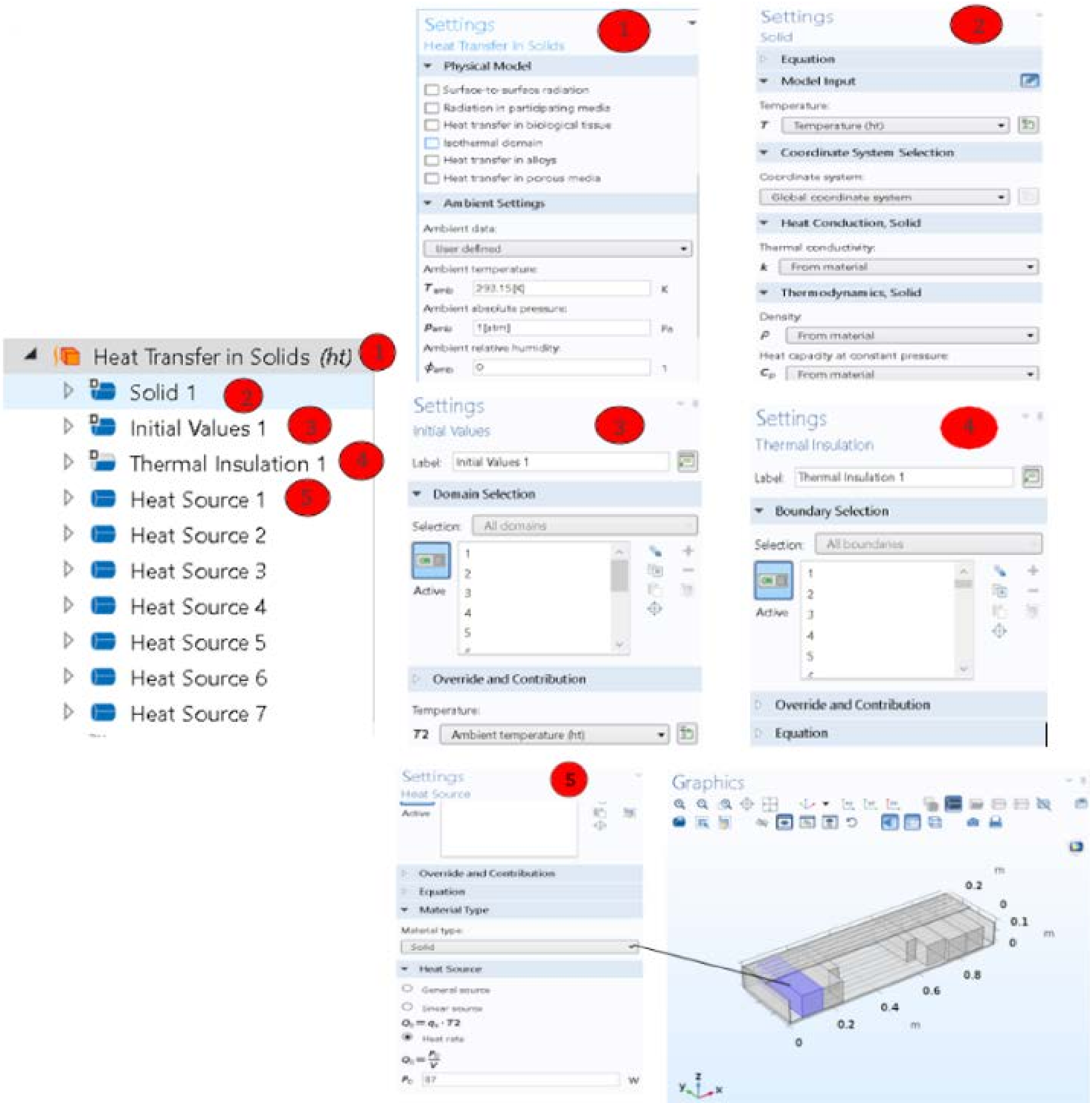


Figure 30. Heat transfer in solids module setup for Comsol model

In the heat transfer in solids node #1 is set as default. The solid node #2 specifies the RBx as solid with the temperature (T_{ht}) and link the properties defined in the materials section. In the initial values node # 3, it is specified the ambient temperature set up in node #1. This corresponds to the initial condition of the system at time 0 [s]. The thermal insulation is set as default. And at last, the heat source node #5 specifies the heat load per module according to figure 16.

Non isothermal pipe flow module setup

Due to the coupling of heat transfer and fluids in the pipe flow module. Is has been establish a section for each physics detail.

Fluid section

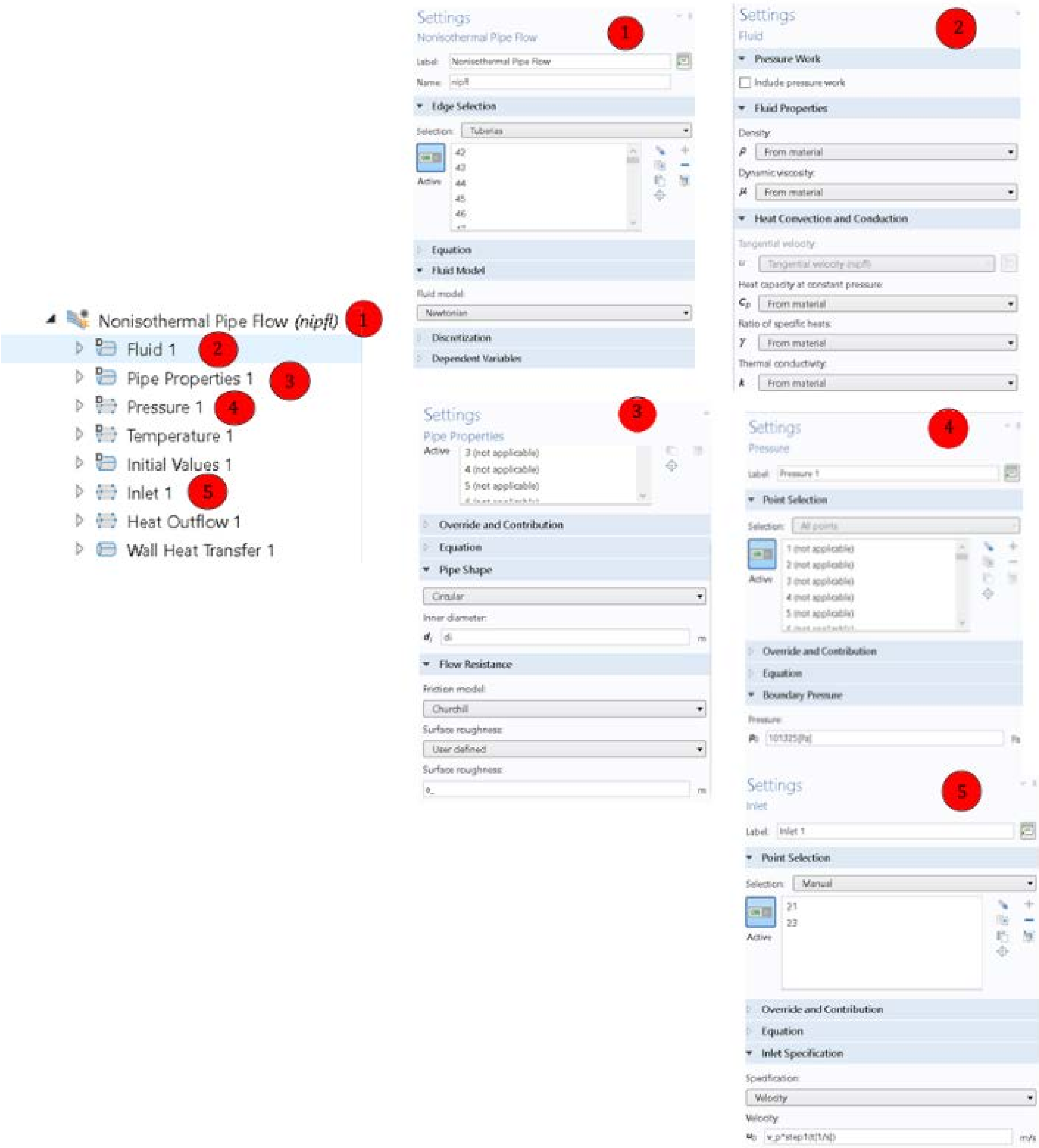


Figure 31. Non isothermal module fluid flow setup for Comsol model

Heat transfer section

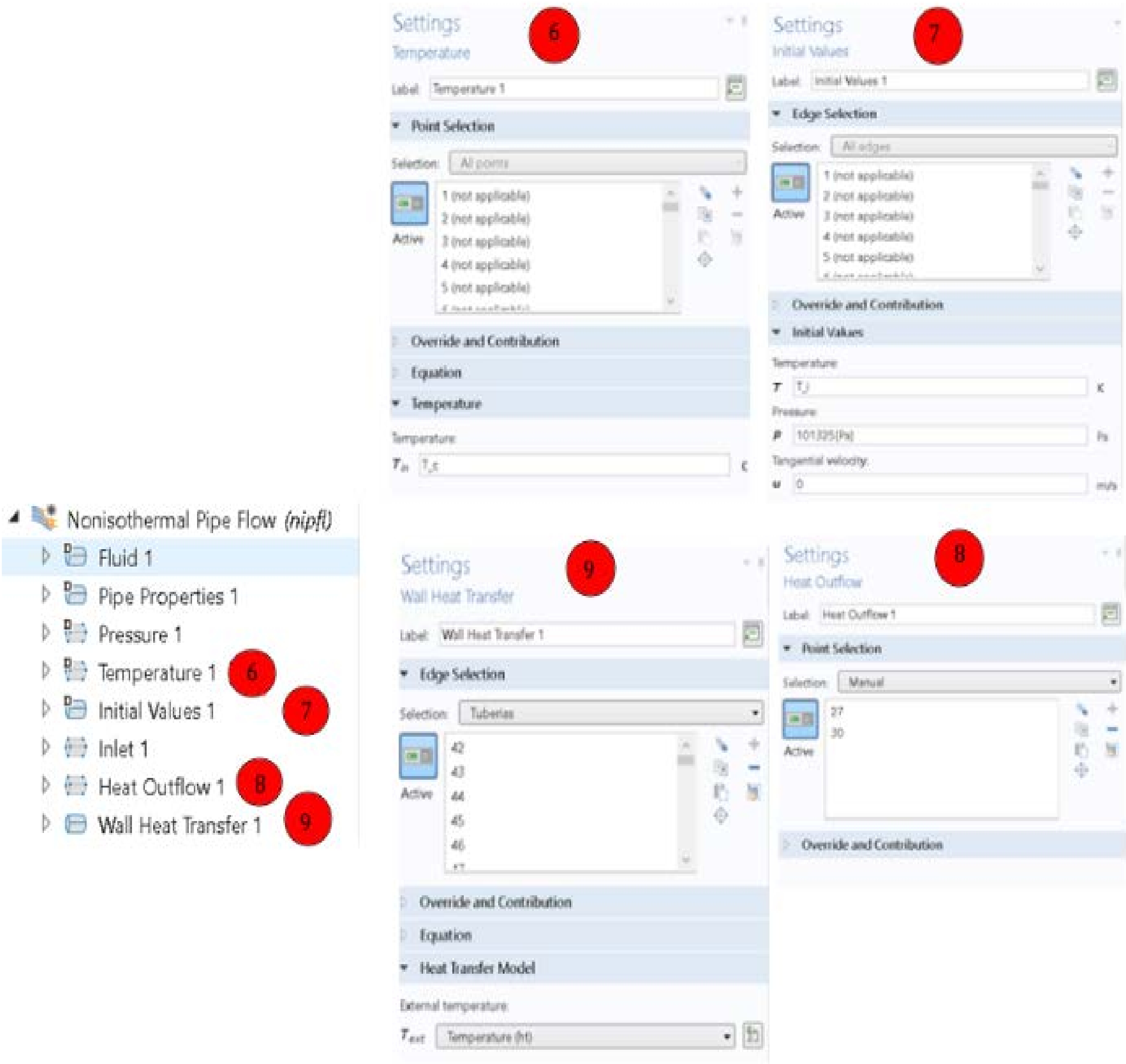


Figure 32. Non isothermal module heat transfer setup for Comsol model

Mesh setup

The mesh is user defined, where is divide by the elements section: edge for the 1D tube curves and free tetrahedral for the rest of the geometry. It is set as follows:

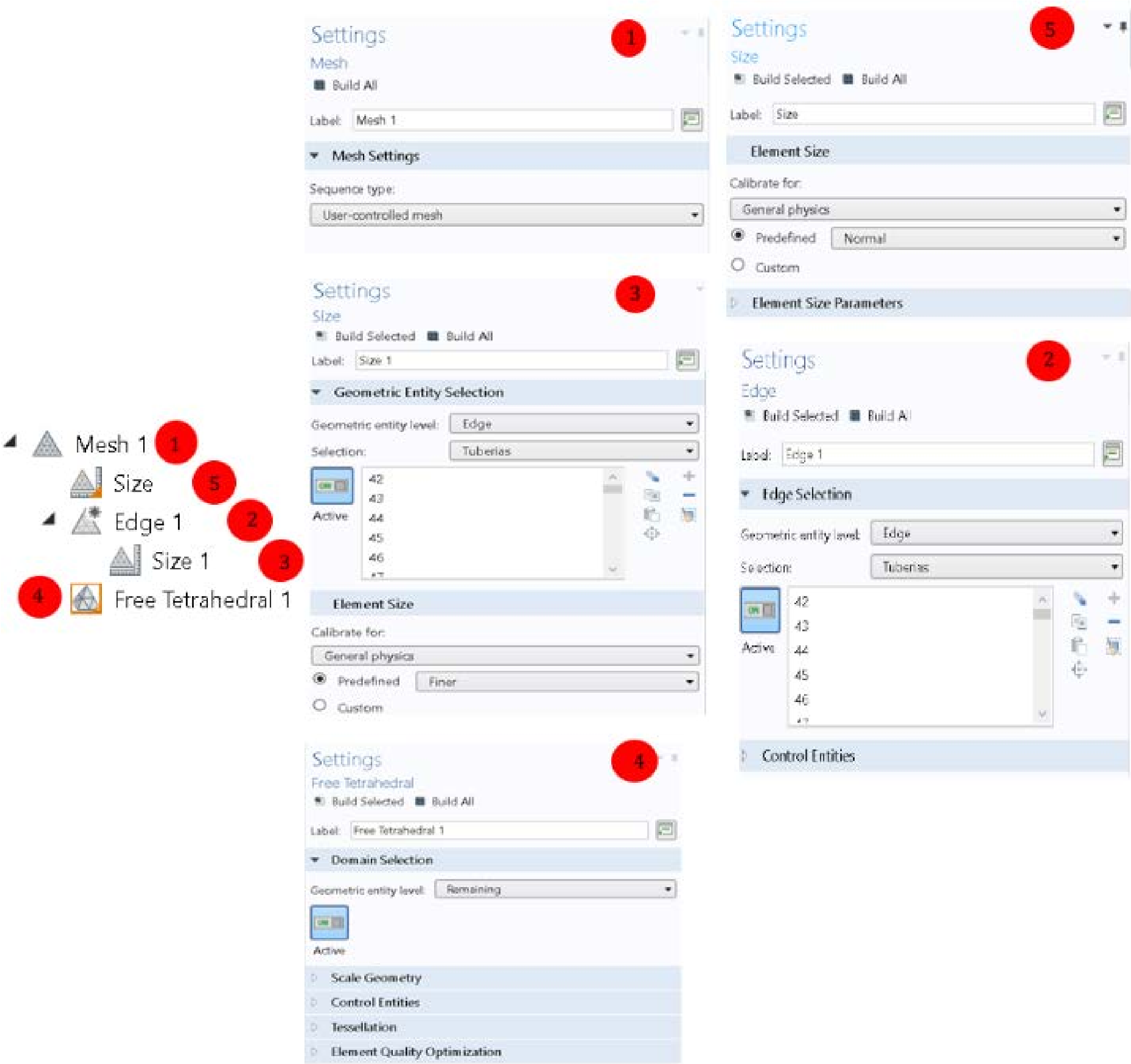


Figure 33. Mesh setup for Comsol model

Study setup

In the study section, is composed of two steps. The first step considers the heat transient study without the presence of the working fluid. The second step activates the cooling system [check box] and uses the solution of the first step [values of dependent section].

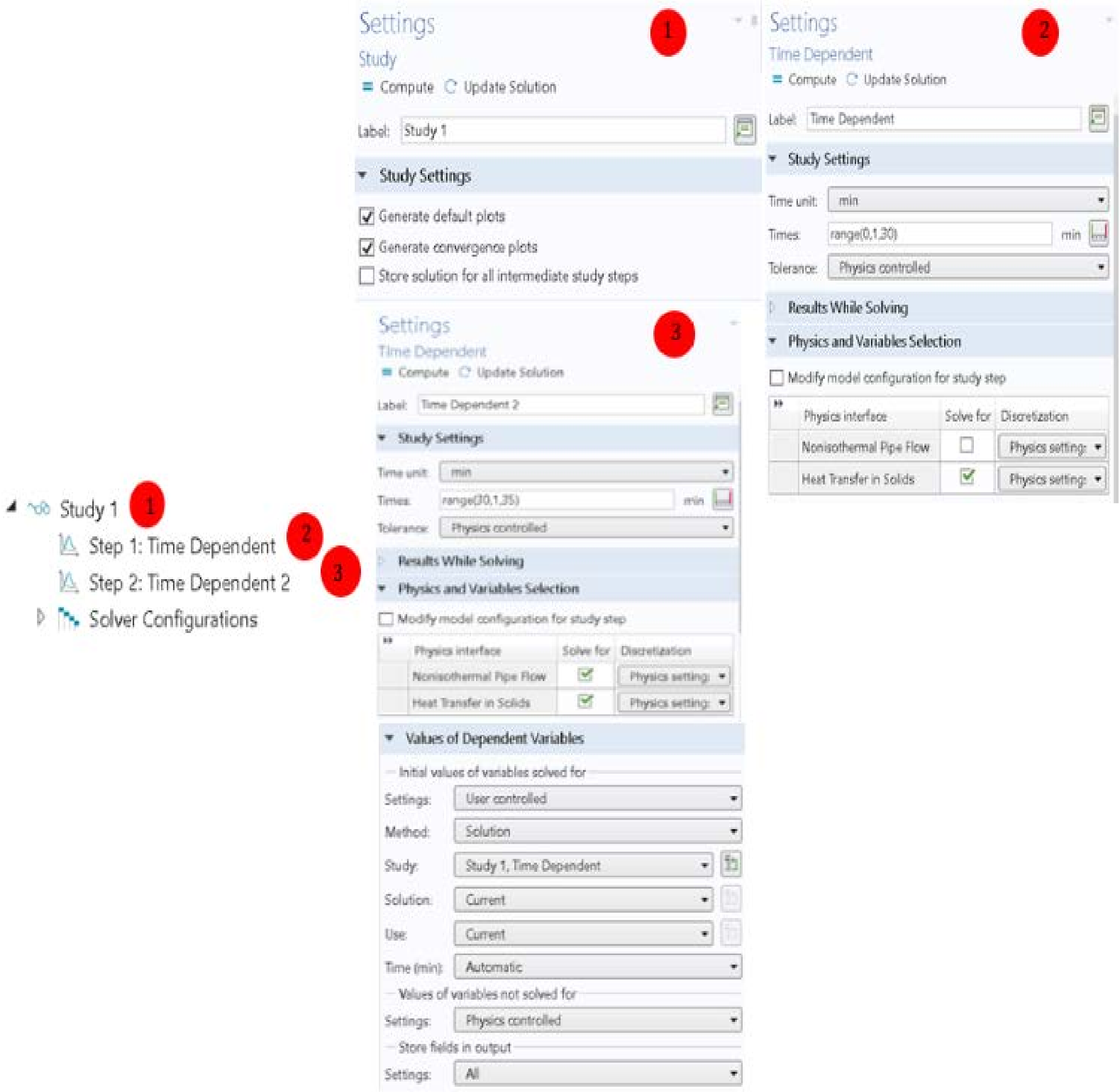


Figure 34. Study setup for Comsol model

In this way the system is setup for the analysis.

3. RESULTS AND DISCUSSION

3.1 Mathematical Results

3.1.1 Temperature distribution.

In this section is computed the function of the temperature difference in terms of the tube length. The calculations for this section and following are computed in Appendix D. Then, under the guide of the mathematical model the fluid properties are computed and the function, as shown:

Table 10. Fluid properties results

Variable	Value
T_{out}	23.5 [°C]
T_m	20.75 [°C]
$\dot{m}$	0.0167 [kg/s]
Re_D	6637.3
f	0.035
Nu	52.89
H	10065.6 [W/(m^2K]

Which lead to the result function:

$$\Delta T_{UA} = q * (2.134E - 4 + \frac{9.965E - 3}{L})$$

To a better comprehension of the function, it is set range of L from 1[m] to 10 [m] in a figure.

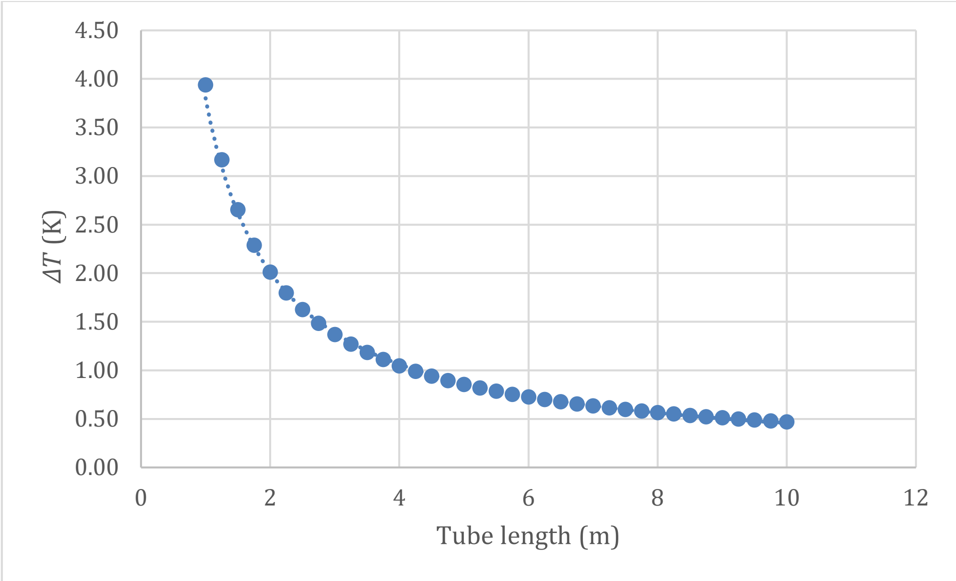


Figure 35. Temperature function graph

This figure detail how as the length of the tube increases, the difference of temperature between the module and the fluid decreases. This mean a better heat transfer dissipation, due that the fact that a low temperature difference implicates a low thermal resistance, which makes easier the heat to flow. At low measure of length, an approximated delta of 4 deg C is obtained in comparison to a 0.50 deg C delta at 10 meters.

Moreover, to show the significance that the length has, a relationship between the cooling area and the surface area of the RBx floor is defined, which is computed as: $r_A = \frac{A_{TOT}}{A_S}$. A figure is presented as shown below:

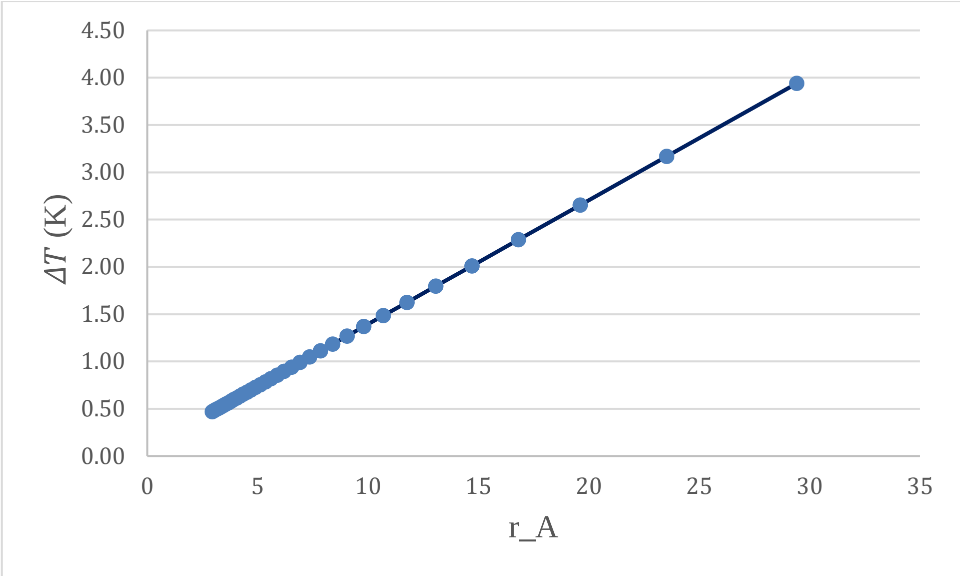


Figure 36. Relationship between cooling area and temperature

It details that a lower temperature difference is obtained by a low relationship proportion of the r_A, meaning that the cooling area is bigger, and more heat can be dissipated, justify it by Newtown's law.

3.1.2 Pipe friction losses.

In this section is computed the pressure losses that the fluid experiences. It is focus on how major losses and minor losses affect it, to demonstrate a figure of delta pressure is graph.

Hence, it is obtained a mean velocity of $\left(v = 2.11 \left[\frac{m}{s}\right]\right)$ and a bending k factor due to a 90-degree bend in a turbulent flow of $(K_B = 0.323)$. Considering this results and equation of Bernouilli, the given function of pressure difference in terms of length is obtained.

$$\Delta p = [0.227(\frac{0.035 * L}{3.175E - 3} + n * 0.323)](9810)$$

The function is graph by setting different number of bends (n) in the range of 1-80 bends and a length range from 1-10 [m].

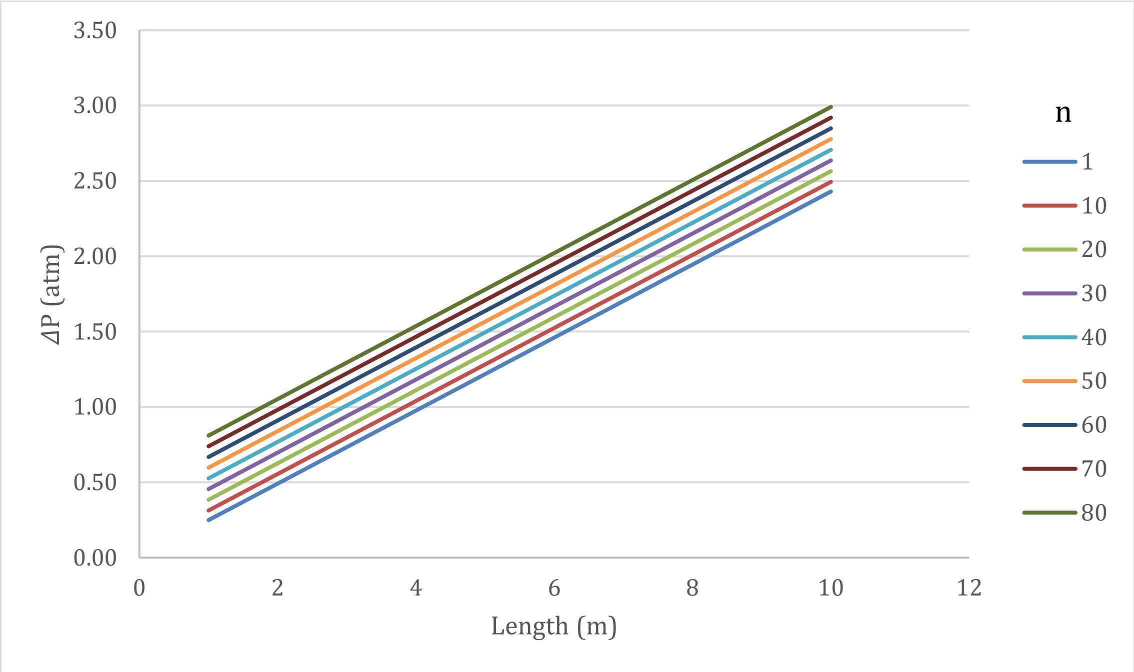


Figure 37. Pressure change in function of length and number of lengths

It is show that in a general overview, the major losses have a significance value more than the minor losses. The effect of the bend is considerable but not as the friction between the pipe and the fluid. A maximum change of pressure from 1 bend to 80 bends considering at 1-10 [m], gives approximate 0.60 (atm), however, the change of pressure from 1-10[m] by the length of the tube gives approximate 2.18 (atm).

3.2 Tube layout results

In this section is presented the layout proposal designs results. This overview the results variables and the layout design drawings for each case in COMSOL. At last, the total length of the tube layout is presented.

Design 1

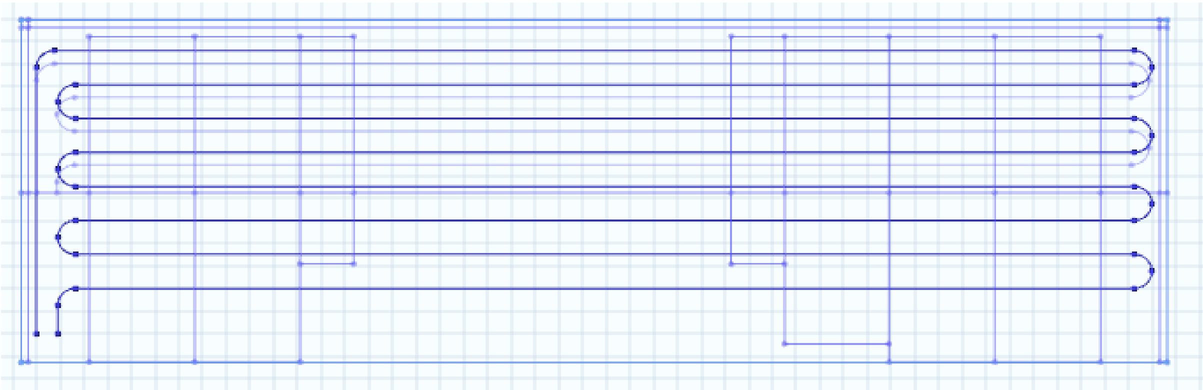


Figure 38. Proposal design 1 result

Design 2

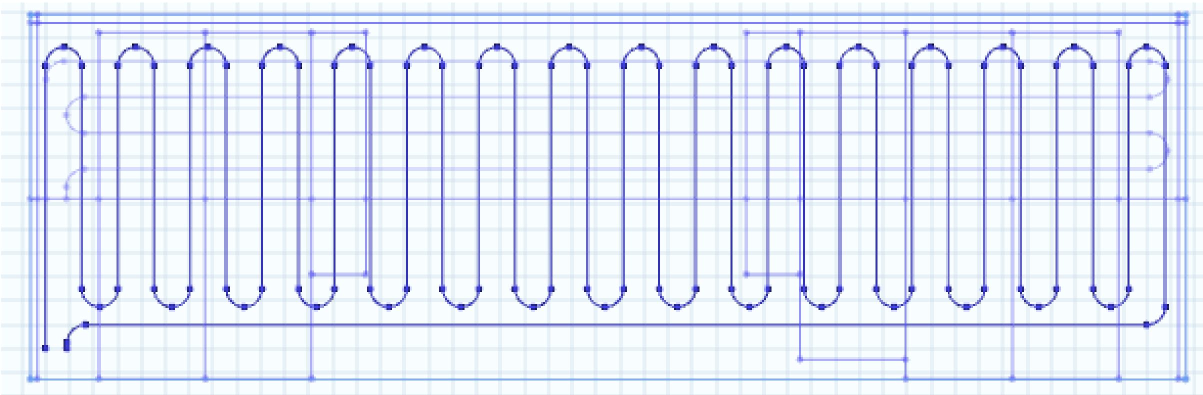


Figure 39. Proposal design 2 result

Design 3

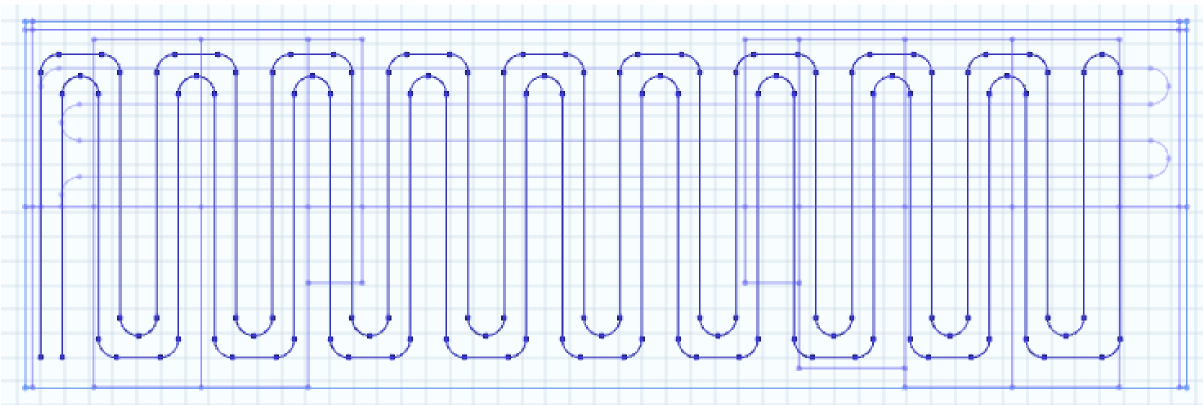


Figure 40. Proposal design 3 result

Table 11. Length per design

Design	Length (m)
1	7.79
2	8.43
3	10.21

As shown, despite having the same bend diameter, it is obtained that the type of distribution enables to obtain more or less tube length.

3.3 Simulation Results

This section is divided in two parts: qualitative results and quantitative results.

3.3.1 Qualitative results.

The qualitative results overview the results from the initial condition of the system and present design temperature distribution. Moreover, in order to compare results, the temperature and heat flux distribution is shown, the floor distribution and a selected module for each design proposal.

The initial condition of the system before the activation of the cooling system, is show below:

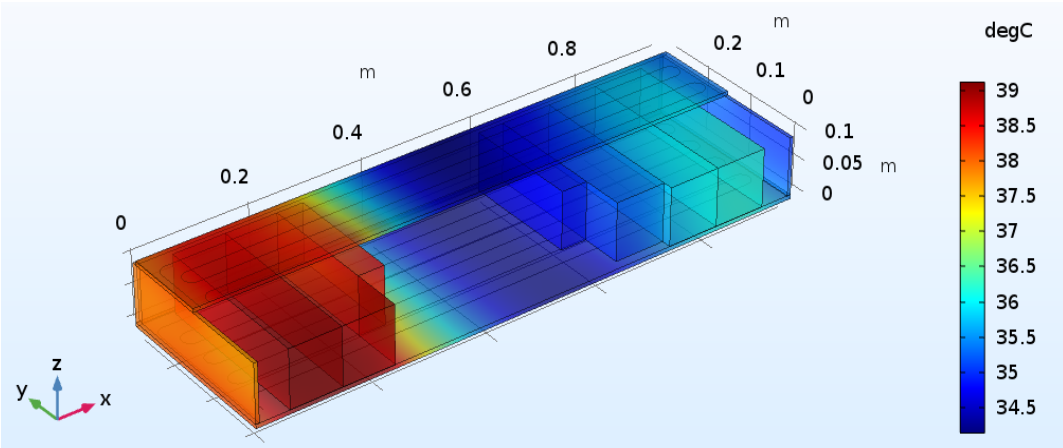


Figure 41. Initial temperature distribution of the system at t = 30 [min]

At first glance, the temperature distribution in the RBx is not uniform due that the heat generation in the left side is higher than the right side. So then, a higher temperature is expected where more heat is generated.

In general, the initial condition is set-up for all the following simulations. In the case for the present design the temperature distribution in the RBx is obtained:

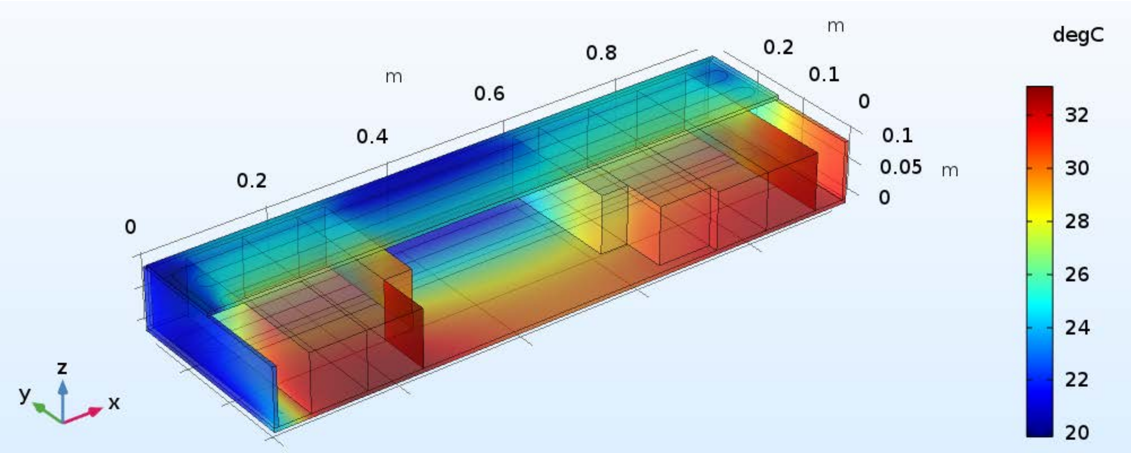
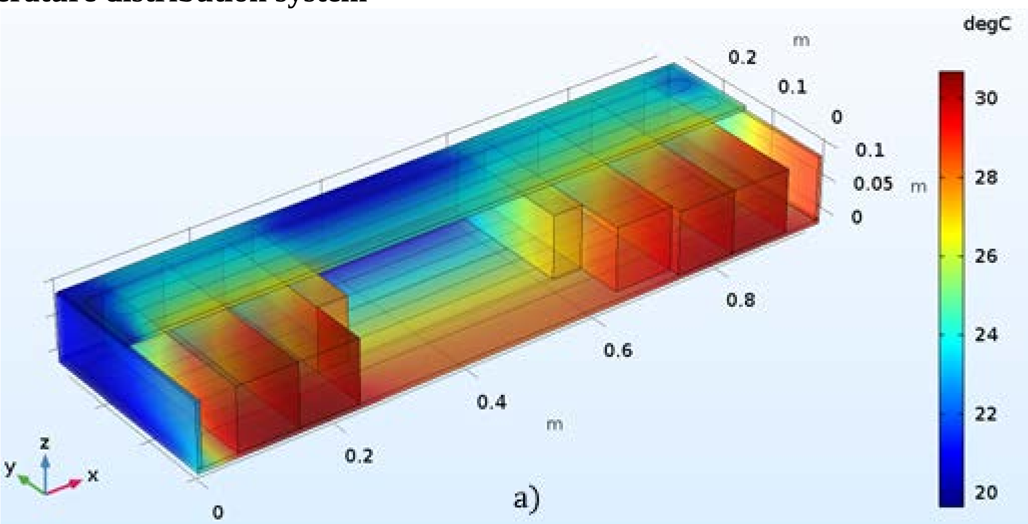


Figure 42. Temperature distributon system of the present design at t = 35 [min]

The temperature distribution shows that the hot spots are due to areas that are not in contact with the cooling system and further it lacks the section that is cut off from the roof, which may be a section to cool off the system. On the other hand, cold spots are due to areas that don't generate heat, so they maintain cold.

Temperature distribution system



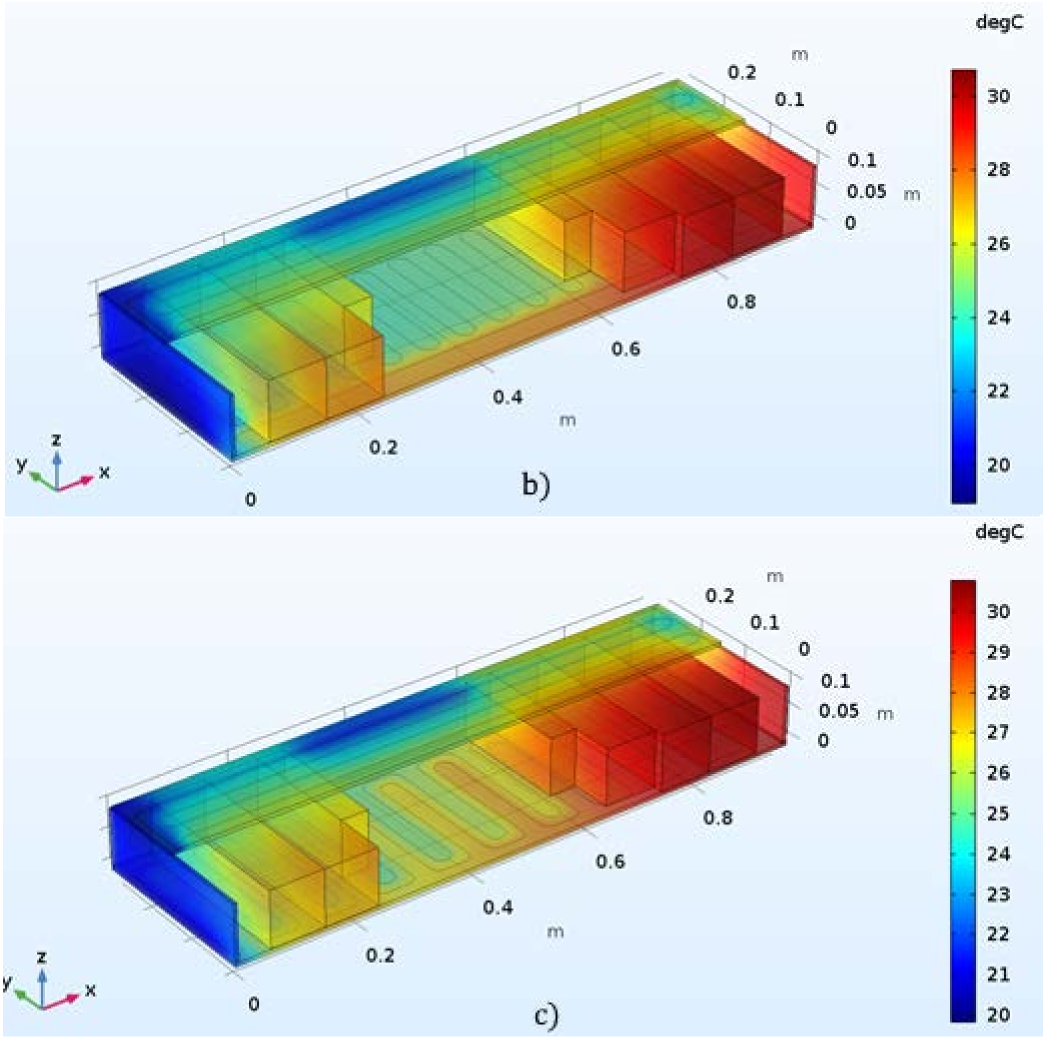


Figure 43. Temperature distribution system at t = 35 [min] a). Design 1 b) Design 2. c) Design 3.

In this figure it can be seen that the distribution temperature is different for each proposal design. The first design has a similar distribution to the present design, because it has the same distribution but with more bends. The second design in comparison with the first one, shows lower temperatures on the left side than the right side, due to the increase of quantity of the cooling system area (length of tube), the same applied between the third design and the first. In the second and third design, the temperature is higher in the right side, because as the fluid advances it gains heat and its temperature increases, then, the heat transfer decreases.

Heat flux distribution

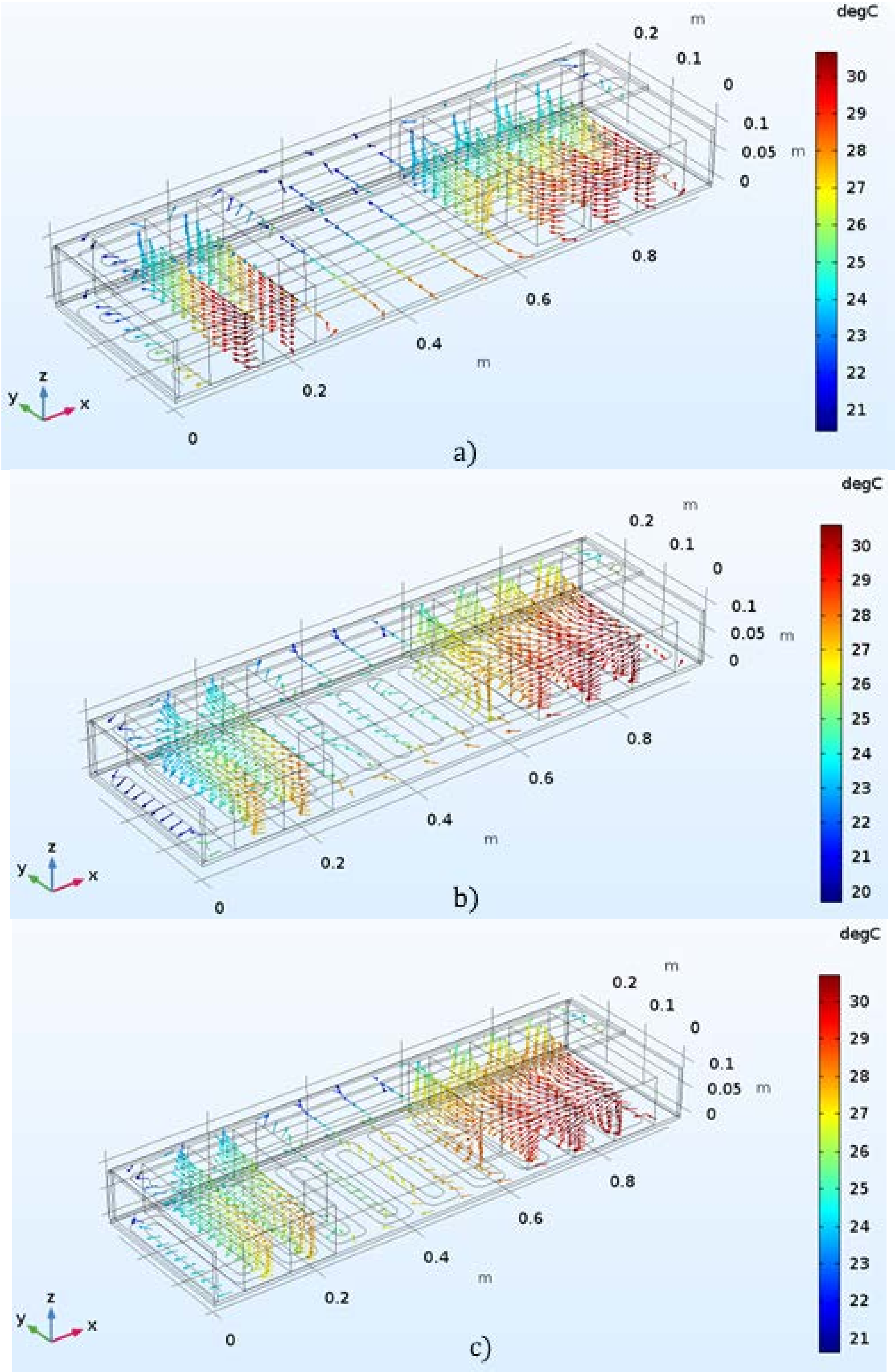
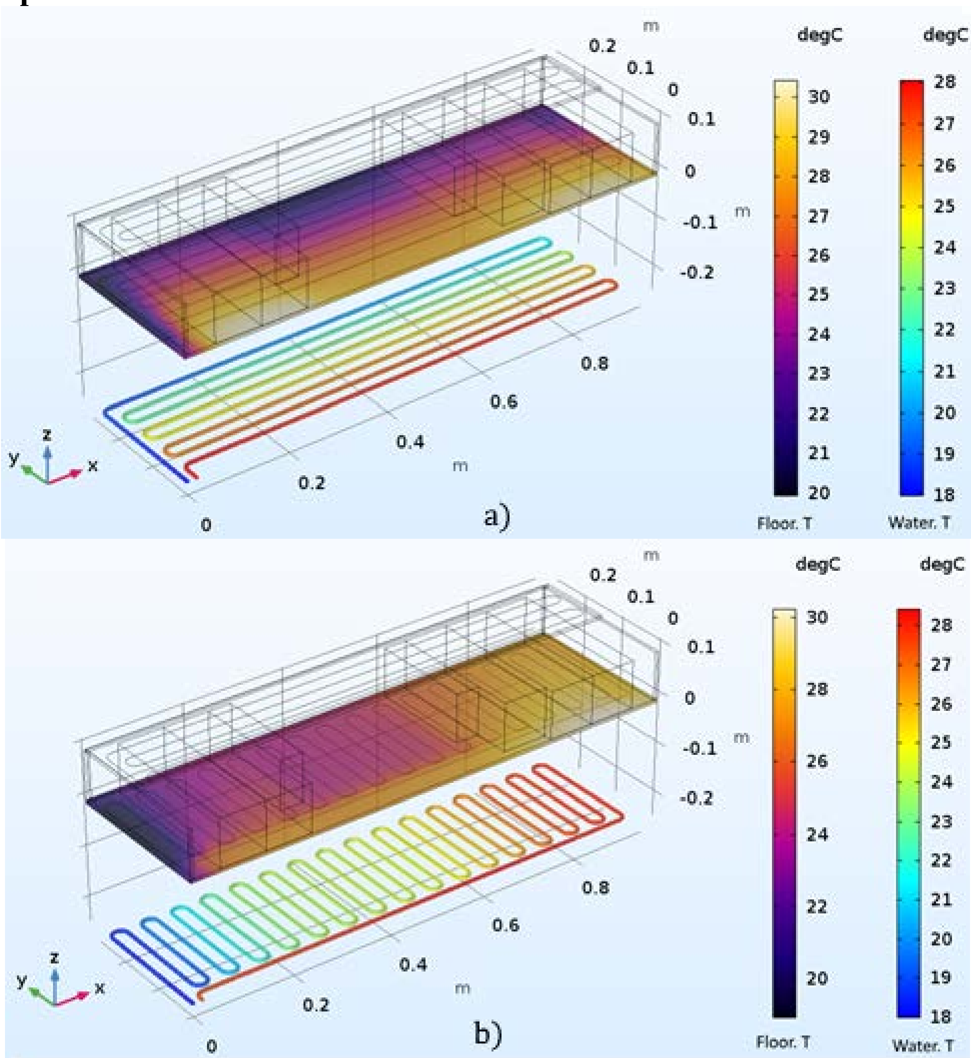


Figure 44. Heat flux distribution at t = 35[min]. a). Design 1 b) Design 2. c) Design 3.

The heat flux distribution of each design proposal is obtained; in the first design, the heat flows from the front to the back of the RBx. In the second design the heat flows more uniform to the cooling system, which means that the left side is colder, the middle section is warm and the right side is hot. Furthemore, the third design has a similar distribution heat flux as the second design, however in the middle section it gets hotter due to the distribuion of the layout.

Floor temperature distribution



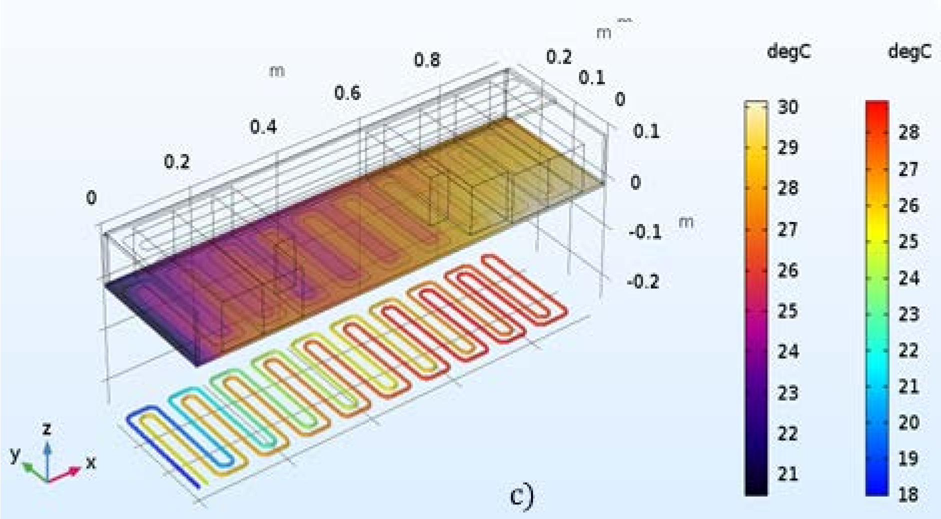


Figure 45. Floor temperature distribution at t = 35[min]. a). Design 1 b) Design 2. c) Design 3.

The floor temperature qualitative results agrees with was said in the results of the temperature distribution system for each design. To emphasize that in the third design, the tubes are closer to each other, therefore there is a heat flow between the tubes and it is justify by the "footprint temperature" that separates the tubes in a cold side and a hot side along the design. Besides that, the fluid temperature profile goes in hand with the temperature distibution on the floor in each design proposal. Also, to point out that in the third design the fluid increases its temperature to a maximum along the tube, but then decreases, due to the heat transfer with the cold tubes at the start.

Module 1 temperature distribution

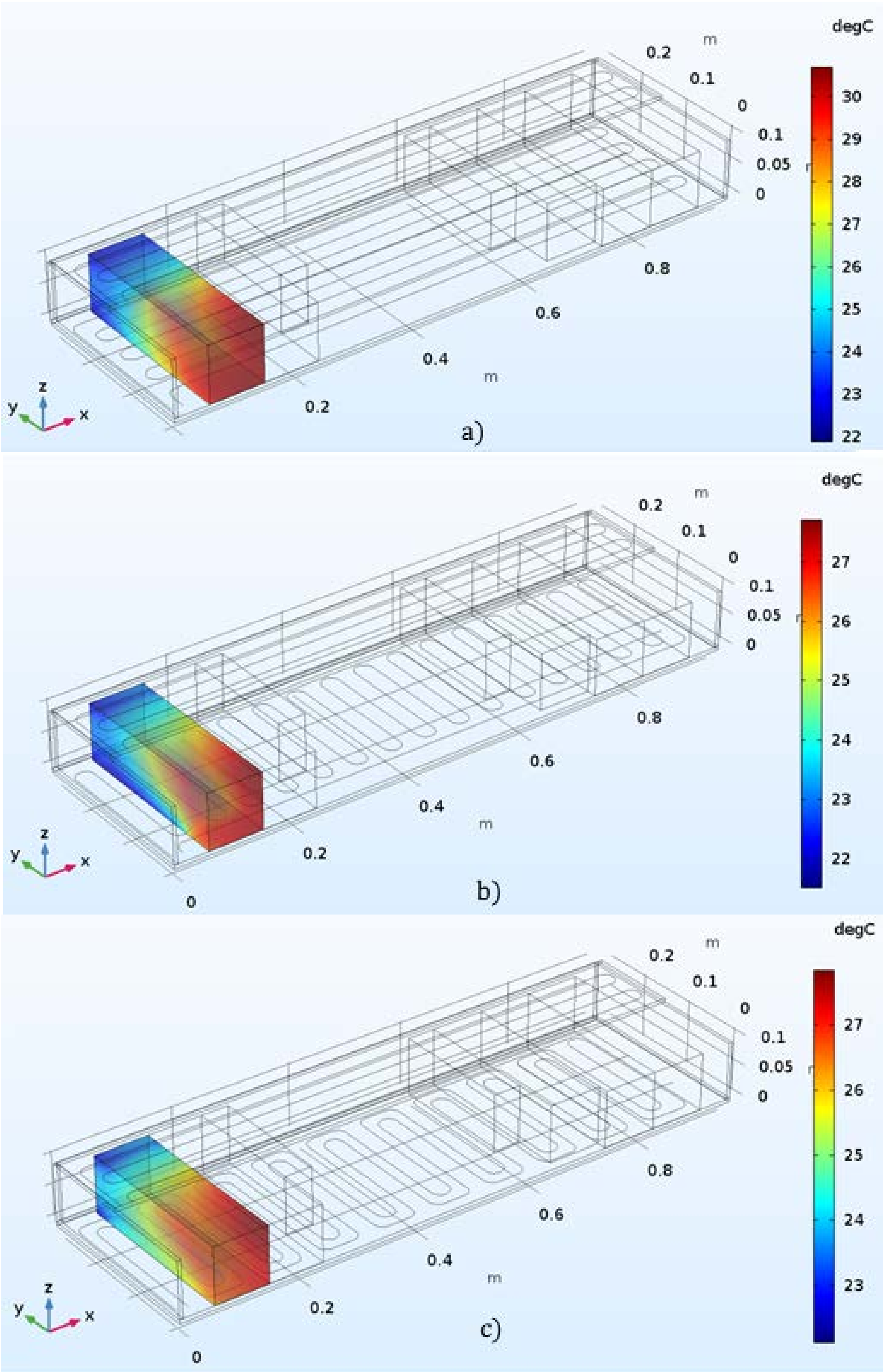


Figure 46. Module 1 temperature distribution at t = 35[min]. a). Design 1 b) Design 2. c) Design 3.

In the module 1 temperature distribution it is show a similar distribution for all the designs. A hot region in the front side and a cold region on the back side. This behaviour is due to the layout proposed, heat is more dissipated in this region. Also, it seems that at the entrance the cooling system is not acting yet as expected in consideration from the previous distribution results.

3.3.2 Quantitative Results.

The quantitative results overview the results of the temperature of water, module 1 and floor. In order to properly compare in each figure is presentend each design proposal layout and the present design.

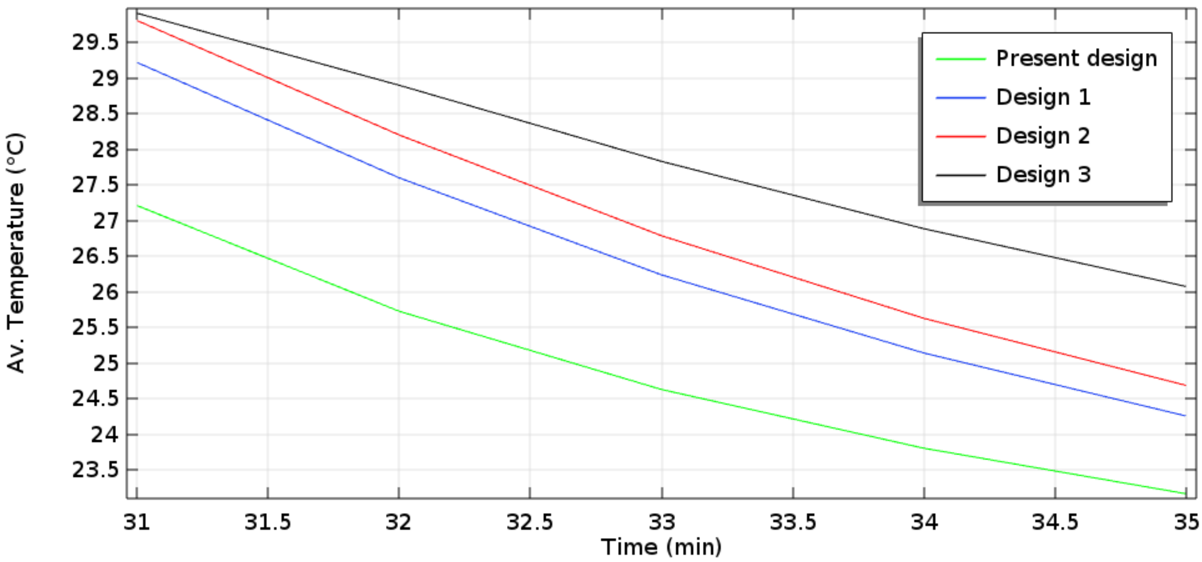


Figure 47. Transient study of water at t = 35[min].

This result shows the developmet of the temperature profile of the water. It has that development, due that starts from the temperature of the system as initial condition. Then, the water starts to look for an equilibrum state, which explain why the temperature as time passes decreases, what it has to be consider in this figure is the temperature difference for each design proposal. It is obtained that the design 1 & 2 both experiences the greater temperature difference aproximated to 5ºC, the present design shows a temperature difference 4ºC and design 3 a temperature difference of approximate 4ºC.

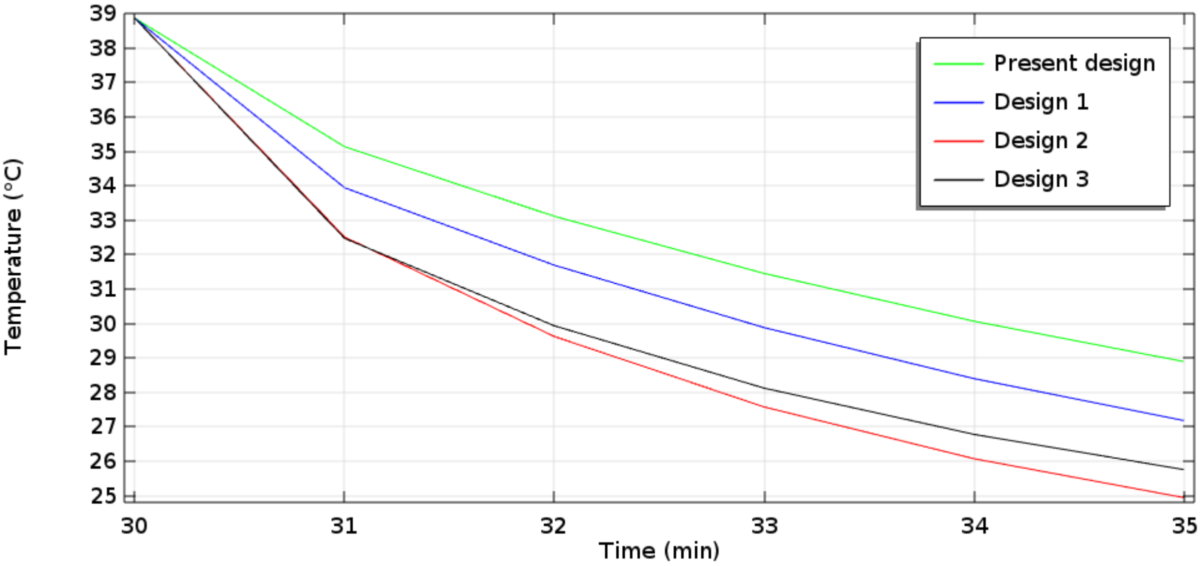


Figure 48. Transient study of module 1 from t = 30 [min] to t = 35 [min].

This figure illustrates the profile development of the module 1 temperature. The results obtained point out that the design 2 reaches lower temperatures (aprroximated 25 deg C), follow by the design 3, design 1 and present design. However, despite the fact of the similarity of the temperature distribution in the qualitative results section for the three proposed designs , it is shown that the temperature at the end is different in each case.

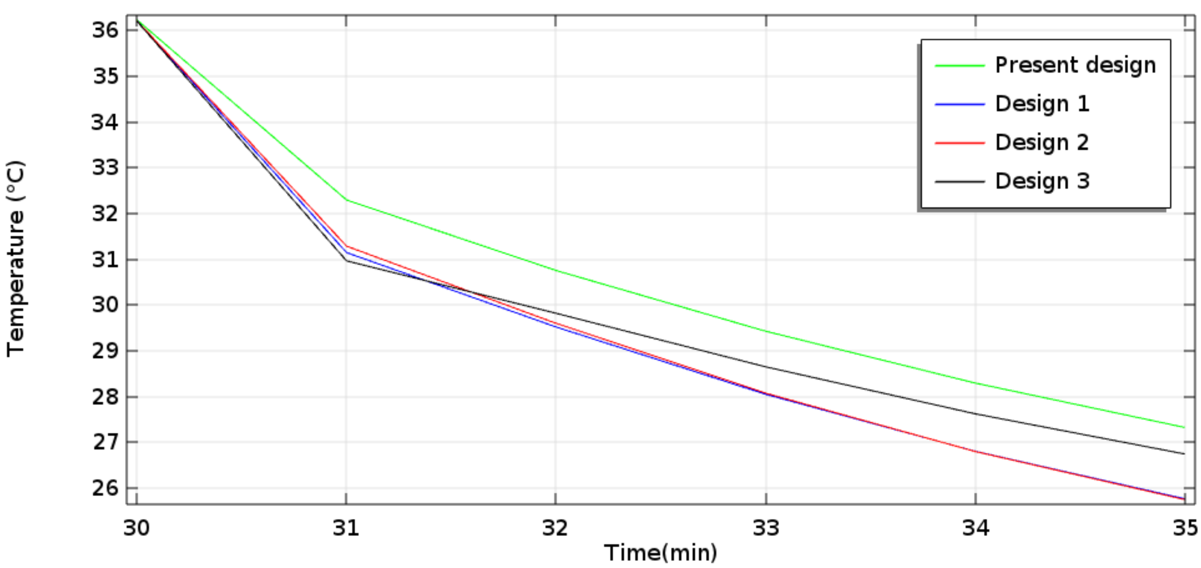


Figure 49. Transient study of floor from t = 30 [min] to t = 35 [min].

This figure illustrates the temperature profile of the floor. The results obtained point out that design 1 & 2 reaches the lower temperatures (aprroximated 26ºC both) than the present design and design 3. This corroborate with the qualitative results obtained of the temperature distribution of the floor, that both design 1 & design 2 experiences a better heat dissipation rate to the cooling system than the other layouts.

Having the results for all the proposed designs, the centralized weight method can be applied to select the correct design layout. The criteria consider for the method are those that are relevant to accomplish the features required for the cooling system. The criteria selected are:heat dissipation (HD) , construction viability (CV) , cost (CST) and fluid control (FC). The results are presented in the following table.

Table 12. Centralized weight method

	CRITERIA					
DESIGN	HD (5)	FEA (5)	CST (3)	FC (3)	∑▒+1	Result
1	4	5	3	3	16	1,6
2	5	5	2	2	15	1,5
3	4	3	1	1	9,5	0,95
					10	1

The given score for each criteria agrees with the degree of importance that the criteria has on the specifications. So, the heat dissipation and the feasibility have a higher value, due that both criteria are the main objectives of the working project than the cost and fluid control. However, that does not mean that the remaining criteria are negligible.

It is determined due to the closeness of the results from the centralized weight method, both design 1 & 2 are selected. Then, the future work will test both designs to properly select the correct one.

4. CONCLUSION AND RECOMMENDATIONS

In conclusion, a new distribution of the pipe was achieved; which is feasible to carry out for a future construction. Firstly, it is determined that the tube layout is a remarkable design factor to consider, since the length of the tube is not enough to ensure a greater heat transfer as shown on the floor temperature distribution (Figure 45) and justify it by the quantitative results from figure 49. Moreover, a simulation model was set-up in Comsol Multiphysics that allowed to simulate the operating conditions of the RBx. Then, in the simulation results the three types of proposal designs were compare , where it was obtained that the second design demostrate better perfomance as a cooling system. However, when the centralized weigth method is consider , the first design also became one of the selected options due to the results obtained. In general, the selected proposal designs accomplish to reduce the temperature of the floor of the RBx to approximate 26ºC. Nevertheless, both designs obtained greater temperature difference of water by approximated 5ºC.

As recommendations of this senior thesis, it is suggested to use the same modules of heat transfer in solids and non-isothermal pipe flow, since that avoids waste of time and inneficient results. Further, if another refrigerant want to be use, the same procedure is maintained; just change the set-up in the material section to specify the fluid type and properties. To finalyze, the implementation of the updated CADs of the modules will help to obtain more realistic results.

5. FUTURE WORK

The work done corresponds to the first part of this global project, the preliminary phase. In which the designs over the system were coupling by simulation. The next phase follows the construction of the prototype to evaluate experimentally the results, in order to validate the results from the simulation and compare them. Therefore, the following experimental setup to study the different cooper line distribution layouts is established, as shown in figure 50.

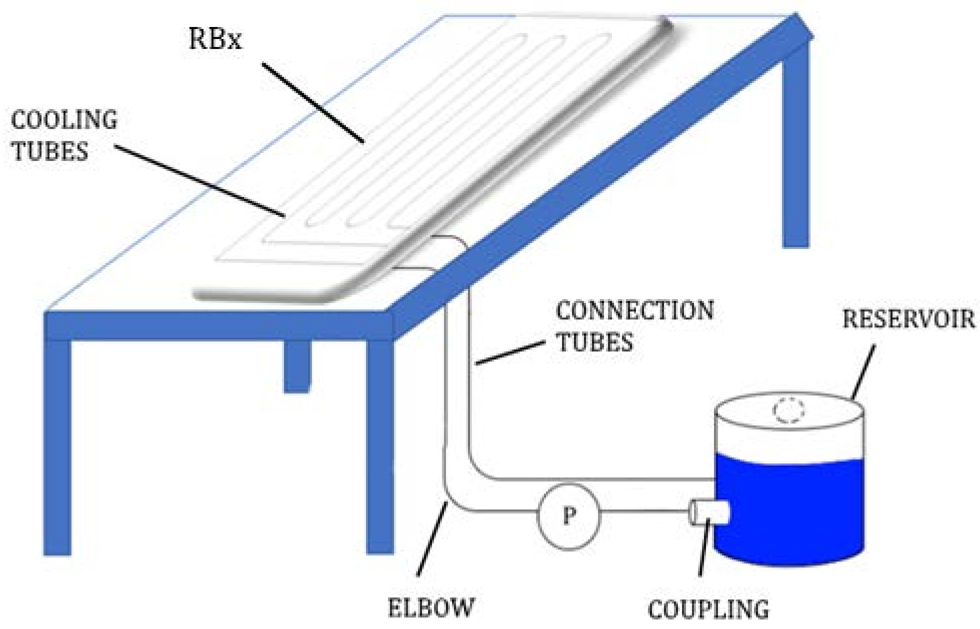


Figure 50. Experimental setup of the prototype cooling system.

Considering the previous experimental setup a projected budget for the construction is proposed; considering it will be constructed in Ecuador.

Table 13. Experimental Setup Proposed Budget

DETAIL	DESCRIPTION	COST	
Manufacture	Machine work: cutting, drilling, milling Operations: bending	Machine work	50
		Operations	30
Materials	Aluminum (plates:6 mm, 2 mm) , Copper(Tubes)	Al 6 mm	250
		Al 2mm	
		Copper	30
PCB Boards	Transformer, Bakelite, Potentiometers, cables	All	100
Cooling system	PVC (Tube connection), accessories (elbows, couplings), reservoir, pump	PVC	5
		Accessories	5
		Reservoir	5
		Pump	50
Measurement equipment	Thermocouples, Arduino	All	75
		TOTAL	$600

The pump selection is done by a fluid mechanics analysis to determine the head of the system and with it select the appropiate pump that satisfies the flow rate. Therefore, the pump selected is a vertical submersible water pump.

6. REFERENCES

CERN Geneva The LHC experiments Committee. (2017). *The Phase-2 Upgrade of the CMS Barrel Calorimeters TDR*. Retrieved from https://cds.cern.ch/record/2283187/files/CMS-TDR-015.pdf

CERN Geneva The LHC Experiments Committee. (1997). *HCAL TDR*. Retrieved from https://cds.cern.ch/record/357153/files/CMS_HCAL_TDR.pdf

INSTITUTE OF PHYSICS PUBLISHING AND SISSA. (2008). *The CMS experiment at the CERN LHC* (Vol. 3). https://doi.org/10.1088/1748-0221/3/08/S08004

Karmgard, D. (2001). RBX Cooling. Retrieved July 4, 2018, from http://physics.bu.edu/cms/hcal/RBX_PRR/prr-cooling.ppt

Mans, J. (2012). *CMS TDR for the Phase 1 Upgrade of the Hadron Calorimeter*. Retrieved from http://cds.cern.ch/record/1481837/files/CMS-TDR-010.pdf

Ruchti, R. (2001). HCAL RBX for HB HE and HO Status and Planning. Retrieved July 4, 2018, from http://physics.bu.edu/cms/hcal/RBX_PRR/RR PRR talk for 3-1-01 updated 2-24-01.ppt

Schmidt, I. (2018). HB RBX Introduction Ecuador.

Incropera, F. P. and DeWitt, D. P. (2011). *Fundamentals of Heat and Mass Transfer*. United States, New York: John Wiley and Sons , Inc

Barney, D. (2003). *An overview of the CMS experiment for CERN guides*. Retrieved from https://cds.cern.ch/record/2283187/files/CMS-TDR-015.pdf

Huc, N. (2014) .*Conjugate heat transfer*. Retrieved July 17, 2018, from https://www.comsol.com/blogs/conjugate-heat-transfer/

Wollblad, C. (2017). *CFD in Comsol Multiphysics*. Retrieved from https://www.kth.se/social/files/591470ecf276542fd65c9a1e/kth170510.pdf

COMSOL. (2014). *Comsol Multiphysics User's Guide*. Retrieved from http://people.ee.ethz.ch/~fieldcom/pps-comsol/documents/User%20Guide%20and%20Examples/COMSOLMultiphysicsUsersGuide.pdf

COMSOL. (2012). *Pipe flow module User's Guide*. Retrieved from http://people.ee.ethz.ch/~fieldcom/pps-comsol/documents/User%20Guide%20and%20Examples/COMSOLMultiphysicsUsersGuide.pdf

Copper Development Association. (2017). *Copper Tube Handbook*. Retrieved from https://www.copper.org/publications/pub_list/pdf/copper_tube_handbook.pdf

YORKSHIRE. (2014). *TECHNICAL GUIDE*. Retrieved from http://yorkshirecopper.com/wp-content/uploads/Technical-Guide-full-issue-02_14_Really2.pdf

CAE ASSOCIATES. (2009). *Heat Exchanger Cooling Analysis*. Retrieved from https://caeai.com/sites/default/files/Heat%20Exchanger%20Cooling%20Analysis.pdf

Yang, J. (2009). *Heat Transfer coefficient in flow convection of pipe cooling system in massive concrete*. Retrieved from https://www.jstage.jst.go.jp/article/jact/9/1/9_1_103/_article

Aalco. (2018). *Aluminium Alloy 6061- T6 Extrusions*. Retrieved from http://www.aalco.co.uk/datasheets/?search=al%206061%20t6

Efunda. (2007). *Absolute pipe roughness*. Retrieved from http://www.efunda.com/formulae/fluids/roughness.cfm

Munson, B. (2009). *Fundamentals of Fluid Mechanics*. United States, Jefferson City: John Wiley and Sons , Inc

RIDGID. (2013). *BENDING & FORMING*. Retrieved from http://www.ridgid.eu/catalog/EN/08_Bending_EN_WEB.pdf

White, F. (2011). *Fluids Mechanics*. United States, New York: Mc Graw Hill

CMS DocDB Administrators (2001). *List of topics*. Retrieved from https://cms-docdb.cern.ch/cgi-bin/DocDB/DocumentDatabase

7. APPENDIX

7.1 Appendix A

Table A.1. Water properties (Incropera, 2011)

Temperature, T (K)	Pressure, p (bars)^b	Specific Volume (m^3/kg) $v_f \cdot 10^3$	v_g	Heat of Vaporization, h_{fg} (kJ/kg)	Specific Heat (kJ/kg · K) $c_{p,f}$	$c_{p,g}$	Viscosity (N · s/m^2) $\mu_f \cdot 10^6$	$\mu_g \cdot 10^6$	Thermal Conductivity (W/m · K) $k_f \cdot 10^3$	$k_g \cdot 10^3$	Prandtl Number Pr_f	Pr_g	Surface Tension, $\sigma_f \cdot 10^3$ (N/m)	Expansion Coefficient, $\beta_f \cdot 10^6$ (K^{-1})	Temperature, T (K)
273.15	0.00611	1.000	206.3	2502	4.217	1.854	1750	8.02	569	18.2	12.99	0.815	75.5	−68.05	273.15
275	0.00697	1.000	181.7	2497	4.211	1.855	1652	8.09	574	18.3	12.22	0.817	75.3	−32.74	275
280	0.00990	1.000	130.4	2485	4.198	1.858	1422	8.29	582	18.6	10.26	0.825	74.8	46.04	280
285	0.01387	1.000	99.4	2473	4.189	1.861	1225	8.49	590	18.9	8.81	0.833	74.3	114.1	285
290	0.01917	1.001	69.7	2461	4.184	1.864	1080	8.69	598	19.3	7.56	0.841	73.7	174.0	290
295	0.02617	1.002	51.94	2449	4.181	1.868	959	8.89	606	19.5	6.62	0.849	72.7	227.5	295
300	0.03531	1.003	39.13	2438	4.179	1.872	855	9.09	613	19.6	5.83	0.857	71.7	276.1	300
305	0.04712	1.005	29.74	2426	4.178	1.877	769	9.29	620	20.1	5.20	0.865	70.9	320.6	305
310	0.06221	1.007	22.93	2414	4.178	1.882	695	9.49	628	20.4	4.62	0.873	70.0	361.9	310
315	0.08132	1.009	17.82	2402	4.179	1.888	631	9.69	634	20.7	4.16	0.883	69.2	400.4	315
320	0.1053	1.011	13.98	2390	4.180	1.895	577	9.89	640	21.0	3.77	0.894	68.3	436.7	320
325	0.1351	1.013	11.06	2378	4.182	1.903	528	10.09	645	21.3	3.42	0.901	67.5	471.2	325
330	0.1719	1.016	8.82	2366	4.184	1.911	489	10.29	650	21.7	3.15	0.908	66.6	504.0	330
335	0.2167	1.018	7.09	2354	4.186	1.920	453	10.49	656	22.0	2.88	0.916	65.8	535.5	335
340	0.2713	1.021	5.74	2342	4.188	1.930	420	10.69	660	22.3	2.66	0.925	64.9	566.0	340
345	0.3372	1.024	4.683	2329	4.191	1.941	389	10.89	664	22.6	2.45	0.933	64.1	595.4	345
350	0.4163	1.027	3.846	2317	4.195	1.954	365	11.09	668	23.0	2.29	0.942	63.2	624.2	350
355	0.5100	1.030	3.180	2304	4.199	1.968	343	11.29	671	23.3	2.14	0.951	62.3	652.3	355
360	0.6209	1.034	2.645	2291	4.203	1.983	324	11.49	674	23.7	2.02	0.960	61.4	697.9	360
365	0.7514	1.038	2.212	2278	4.209	1.999	306	11.69	677	24.1	1.91	0.969	60.5	707.1	365
370	0.9040	1.041	1.861	2265	4.214	2.017	289	11.89	679	24.5	1.80	0.978	59.5	728.7	370
373.15	1.0133	1.044	1.679	2257	4.217	2.029	279	12.02	680	24.8	1.76	0.984	58.9	750.1	373.15
375	1.0815	1.045	1.574	2252	4.220	2.036	274	12.09	681	24.9	1.70	0.987	58.6	761	375
380	1.2869	1.049	1.337	2239	4.226	2.057	260	12.29	683	25.4	1.61	0.999	57.6	788	380
385	1.5233	1.053	1.142	2225	4.232	2.080	248	12.49	685	25.8	1.53	1.004	56.6	814	385

Table A.2. Aluminum 6061 properties (Aalco,2018)

Property	Value
Density	2.70 g/cm^3
Melting Point	650 °C
Thermal Expansion	23.4 x10^{-6} /K
Modulus of Elasticity	70 GPa
Thermal Conductivity	166 W/m.K
Electrical Resistivity	0.040 x10^{-6} Ω .m

Table A.3. Cooper properties (Chapman, 2014)

Property (at 20°C)	Copper (C101)
Electrical conductivity (annealed)	101
Electrical resistance (annealed)	17.2
Temperature coefficient of resistivity	0.0039
Thermal conductivity	397
Specific heat	385
Coefficient of expansion	17×10^{-6}
Tensile strength (annealed)	200-250
Tensile strength (half hard)	260-300
0.2% proof strength (annealed)	50-55
0.2% proof strength (half hard))	170-200
Elastic modulus	116-130
Density	8910
Melting point	1083

Table A.4. Common materials rugosity (Efunda,2007)

Pipe Material	**Absolute Roughness, *e***	
	$\times 10^{-6}$ feet	micron (unless noted)
drawn brass	5	1.5
drawn copper	5	1.5
commercial steel	150	45
wrought iron	150	45
asphalted cast iron	400	120
galvanized iron	500	150
cast iron	850	260
wood stave	600 to 3000	0.2 to 0.9 mm
concrete	1000 to 10,000	0.3 to 3 mm
riveted steel	3000 to 30,000	0.9 to 9 mm

7.2 Appendix B

Equation B.1. K formula for 90 degree bend in turbulent flow (White, 2011)

$$K_B = 0.388\,\alpha\,(\frac{R}{d})^{0.84} * (Re_D)^{-0.17}$$
$$\alpha = 0.95 + 4.42(\frac{R}{d})^{-1.96}$$

TABLE B.1. Level Bender Specification for Copper (RIDGID, 2013)

Catalog No.	Model No.	Description	Capacity		Weight	
			Tube Size (actual O.D.)	Bend Radius	lb.	kg
36117	403	Instrument Bender	3⁄16"	5⁄8"	1.5	0,86
36122	404	Instrument Bender	1⁄4"	5⁄8"	1.5	0,86
36092*	405	Instrument Bender	5⁄16"	5⁄16"	2.75	1,18
36097	406	Instrument Bender	3⁄8"	5⁄16"	2.75	1,18
36132	408	Instrument Bender	1⁄2"	1 1⁄2"	5.5	2,45
36112	406M	Instrument Bender	6 mm	16 mm	1.5	0,68
36092*	408M	Instrument Bender	8 mm	24 mm	2.75	1,18
36102	410M	Instrument Bender	10 mm	24 mm	2.75	1,18
36127	412M	Instrument Bender	12 mm	38 mm	5.5	2,45

7.3 Appendix C

Script C.1. Octave script for design proposal 1

Your Code ...

```
clc
%Datos
%Ld=7.5;
H=0.304;W=0.965;w=0.018;
BF=2.5;OD=0.0047625;
BR=BF*OD;
a=0.0127;
b=0.0144;
c=a+b;;
Ly=0.0254;
H_=H-c-Ly;
W_=W-(2*a);
C1=(pi*BR)/2;
h=OD;
% Relaciones
while BF<4.6
 fprintf('Table con n :%6.1f\n',BF);
fprintf('%9s %13s %13s %13s %13s %13s %13s %13s %13s %13s','n','L1','Lx','L3','L4','L2','C1','BR','Lt','h_');
for n=1:5
n1=(2*n)-1;
n2=4*n;
Lx=1.5*OD;
L3=(2*BR)+Lx;
LF=W_-w-BR;
L1=H_-BR;
L2=W_-(2*BR);
nL3=L3*n1;
L4=L2-w;
nL4=L4*n1;
nLx=n1*Lx;
%/n
nC1=C1*n2;
h_=H_-nL3-BR;
Lt=L1+L2+nC1+nL4+h_+nLx;
if h_>h
fprintf('\n');
     fprintf('%9.0f',n);
     fprintf('%16f',L1);
     fprintf('%16f',Lx);
        fprintf('%14.5f',nL3);
   fprintf('%14.5f',L4);
   fprintf('%14.5f',L2);
     fprintf('%14.5f',C1);
      fprintf('%14.5f',BR);
        fprintf('%14.5f',Lt);
        fprintf('%14.5f',h_);
end
end
BF=BF+0.1;
BR=BF*OD;
C1=(pi*BR)/2;
fprintf('\n');
fprintf('\n');
end
```

Script C.2. Octave script for design proposal 2

Your Code ...

```
clc
%Datos
%Ld=7.5;
H=0.304;W=0.965;w=0.018;
BF=2.6;OD=0.0047625;
BR=BF*OD;
a=0.0127;
b=0.0144;
c=a+b;;
Ly=0.0254;
H_=H-c-Ly;
W_=W-(2*a);
C1=(pi*BR)/2;
h=OD;
% Relaciones
while BF<4.6
 fprintf('Table con n :%6.1f\n',BF);
fprintf('%9s %13s %13s %13s  %13s %13s %13s %13s %13s %13s %13s %13s %13s','n','L1','L2','nL2','L3','nL3','L4','L5','Lx','BR','Lt','x_','C1');
for n=1:14
n1=(2*n)+1;
n2=(4*n)+4;
n3=2*n;
Lx=1.5*OD;
L4=H_-(3*BR)-h;
L3=L4-BR;
L1=H_-BR;
L2=(2*BR)+Lx;
L5=W_-w-(2*BR);
nL2=n1*L2;
nL3=n3*L3;
nLx=n1*Lx;
%/n
nC1=C1*n2;
x_=W_-nL2;;
Lt=L1+nC1+nL3+L4+h+nLx+L5+h;
if x_>0
fprintf('\n');
     fprintf('%9.0f',n);
       fprintf('%16f',L1);
     fprintf('%16f',L2);
        fprintf('%14.5f',nL2);
   fprintf('%14.5f',L3);
   fprintf('%14.5f',nL3);
     fprintf('%14.5f',L4);
      fprintf('%14.5f',L5);
      fprintf('%14.5f',Lx);
      fprintf('%14.5f',BR);
        fprintf('%14.5f',Lt);
        fprintf('%14.5f',x_);
          fprintf('%14.5f',C1);
end
end
BF=BF+0.1;
BR=BF*OD;
C1=(pi*BR)/2;
fprintf('\n');
fprintf('\n');
end
```

Script C.3. Octave script for design proposal 3

Your Code ...

```
clc
%Datos
%Ld=7.5;
H=0.304;W=0.965;w=0.018;
BF=2.5;OD=0.0047625;
BR=BF*OD;
a=0.0127;
b=0.0144;
c=a+b;;
Ly=0.0254;
H_=H-c-Ly
W1=W-(2*a)
C1=(pi*BR)/2;
h=OD;
d=0;
% Relaciones
while BF<4.6
 fprintf('Table with a BR of :%6.1f\n',BF);
fprintf('%9s %13s %13s %13s %13s %13s %13s %13s %13s %13s %13s %13s','n','L1','L3','L6','LD','Lx','L5','BR','Lt','Wx','C1','Lf');
for n=1:9
n1=n;
n2=n+1;
n3=(8*n)+2;
n4=(2*n)-1;
n5=(4*n);
n6=(2*n)+1;
Lx=1.5*OD;
L1=H_-BR;
L4=(2*BR)+Lx;
L2=(2*w)+L4;
L3=H_-d-w-(2*BR);
LD=(2*w)+L4-(2*BR);
L6=H_-(2*BR);
L5=H_-w-BR;
nL2=n1*L2;
nL4=n2*L4;
nLD=n4*LD;
nL3=n5*L3;
nL6=Lx*n6;
%/n
nC1=C1*n3;
x1=L4-BR;
x2=w-BR;
Lf=x1+x2+L4;
W2=nL2+nL4;
Wx=W1-W2;;
Lt=L1+nC1+nLD+nL3+nL6+Lf+L5;
if Wx>0
fprintf('\n');
     fprintf('%9.0f',n);
     fprintf('%16f',L1);
        fprintf('%14.5f',L3);
         fprintf('%14.5f',L6);
   fprintf('%14.5f',LD);
   fprintf('%14.5f',Lx);
   fprintf('%14.5f',L5);
      fprintf('%14.5f',BR);
        fprintf('%14.5f',Lt);
        fprintf('%14.5f',Wx);
        fprintf('%14.5f',C1);
         fprintf('%14.5f',Lf);
end
end
BF=BF+0.1;
BR=BF*OD;
C1=(pi*BR)/2;
fprintf('\n');
fprintf('\n');
end
```

7.4 Appendix D

Temperature distribution calculation

1.-

$$q = \dot{m}Cp\Delta \mathrm{T}_{IE} = \dot{m}Cp(T_{out} - T_{in})$$

$$T_{out} = \frac{q}{\dot{m}Cp} + T_{in}$$

$$T_{out} = \frac{387}{(0.0167) * (4180)} + 18$$

$$T_{out} = 23.5\ [degC]$$

2.-

$$T_m = \frac{T_{out} + T_{in}}{2}$$

$$T_m = \frac{23.5 + 18}{2}$$

$$T_m = 20.75\ [degC] = 293.9[K]$$

3.-

Water properties:

$$\mu_{@293.9} = 1009E - 6\ [\frac{Ns}{m^2}]$$
$$k_{@293.9} = 604.24E - 3\ [\frac{W}{mK}]$$
$$Pr_{@293.9} = 6.827$$

$$Re_D = \frac{4\dot{m}}{\pi D \mu}$$
$$\dot{m} = \rho Q$$

$$\dot{m} = (1000)(1.67E - 5)$$

$$\dot{m} = 0.0167\ [\frac{Kg}{s}]$$

$$Re_D = \frac{4(0.0167)}{\pi(3.175E - 3)(1009E - 6\ )}$$
$$Re_D = 6637.3$$
$$\frac{1}{\sqrt{f}} = -2\log(\frac{\frac{\varepsilon}{ID}}{3.7} + \frac{2.51}{(Re)f^{0.5}})$$

$$\frac{1}{\sqrt{f}} = -2\log(\frac{\frac{1.5E-6}{3.175E-3}}{3.7} + \frac{2.51}{(6637.3)f^{0.5}}$$

Initial guess: $f = 0.02$

$$\frac{1}{\sqrt{0.02}} = -2\log(\frac{\frac{1.5E-6}{3.175E-3}}{3.7} + \frac{2.51}{(6637.3)0.02^{0.5}}$$

$$7.07 = 5.1$$

By iteration:

$$f = 0.035$$

$$Nu = \frac{\left(\frac{f}{8}\right)(Re_D - 1000)Pr}{1 + 12.7\left(\frac{f}{8}\right)^{\frac{1}{2}}(Pr^{\frac{2}{3}} - 1)}$$

$$Nu = \frac{\left(\frac{0.035}{8}\right)(6637.3 - 1000)6.827}{1 + 12.7\left(\frac{0.035}{8}\right)^{\frac{1}{2}}(6.827^{\frac{2}{3}} - 1)}$$

$$Nu = 52.89$$

$$\mathrm{Nu} = \frac{hD}{k}$$

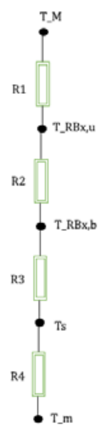
$$\mathrm{h} = \frac{Nuk}{D}$$

$$\mathrm{h} = \frac{(52.89)(604.24E-3)}{(3.175E-3)}$$

$$\mathrm{h} = 10065.6\ [\frac{W}{m^2K}]$$

4.-

Applied on the system, from the module temperature to the fluid temperature.



$$R_{tot} = \sum R_t$$

$$R_{tot} = R_1 + R_2 + R_3 + R_4$$

R_3 is negligible, then:

$$R_{tot} = R_1 + R_2 + R_4$$

$$R_{cd} = \frac{\Delta x}{k * A}$$

$$R_{cv} = \frac{1}{h * A}$$

Aluminium 6061 properties:

$$k_{Al} = 166 \ [\frac{W}{mK}]$$

$$R_{tot} = \frac{\Delta x_1}{k * A_1} + \frac{\Delta x_2}{k * A_2} + \frac{1}{h * A_3}$$

$$R_{tot} = \frac{0.002}{166 * 0.1451} + \frac{0.00635}{166 * 0.2934} + \frac{1}{10065.6 * \pi(3.175E - 3)L}$$

$$R_{tot} = 2.134E - 4 + \frac{9.965E - 3}{L}$$

5.-

$$UA = \frac{1}{R_{tot}}$$

$$UA = \frac{1}{2.134E - 4 + \frac{9.965E - 3}{L}}$$

$$\mathrm{q} = \mathrm{UA}\ \Delta T_{UA}$$

$$\mathrm{q} = \frac{1}{2.134E - 4 + \frac{9.965E - 3}{L}} \Delta T_{UA}$$

$$\Delta T_{UA} = \ q * (2.134E - 4 + \frac{9.965E - 3}{L})$$

Pipe friction losses calculation

$$\frac{v_1^2}{2g} + \frac{p_1}{\rho g} + z_1 = \frac{v_2^2}{2g} + \frac{p_2}{\rho g} + z_2 + h_l$$

$$h_l = h_f + h_m$$

$$\frac{v_1^2}{2g} + \frac{p_1}{\rho g} + z_1 = \frac{v_2^2}{2g} + \frac{p_2}{\rho g} + z_2 + h_f + h_m$$

$$h_m = \frac{kv^2}{2g}$$
$$h_f = \frac{fLv^2}{D2g}$$

$$\frac{\Delta p}{\rho g} = \frac{v^2}{2g}(\frac{fL}{D} + \sum K_B)$$

1.-

By using equation B.1:

$$K_B = 0.388\,\alpha\,(\frac{R}{d})^{0.84} * (Re_D)^{-0.17}$$
$$\alpha = 0.95 + 4.42(\frac{R}{d})^{-1.96}$$
$$\frac{R}{d} = \frac{\frac{19}{32}}{\frac{3}{16}}$$

$$\frac{R}{d} = 3.167$$

This result is then used to select the correct K_B to use. In order to do this, the equation of K_B is graph in the range of 1- 15 [R/d] to show how the factor is affected by the relationship.

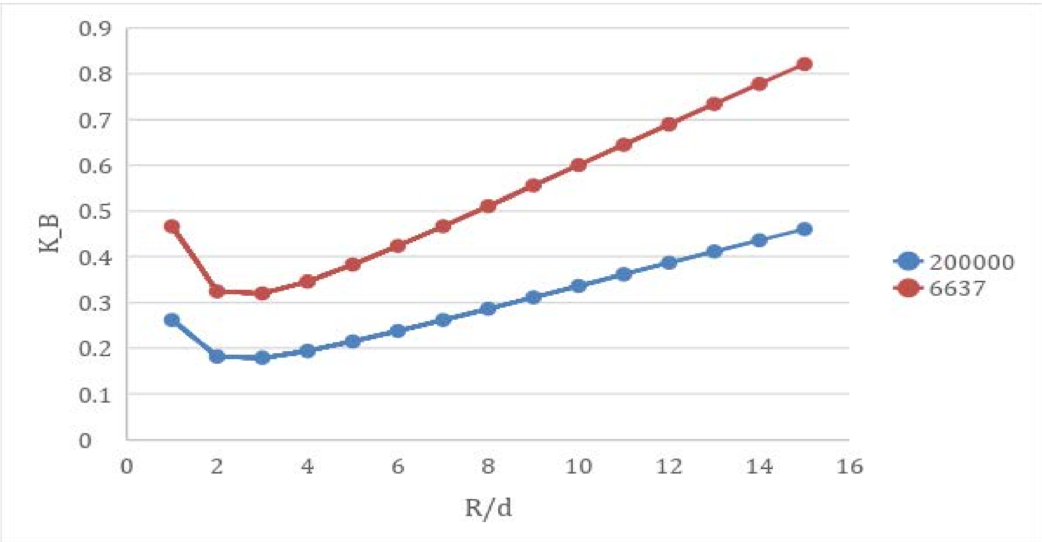


Figure D1. Bending factor graph

Furthermore, by using the figure D.1:

$$K_B = 0.323$$

2.-

$$Q = v * A$$

$$v = \frac{Q}{A}$$

$$v = \frac{1.67E-5}{(\frac{\pi}{4})(3.175E-3)^2}$$

$$v = 2.11\ [\frac{m}{s}]$$

3.-

$$\Delta p = [\frac{2.11^2}{2(9.81)}(\frac{0.035*L}{3.175E-3} + n*0.323)](1000)(9.81)$$

$$\Delta p = [0.227(\frac{0.035*L}{3.175E-3} + n*0.323)](9810)$$

Linear momentum calculation

For this analysis is taken a section (a bend) of the system, as show in the figure.

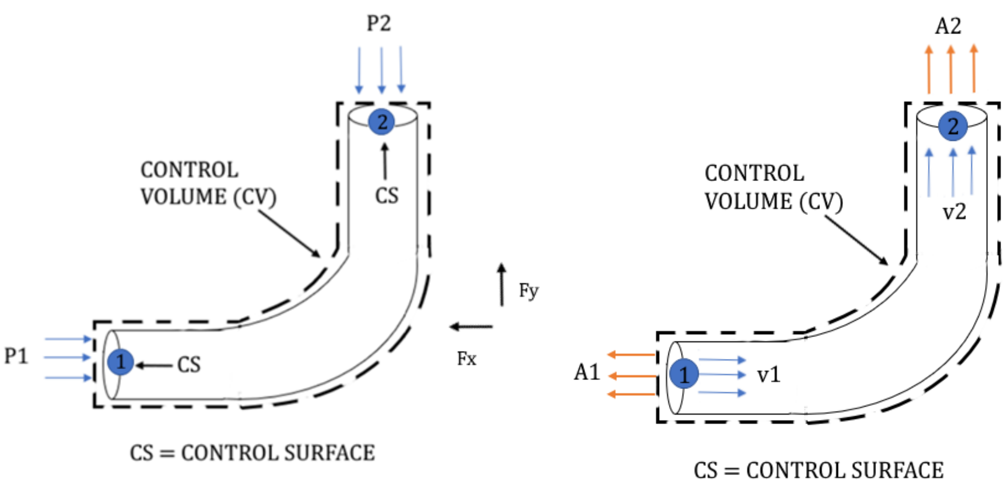


Figure D2. Diagram for linear momentum analysis

$$\sum F = \frac{d}{dt}\int_{cv}\rho \mathrm{v} d\bar{v} + \sum_{cs}\rho v(v*A)$$

Linear momentum in x direction (→)

$$\sum F = \frac{d}{dt}\int_{cv}\rho \mathrm{v}_x d\bar{v} + \sum_{cs}\rho \mathrm{v}_x(v*A)$$

$$P_1A_1cos0 - P_2A_2cos\ 90 - \mathrm{F}_x = \rho \mathrm{v}_2A_2\mathrm{v}_2\ cos\ 90 + \rho \mathrm{v}_1A_1\mathrm{v}_1cos\ 180$$

$$\mathrm{F}_x = P_1A_1 + \rho \mathrm{v}_1A_1\mathrm{v}_1$$

Linear momentum in y direction (↑)

$$\sum F = \frac{d}{dt}\int_{cv}\rho \text{v}_y d\bar{v} + \sum_{cs}\rho \text{v}_y (v * A)$$

$$P_1 A_1 cos\ 90 + P_2 A_2 cos 180 + \text{F}_y = \rho \text{v}_2 A_2 \text{v}_2\ sin\ 90 - \rho \text{v}_1 A_1 \text{v}_1 cos\ 90$$

$$\text{F}_y = P_2 A_2 + \rho \text{v}_2 A_2 \text{v}_2$$

Resultant force

$$\text{F}_R = \sqrt{\text{F}_x{}^2 + \text{F}_y{}^2}$$

The linear momentum analysis determines that if the pipe is not hold on the bend, it will have movement in the direction of the resultant force. Nevertheless, if the pipe is fix is determined that the fluid change only its direction and that the bend does not affect is potential behavior. To indicate how the degree of bend affect the resultant force on the pipe, a figure is show below considering the following results.

Table D.1. General parameters of the tube

Variable	Value
ΔP	277[$\frac{N}{m^2}$]
Cross sectional area of tube	7.92E-6[m^2]
Mean velocity	2,11 [$\frac{m}{s}$]
Mass flow rate	0,0167 [$\frac{Kg}{s}$]
Pressure at exit of tube	101325 [$\frac{N}{m^2}$]

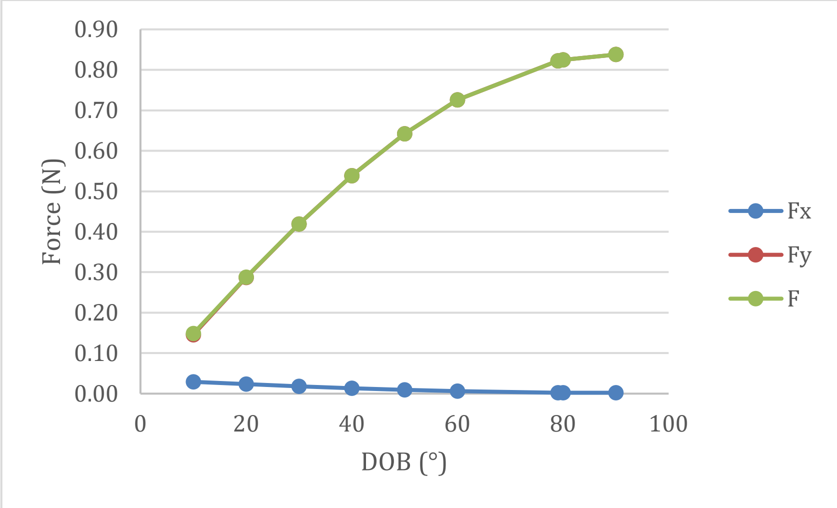


Figure D3. Effect of DOB on the resultant force

This indicate that the force in the x direction does not occupied a value importance in general, due that the force that considers more is in the y direction. It is justified by the fact that as the degree of bend increases, the x force decreases to approximate 0. The values obtained demonstrate that the force impact on the bend is not critically and effects of it can be disregarded.

Therefore, the conservation of momentum establishes that the fluid does not experience changes in it, the same energy is conserved. However, from the friction losses analysis is determine that the friction forces affected it, then, its kinetic energy change.

Pump calculation

$$\frac{v_1^2}{2g}+\frac{p_1}{\rho g}+z_1+H_P=\frac{v_2^2}{2g}+\frac{p_2}{\rho g}+z_2+h_l$$

$$h_l=h_f+h_m$$

$$\frac{v_1^2}{2g}+\frac{p_1}{\rho g}+z_1+H_P=\frac{v_2^2}{2g}+\frac{p_2}{\rho g}+z_2+\frac{fLv^2}{D2g}+\frac{kv^2}{2g}$$

$$H_P=\frac{v^2}{2g}(\frac{fL}{D}+\sum K_B+1)$$

$$H_P=\frac{Q^2}{2gA^2}(\frac{fL}{D}+\sum K_B+1)$$

$$H_P=\frac{Q^2}{2gA^2}(\frac{fL}{D}+\sum K_B+1)$$

Is consider 60 bends and Q changes from 0.1 L/min to 5 L/min. Also, L is the sum of the length of connections and the tubes of the cooling system. The length connection tubes are approximated 3 [m] and the length of the design is varied in the range 6-10[m]. Therefore:

$$H_P = \frac{Q^2}{2(9.81)(7{,}92E-06^2)}(\frac{(0.035)L}{3.175E-03} + (60 * 0.323) + 1)$$

The followings figures are obtained:

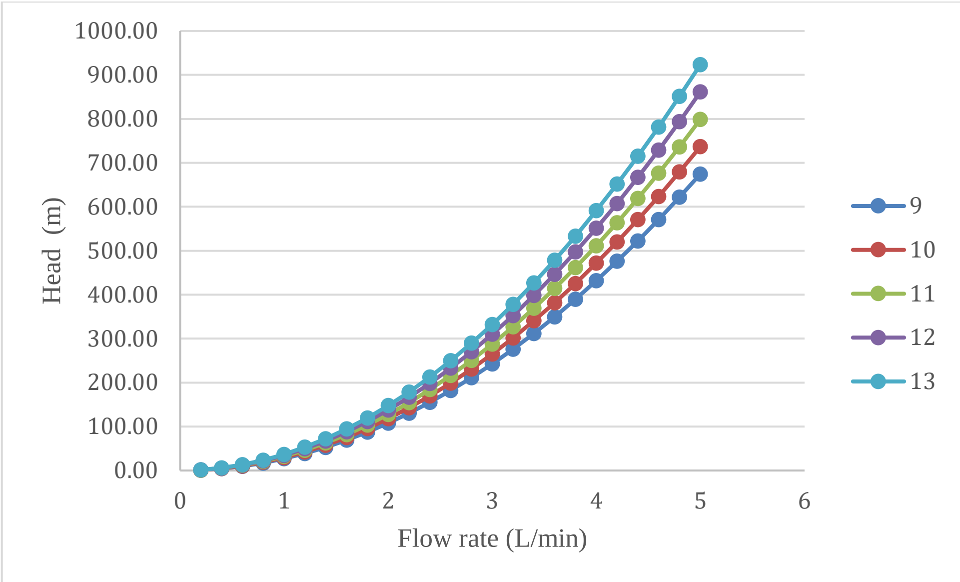


Figure D4. System curve of prototype

Where, the average head for a flow rate of water at 1lit/min is 35 [m]