Design of an evaporator for a buoyant two phase flow cooling loop of a telecommunication satellite

CICORIA David

Abstract

Satellites are constituted of electronic components that release heat flux. The heat released must be evacuated otherwise there are some risks to damage components in the satellite. In order to guarantee the correct working of the satellite, a thermal loop crossed by a refrigerant fluid must be added. The loop is constituted by a heater, a pressure regulator, a condenser, and a pump. The evaporator is the element of the loop in which the heat released by the electronic components is transferred to the fluid. The fluid, receiver of the components entailed energy will then be heated and will evaporate itself little by little. This breeds the appearance of a two phase flow.
A program, developed with the MATLAB code will take into account the different models of two phase flow that we can consider in our case. We will then be able to follow the evolution of the physical variables which characterize the system like the component temperature, the drop loss, void fraction and the quality. So, this study will provide a visualization of the thermo-hydraulics phenomenon in the evaporator, in the domain of specifications.

Keywords: Evaporator, two-phase flow, heat transfer, micro-gravity

Contents

Introduction

General context

Complex devices have more and more electronic components. However, as the power surges, heat production becomes important. It is therefore necessary to evacuate it to ensure the proper operation of the appliance. This evacuation can become a vital system for the latter in some cases. This is the case in satellites, which are based on the proper functioning of their equipment. However, the evacuation of the heat is here particularly delicate because the mechanisms involved are limited.

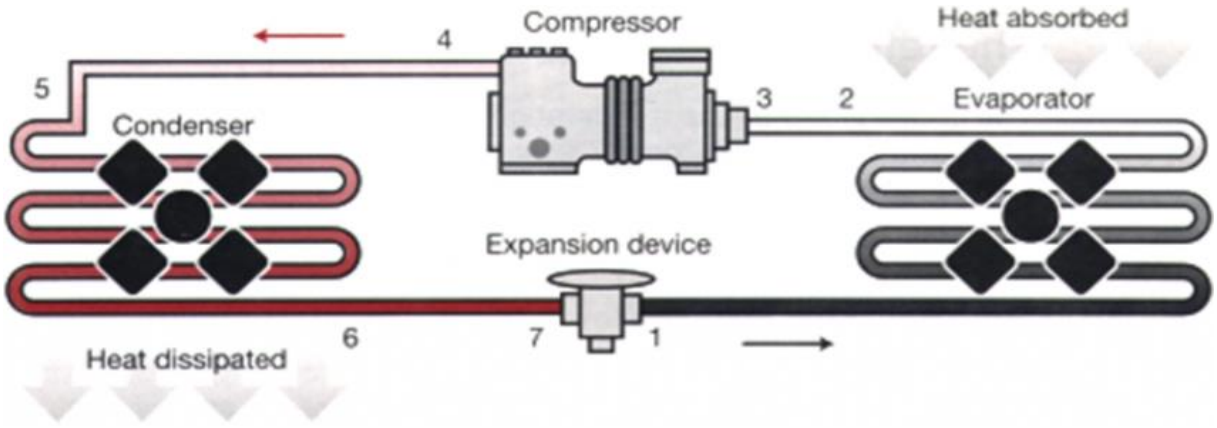


Fig. Working cycle of refrigeration system

In fact, on board a satellite, the heat is mainly emitted by radiation at the level of the external panels. The purpose of this project is to study a cooling loop, from the condenser to the evaporator. In our case, the refrigerant fluid circulates in a test evaporator, the electronic components being replaced by imitators producing only heat. This heat is subsequently evacuated by vaporizing the fluid when passing the fluid at the component level. A diphasic flow occurs in the evaporator. This method is more efficient than a conventional cooling, in monophasic flow, because the flow of heat transported is more important. This system has a considerable advantage over the capillary pumping system: the radiation temperature at the condenser level is about 100 °C vs. 40 °C with a conventional heat pump system. This makes it possible to reduce the surface of the side panels by 60% and therefore the weight of the installation. However, due to the diphasic flow, the simulation of it is more delicate.

Objective

The project deals with the sizing of an evaporator for a thermal control loop of a telecommunication satellite. The objective is to develop a MATLAB code in order to calculate different parameters of the evaporator, such as the pressure losses within the evaporator, or the temperature of the components, in order to validate the load specifications: **Do not have a loss of pressure too large or do not exceed a critical value for the temperature of the components**.

We will have to take the right simulation models according to the convective boiling regime (in diphasic), in which we find ourselves. We also need to consider the loss of load that can occur at the level of our flow, and especially the singular load losses here. In fact, the presence of the elbows will lead to load losses. The problem is that these losses are difficult to model because our flow is diphasic. To do this, we need to use experimental studies that have already been carried out. In order to

characterize our convective heat exchanges in the tube, diphasic correlations will be used to calculate a heat-exchange transfer coefficient (h) that will therefore be variable within our flow.

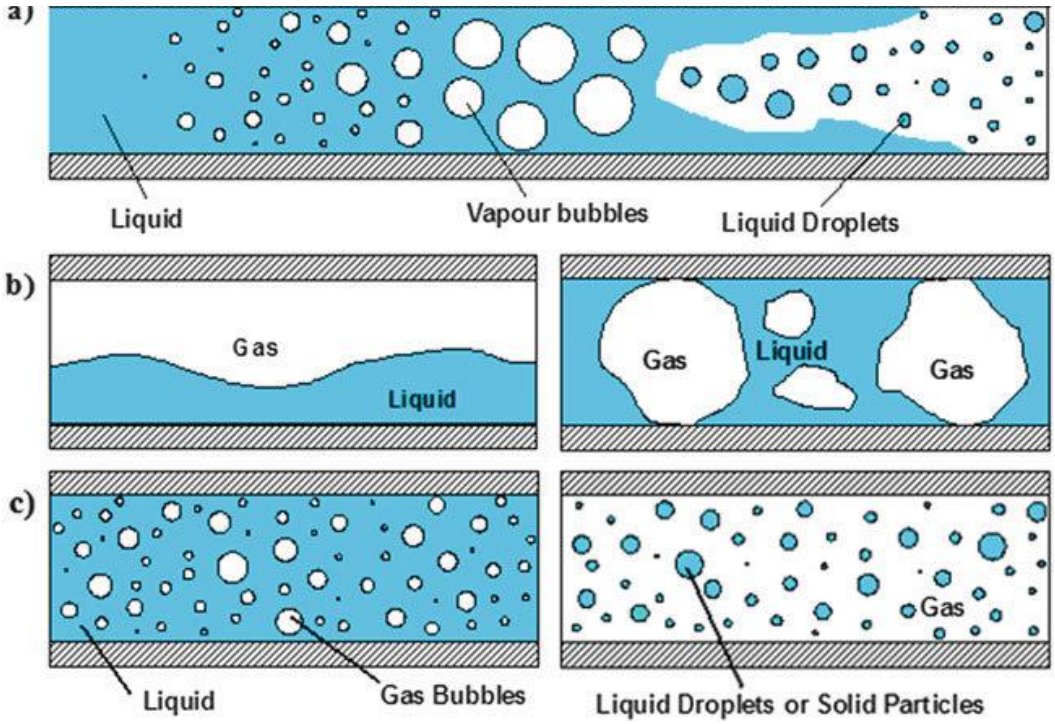


Fig. Evolution form a monophasic flow to a diphasic flow

Notations

- **Indices**

Letter	Meaning
i	Interface
l	Liquid
g	Gas
sat	Saturation
p	Wall
e	Entry
s	Output
m	Mixture

- **Problem Variables**

Parameter	Meaning
x	Mass title
T	Temperature
P	Pressure
Rg, Rl	Void rate (gas or liquid)
z	Tube axis
E	Training rate

- **Evaporator parameters**

Parameter	Meaning
L	Length of the Evaporator
D	Tube diameter
A	Tube Section
e	Tube thickness
Sp	Wet perimeter

- **Flow parameters**

Parameter	**Meaning**
$\dot{m}$	Mass flow rate
D_h	Hydraulic diameter
G	Surface mass Flow
J	Surface Velocity
U	Average Speed
f	Friction coefficient

- **Parameters related to the thermal**

Parameter	Meaning
q	Flux surface density exchanged with the wall
R	Thermal resistance

- **Fluid Properties**

Parameter	Meaning
Cp	Heat capacity
h_{lv}	Latent heat of vaporization
ρ	Density
μ	Dynamic viscosity
σ	Tension
k	Thermal conductivity

- **Non-dimensional names**

Number	Meaning
Re	Reynolds number
Fri	Froude number
Pr	Prandtl number

Specifications

The prescribed specifications are presented below:

Structure

- Imposed refrigerant fluid (R245fa)
- Parallel hydraulic network (3 tubes in parallel)
- Size of fins
- Total system length
- Shape of the imposed tubes (cylindrical)
- Positioning of imitators and dissipated heat fluxes
- Contact resistance fin/mimic (2*10^-4 K. m ^ 2/W)
- Contact resistance fin/tube (to be determined experimentally)

Temperatures and flow rates

- Maximum operating temperature of components to check
- Imposed flow range (10 g/s per tube)
- Entry title imposed (between 0 and 0.3)

Geometry

The geometry is basically composed of several loops like this and can't be revealed, as It belongs to the industry partner

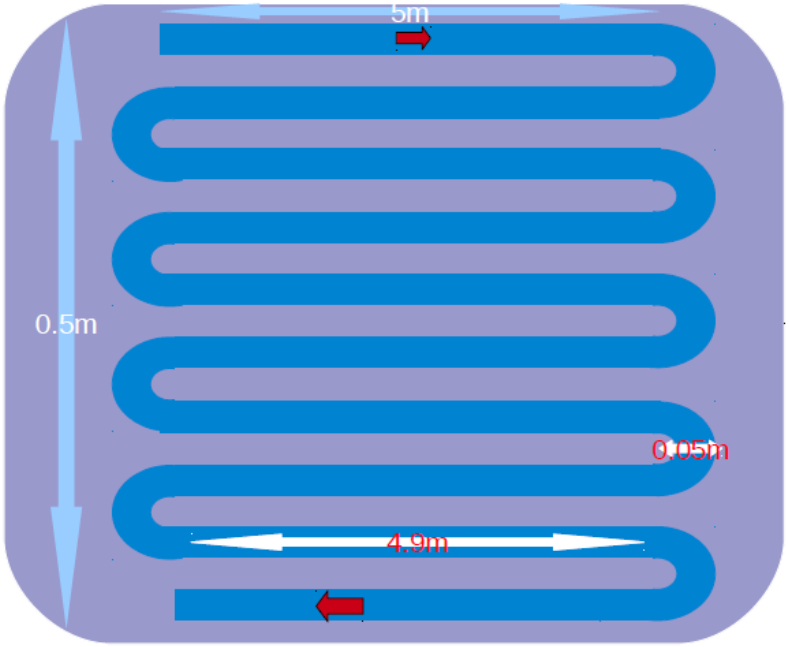


Fig. Scheme of an evaporator loop for the geometry.

1.) Modeling physical Phenomena

In order to best model the thermo-hydraulic phenomena, we have used different diphasic models.

Due to the results of the preliminary studies, the inlet flow of the evaporator is assumed to be annular with the gas circulating in the middle of the pipe and the liquid lying at the level of the wall. The heat emitted by the electronic components will cause the liquid to change phase. The steam title will grow, and the surface speed of the gas will increase. Due to the difference in speed between gas and liquid, there is going to be a tearing of liquid droplets at the interface between the two phases from a critical velocity. From a study carried out previously, we could see that the grubbing-up had no effect on the flow. This is surely because the number of droplets pulling out is low vis-à-vis the annular flow. Thus, we decided not to model this phenomenon of grubbing because of its negligible weight.

From a purely thermal point of view, we place ourselves in a saturated boiling regime and model the heat exchange coefficient of Kandlikar, Gunger and Winterton, and Schrock and Grossman. When the title of the vapor phase is important enough to consider that the flow is monophasic, we use the monophasic thermal model. We modeled the interfacial frictions with the Wallis and parietal friction with the models of Lockhart&Martinelli, of Baroczy, and of Awad. We will also note that a singular load loss model at the elbows (valid in no gravity) has been implemented. From our various studies, we have found that the homogeneous model should be used for weaker titles, than the ones we are studying. Indeed, we find that the velocity changes are increasingly weak when the steam title is also. Finally, when the steam title is unitary, we use a monophasic model.

1.1) Hydraulic Models

1.1.1) Annular model without tearing

When the gas flow is important, the forces of gravity are highly negligible in front of the inertia forces. The liquid then completely moistens the wall by forming an annular film around the gas flow. The flow remains at separate phases. In addition, at very high gas flow drops of liquid are torn off and driven by the gas phase. This flow is said to be annular to droplets (annular dispersed).

The annular model without tearing [1] Consists of two equations of movement quantity as well as the equation of conservation of the enthalpy. This system allows us to determine the pressure gradient, the evolution of the void rate as well as the title of the coolant fluid.

Movement Quantity Conservation equations

Equation 1:

$$\frac{dRg}{dz}G^2\left(\frac{R_l x^2}{\rho_g {R_g}^2}+\frac{R_g(1-x)^2}{\rho_l R_l^2}\right)=-\frac{\tau_{ig}4}{D}\sqrt{R_g}+R_g\frac{\tau_p 4}{D}-(\rho_l-\rho_g)R_g R_l g$$
$$+G^2\frac{dx}{dz}\left(\frac{2xR_l}{\rho_g R_g}+\frac{(1-x)(2R_g-1)}{\rho_l R_l}\right)$$

Equation 2:

$$\frac{dp}{dz}=-\frac{d}{dz}\left(\frac{G^2x^2}{\rho_g R_g}\right)-\frac{d}{dz}\left(\frac{G^2(1-x)^2}{\rho_l R_l}\right)+\frac{\tau_p 4}{D}-(\rho_l R_l+\rho_g R_g)g$$

Equations for the conservation of enthalpy

Equation 3:

$$\frac{dx}{dz} = \frac{4q}{GDh_{lv}}$$

Equation 4:

$$\delta = 0.5D(1 - \sqrt{R_g})$$

These equations are used when the refrigerant is at a component level. When this is not the case, we consider the constant vacuum rate, as well as the title (there is no heat exchange). The terms of derivatives in the right-hand member of Equation 2 are then null, which also simplifies its expression.

1.1.2) Homogeneous model

The Homogeneous model [1] is valid in the context of dispersed flows with low speed of sliding of the gas relative to the liquid. This model assumes that the gas velocity is equal to the speed of the liquid either $U_l = In_g = In_M$.

In our configuration, we do not use the homogeneous model because the equal velocity hypothesis tends to be validated when the steam title is in a lower range than the one we are studying. We put it in this report, only for an informative purpose.

The hypothesis that $U_l = In_g = In_M$ gives us 2 equations that allow the simplification of the model.

Equations 5 and 6:

$$\rho_M = R_g\rho_g + (1 - R_g)\rho_l \quad (5) \quad et \quad \frac{Gx}{\rho_g R_g} = \frac{G(1-x)}{\rho_l(1-R_g)} \quad (6)$$

The other equations to be computed also correspond to a movement quantity conservation equation and a pressure-resolution equation.

Equation 7:

$$\frac{\partial \rho_M U_M}{\partial t} + \frac{\partial}{\partial z}[\rho_M U_M^2] = \frac{\partial G}{\partial t} + \frac{\partial}{\partial z}\left[\frac{G}{\rho_M}\right] = -\frac{dP}{dz} + \frac{\tau_p S_p}{A} - \rho_M g sin\theta$$

Equation 8:

$$\left(\frac{dp}{dz}\right)_{fr} = \frac{\tau_p S_p}{A} = -\frac{S_p}{A}\frac{1}{2} f_{pM}\frac{G^2}{\rho_M} = -\frac{S_p}{A}\frac{1}{2} f_{pM}\rho_M U_M^2$$

The coefficient f_{pm} corresponds to the coefficient of skin friction. It is determined according to the value of the average Reynolds, noted Re_M. On a $Re_M = \frac{GD}{\mu_M}$

The value of the average viscosity μ_M is given by $\mu_M = R_g\mu_g + (1 - R_g)\mu_l$

- If $Re_M < 2000$ Then we have $f_{pm} = \frac{16}{Re_M}$
- If $Re_M > 2000$ so we got $f_{pm} = 0.079Re_M^{-0.35}$

1.1.3) Monophasic model

When the steam title reaches the unit, we then use the Monophasic model[1]. Our specifications imposing on output, a mass title of 1 at the level of the vapor phase, it is normal that we are interested, in a one fluid model to calculate the pressure loss.

Equation 9:

$$\frac{dp}{dz} = \frac{4\tau_p}{D}$$

With $\tau_p = -0.5 * f_c\rho_g U_g^2$

In addition, the value of f_c depends on which flow regime it is located:

- If the flow is laminar, or $Re_g < 2000$ Then $f_c = \frac{16}{Re_g}$
- If the flow is turbulent or $Re_g > 2000$ Then $f_c = 0.079\ Re_g^{-0.25}$

1.2) Model complements

1.2.1) Interfacial friction

Wallis model

The Wallis model allows us to model the interfacial friction τi between the liquid phase and the gas phase. We have the following system of equations:

Equation 10:

$$\tau_i = -\frac{1}{2} f_i \rho_g |U_g - U_l| (U_g - U_l)$$

Equation 11:

$$f_i = 0.005\left(1 + 300\frac{\delta}{D}\right) = 0.005\left[1 + 150\left(1 - \sqrt{R_g}\right)\right]$$

The gas and liquid velocities are determined from the surface velocities calculated at each step of space.

Equation 12:

$$U_g = \frac{J_g}{R_g} \text{ And } U_l = \frac{J_l}{R_l} \text{ With } R_g + R_l = 1$$

Equation 13:

$$J_g = \frac{Gx}{\rho_g} \text{ And } J_l = \frac{G(1-x)}{\rho_l}$$

1.2.2) Skin friction

1.2.2.1) Lockhart and Martinelli model

The Lockhart and Martinelli model[1] allows to determine the skin friction at the level of our system. At the level of this model, we must calculate the parameter C first. This parameter is necessary for us to calculate the pressure loss. The Coefficients of wall friction are defined in the following way:

Equation 14:

$$f_{pl} = K\left(\frac{J_l D}{\nu_l}\right)^{-n}$$

Equation 15:

$$f_{pg} = K\left(\frac{J_g D}{\nu_g}\right)^{-n}$$

In laminar flow n = 1, k = 16 and in turbulent flow n = 1/4, k = 0.079.

In order to calculate the parameter C, we must first look at the flow regime in which we are located, both at the gas level and the liquid. So we need to know the values of the Re.

The following table shows the value of parameter C based on flow regimes.

Liquid	Gas	C
Turbulent	Turbulent	20
Laminar	Turbulent	12
Turbulent	Laminar	10
Laminar	Laminar	5

The Laminar regime is Defined for a Reynolds less than 2000 and the turbulent regime for a Reynolds greater than 3000. So, there is no correlation in the transition zone. In our simulations, the regime change caused discontinuities in the expression of the diphasic friction constraint. The problem was solved using the smoothing proposed by Julien Hugon. This linear interpolation of C is a function of log (Rel) and/or log (Reg). When only one of the two phases are in the transition zone (most frequently encountered), linear interpolation of C is Type C (rel) = a * log (rel) + B or C (reg) = a ' * log (reg) + B '.

For example, in the case where the gas is in turbulent conditions and the liquid is in transition, the correlation used is:

Equation 16:

$$C(Re_l) = \frac{20-12}{\log(3000)-\log(2000)} * (\log(Re_l) - \log(2000)) + 12$$

In the case where the two phases are in transition zones, two linear C-interpolations of type C (rel, REG) = A * log (rel) + b * Log (REG) + C are used, one for rel > Reg and the other for rel < Reg.

Then we can calculate φ_l And φ_g :

Equation 17:

$$\varphi_l^2 = (1 + \frac{C}{X} + \frac{1}{X^2})$$

Equation 18:

$$\varphi_g^2 = (1 + CX + X^2)$$

To calculate these 2 parameters, we need to X:

Equation 19:

$$X = \frac{J_l}{J_g} * \sqrt{\frac{\rho_l}{\rho_g} * \frac{f_{pl}}{f_{pg}}}$$

We can calculate load losses in the liquid phase and the gas phase.

Equation 20:

$$(\frac{dp}{dz})_l = -\frac{S_p}{A} f_{pl} \frac{\rho_l J_l^2}{2}$$

Equation 21:

$$(\frac{dp}{dz})_g = -\frac{S_p}{A} f_{pg} \frac{\rho_g J_g^2}{2}$$

We can calculate load losses by friction:

Equation 22:

$$(\frac{dp}{dz})_{fr} = \frac{\tau_p S_p}{A} = \varphi_l^2 \left(\frac{dp}{dz}\right)_l = \varphi_g^2 \left(\frac{dp}{dz}\right)_g$$

1.2.2.2) Baroczy model

The Baroczy correlation[4] expresses the loss of pressure by diphasic friction depending on the friction loss (DP/DZ), that one would have if the liquid phase flowed alone with the same mass flow rate as the diphasic flow, $\dot{m}$. For this, it is necessary to be able to express the multiplicative coefficient φ_{l0} defined in the following way:

Equation 23:

$$\varphi_{l0}^2 = \frac{(\frac{dp}{dz})_{fr}}{(\frac{dp}{dz})_l}$$

For this, we will also use friction loss $(DP/DZ)_g$, that one would have if the vapor phase flowed alone with the same mass rate as the diphasic flow, and introduce the variable Y defined in the following way:

Equation 24:

$$Y^2 = \frac{(\frac{dp}{dz})_g}{(\frac{dp}{dz})_l}$$

$(dP/dz)_l$ And $(dP/dz)_g$ are defined in the same way as in the Lockhart and Martinelli model (**See equations 20 and 21**).

The two phases flowing alone, with the same mass flow as the diphasic flow, would have respectively the average velocities $V_l = G/\rho_l$ And $V_g = G/\rho_g$, from which we can define Reynolds numbers Re_{vl} And Re_{vg} . So, we have:

Equation 25:

$$Y^2 = \frac{\rho_l}{\rho_g}(\frac{\mu_g}{\mu_l})^n$$

Baroczy gives an expression of φ_{l0} , from many experimental points grouping different fluids. The correlation obtained is graphic, but its curves were subsequently correlated by Chisholm in 1973, which obtained the following expression:

Equation 26:

$$\varphi_{l0}^2 = 1 + (Y^2 - 1)[Bx^{\frac{2-n}{2}}(1-x)^{\frac{2-n}{2}} + x^{2-n}]$$

n is the opposite of the exponent of Re_{vl} That we have in F (Re_{vl}), or n = 1 if the liquid flow alone is laminar (Re_{vl} < 2000) and n = 0.25 If it is turbulent (Re_{vl} > 3000). For the transition zone, the same value is used in the model as in the turbulent case, i.e. N = 0.25. B is a Y-and G-dependent variable whose expression varies depending on the value of y:

Y value	0 < and < 9.5	9.5≤and < 28	28≤Y
Value of B	$\frac{55}{G^{1/2}}$	$\frac{520}{YG^{1/2}}$	$\frac{15000}{Y^2G^{1/2}}$

Finally, the pressure losses can be calculated by friction:

Equation 27:

$$(\frac{dp}{dz})_{fr} = \varphi_{l0}^2(\frac{dp}{dz})_l$$

1.2.2.3) Awad model

The Awad model [8] is another empirical model for determining parietal friction. The Reynolds number of each phase is calculated first:

Equation 28: $Re_l = \frac{J_l D}{\nu_l}$ And $Re_g = \frac{J_g D}{\nu_g}$

The liquid and gaseous parietal friction coefficients are defined in the same way as for the Lockhart and Martinelli model (**See Equations 14 and 15**). Depending on the regime, the value of the parameter K and n are inferred:

Laminar flow = > K = 16 and n = 1

Turbulent flow = > K = 0.0079 and n = 0.25

We then calculate X, in the same way as for the Lockhart and Martinelli model:

Equation 29:

$$X = \frac{J_l}{J_g} * \sqrt{\frac{\rho_l}{\rho_g} * \frac{f_{pl}}{f_{pg}}}$$

The definition of φ_lAnd φ_g change, though.

Equation 30:

$$\varphi_l^2 = (1 + \left(\frac{1}{X^2}\right)^p)^{\frac{1}{p}}$$

Equation 31:

$$\varphi_g^2 = [1 + (X^2)^p]^{\frac{1}{p}}$$

The value of the p parameter is difficult to determine. It varies depending on the case, and generally it is chosen as the minimum of the RMS error. To ensure robustness conditions, we will take **p = 2/7**. You can then calculate the liquid pressure gradient and the gaseous one.

Equation 32:

$$(\frac{dp}{dz})_l = -\frac{S_p}{A} f_{pl} \frac{\rho_l J_l^2}{2}$$

Equation 33:

The friction pressure gradient is given to us by the following formula.

Equation 34:

$$(\frac{dp}{dz})_g = -\frac{S_p}{A} f_{pg} \frac{\rho_g J_g^2}{2}$$

The friction pressure gradient is given to us by the following formula.

Equation 35:

$$(\frac{dp}{dz})_{fr} = \frac{\tau_p S_p}{A} = \varphi_l^2 \left(\frac{dp}{dz}\right)_l = \varphi_g^2 \left(\frac{dp}{dz}\right)_g$$

Finally, the model of Awad takes up the same equations as the model of Lockhart and Martinelli. The only parameters that change are the factors diphasic φ_l And φ_g. The level of the calculation method, it is also like this model as well as to the results, by the tests carried out.

1.2.3) Model of pressure loss in the elbows

In the literature, we have determined the empirical model suitable for load losses [3]. Δp_{rb} corresponds to the total load loss in the elbow for a diphasic flow.

Equation 36:

$$\Delta p_{rb} = \phi \Delta p_{sp}$$

With Δp_{sp}, total loss of load in the elbow in monophasic situation, expressed in the following manner:

Equation 37:

$$\Delta p_{sp} = K_{sp} \frac{G^2}{2\rho_l}$$

In Equation 23, *Ksp* corresponds to the local charge loss coefficient for monophasic liquid flow. To estimate the value of *Ksp*, [Idelshik 1986] suggests the following expression:

Equation 38:

$$K_{sp} = f_{pl} \frac{L}{D} + 0.294 (\frac{R}{D})^{0.5}$$

Where R is the radius of curvature and f_{pl} the coefficient of parietal friction calculated by the models mentioned above.

The multiplier factor for diphasic flow ϕ is given by:

Equation 39:

$$\phi = 1 + \left(\frac{\rho_l}{\rho_g} - 1 \right) x[b(1-x) + x]$$

With b expressed in the following way:

Equation 40:

$$b = 1 + \frac{2.2}{K_{sp}(2 + \frac{R}{D})}$$

1.3) Thermal models

1.3.1) Temperature calculation

1.3.1.1) Convective model

At the level of our geometry, we know all the constant fluxes released by the electronic components. This heat created is of course dissipated by the vaporization of the fluid circulating within the evaporator. Our goal is to determine the temperature of the different components. For this, we consider that the fluid enters the evaporator at its Tsat saturation temperature. The entire heat flux will be used to vaporize the refrigerant. Locally, a thermal balance gives us:

Equation 41:

$$G * C_p * dT = q\pi D dz$$

For each step of space, we then obtain the value of the temperature at the wall of our evaporator. Furthermore, it is assumed that when a state change occurs at the fluid level, the vapor and liquid phases are in thermodynamic equilibrium. It is then possible to access the evolution of the mass title along the evaporator:

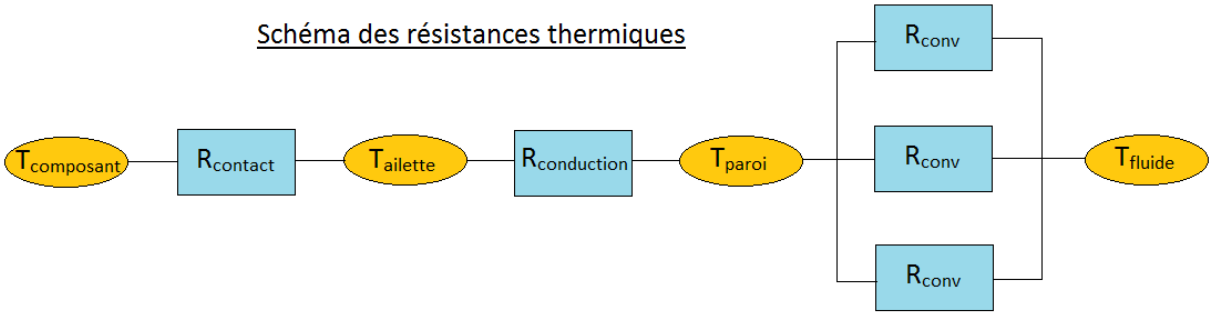
Equation 42:

$$\frac{dx}{dz} = \frac{4 * q}{m * h_{lv} * D}$$

Now that we know the evolution of the temperature of the wall of our evaporator, we are able to determine the evolution of the temperature of the different components. Indeed, a "circuit" of thermal resistances allows to connect these 2 temperatures. At the level of our geometry, the cylindrical tubes are attached to fins. These fins are in contact with the components in question. Our components, as previously said, are considered to be imitators that dissipate energy in a homogeneous way. The different resistances considered are:

- A convection resistance at the level of the fluid/wall exchange, determined by heat exchange coefficient models.
- A conduction resistance between the wall of the tubes and the fins. As this resistance has not yet been determined, we have carried out a simulation under COMSOL to derive an order of magnitude.
- A contact resistance between the sole of the evaporator and the copycat which is **2e-4 K. m²/w.**

Below, the figure shows us a pattern of thermal resistances. This pattern is right there with a representative character. Indeed, some components include 9 tubes (3 * 3 tubes), and in this case the direction of flow is not the same in the different tubes covered by the component (change of the sign of gravity).



We can nevertheless from the previous figure find how to calculate our component temperature from that of the fluid, all other quantities being known.

Equation 43: $T_{wall} = T_{sat} + \frac{q}{h}$

Equation 44: $T_{fin} = T_{wall} + R_{cond} * \frac{Q*3*n_{passages}*\pi D}{Ll^2}$

Equation 45: $T_{component} = T_{fin} + R_{contact} * \frac{Q}{Ll}$

Where Q is the power dissipated by the W-element, the width of the fin, the length of the component and $n_{passages}$ The number of fluid passages.

1.3.1.2) Conductive model

In the case where the liquid film has a laminar flow (Rel < 2000), the wall temperature can be calculated from a conductive model.

The following equation is obtained:

Equation 46:

$$q_p = k_l \frac{T_p - T_{sat}}{\delta}$$

With δ the thickness of the film and k_l the thermal conductivity of the fluid.

1.3.2) Model of convective exchange coefficient

At the convective exchange rate, we decided to test the several models at our disposal applying to the saturated boiling regime. Indeed, this regime is the one that is correlated with the annular regime, from a thermal point of view. Since in the evaporator, the bulk of the fluid will be in this form there in diphasic, it makes sense to be interested in these different models. When the mass Title X → 1, we know we're getting closer to a monophasic situation. At this time, of course, a monophasic model is

used for the thermal aspect. In all models that will follow, the mass title is always calculated by the following enthalpy balance:

Equation 47:

$$x(z) = \frac{4q_p}{DGh_{lg}}(z - z_s)$$

1.3.2.1) Kandlikar model

The correlation of Kandlikar (1989) [1] is described by the following equation:

Equation 48:

$$h = h_l\left[C_1 C_0^{C_2}(25Fr)^{C_5} + C_3 Bo^{C_4} F_k\right]$$

In order to determine the different constants of equation 48, it is necessary to be interested in our flow in the fluid region. For this, we will calculate various numbers dimensionless:

- $C_0 = \left(\frac{1-x}{x}\right)^{0.8}\sqrt{\frac{\rho_g}{\rho_l}}$
- $Bo = \frac{q}{Gh_{lg}}$
- $Fr = \frac{G^2}{\rho_l^2 gD}$

Now we can determine the different C_i from the value of C_0 :

	$C_0 < 0.65$ Convective region	$C_0 > 0.65$ Nucleate Boiling Region
C_1	1.1360	0.6683
C_2	-0.9	-0.2
C_3	667.2	1058
C_4	0.7	0.7
C_5	0.3	0.3

$C_5 = 0$ for vertical tubes and horizontal tubes when Fr > 0.04. For our fluid, we have Fr = 1.4

All that remains is to calculate h_l and then we will have the value of h correlated.

Equation 49:

$$h_l = 0.023\frac{\lambda_l}{D}\left(\frac{G(1-x)D}{\mu_l}\right)^{0.8} Pr^{1/3}$$

1.3.2.2) Model of Gunger and Winterton

For the model of Gunger and Winterton (1986) [1], the equation giving H is:

Equation 50:

$$h = h_l \left[1 + 3000 \left(\frac{q}{G h_{lg}} \right)^{0.86} + \left(\frac{x}{1-x} \right)^{3/4} \left(\frac{\rho_l}{\rho_g} \right)^{0.41} \right]$$

The value of h_l given by **Equation 49**.

1.3.2.3) Model of Schrock and Grossman

The model of Schrock and Grossman (1959) [1] determines h by:

Equation 51:

$$h = 7.39.10^3 h_l \left[\frac{q}{G h_{lg}} + 0.00015 \left(\frac{1}{X_{tt}} \right)^{0.66} \right]$$

$X_{tt}$is determined by the following equation and h_l always by **the no. 49**.

Equation 52:

$$X = \frac{1-x}{x} \sqrt{\frac{\rho_g f_{pl}}{\rho_l f_{pg}}}$$

1.3.2.4) Model Monophasic

At the Monophasic model level, the equation to solve is:

Equation 53:

$$q = h_l (T_p - T_l(z))$$

T_l corresponding to the temperature of the liquid in our system.

In our case of study, the heating is carried out at imposed flux q. In order to know T_p, we will have to first calculate the value of h_l. A correlation of Nusselt for forced internal flows gives us:

Equation 54:

$$Nu = \frac{h_l D}{\lambda_l} = 0.023\left(\frac{GD}{\mu_l}\right)^{0.8} Pr^{1/3} \text{ Is } h_l = 0.023\frac{\lambda_l}{D}\left(\frac{GD}{\mu_l}\right)^{0.8} Pr^{1/3}$$

A heat balance on the flow of fluid in the cylinder implies:

Equation 55:

$$T_l(z) - T_{le} = \frac{qS_p}{AGC_{pl}}(z - z_e)$$

We can finally calculate the wall temperature:

Equation 56:

$$T_p(z) - T_l(z) = \frac{q}{h_l}$$

2.) MATLAB Program

Our MATLAB program consists of a main program that is responsible for calling the other scripts and consolidating the results.

First, this one imports the geometry and the data of the refrigerant in order to be able to initialize the remainder of the program. The geometry has been divided into several parts: presence or not of a component, elbow, ... Each party has its own specifics.

Once this data is retrieved, we initialize a first program that solves the equations for the vacuum rate, the title, and the pressure. The latter defines which model to use (annular or monophasic model) according to the previous results.

Once these equations are resolved over all geometry, the program calls the scripts that calculate the temperature of the components, one script per model.

These data, as well as the previous ones, are subsequently drawn up and grouped in order to be exploited.

➔ **MATLAB Program**

Determination of Position Z: Presence/absence of elbows, presence/absence of components (activation or not of thermal models), sense of gravity, power of the component.

Refrigerant properties: Change depending on the pressure, the saturation temperature.

Using the annular model without tearing

- **Hydraulic**: Annular model without grubbing
  - Interfacial friction: Model of Wallis
  - Skin friction: Lockhart and Martinelli, Awad, and Baroczy models tested
- **Thermal**
  - Convective models (Kandlikar, Gunger and Winterton or Schrock and Grossman)

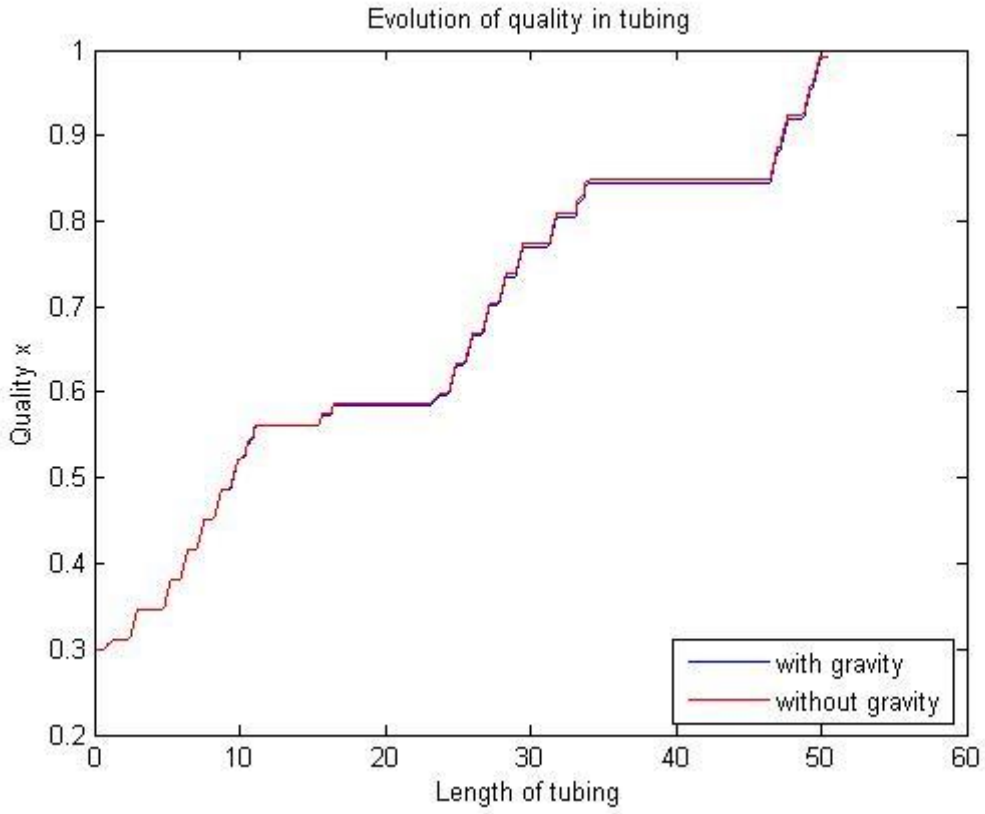
If vapor title > 0.99

- **Hydraulic**: Monophasic model
  - Monophasic type skin friction (negligible)
- **Thermal**: Monophasic model

3.) Presentation of the results

The simulations to determine the different parameters were carried out for two cases: with and without gravity. We can compare the different results obtained. In addition, to simplify the resolution, we have calculated these parameters for only one single tube.

3.1) Title X



This first curve allows us to check the evolution of the title along the evaporator. One of the conditions of good functioning of it, is the disappearance of the liquid phase in its exit. In fact, this steam will then be overheated before radiating into space. If liquid is still present, the compressor will not be able to function properly.

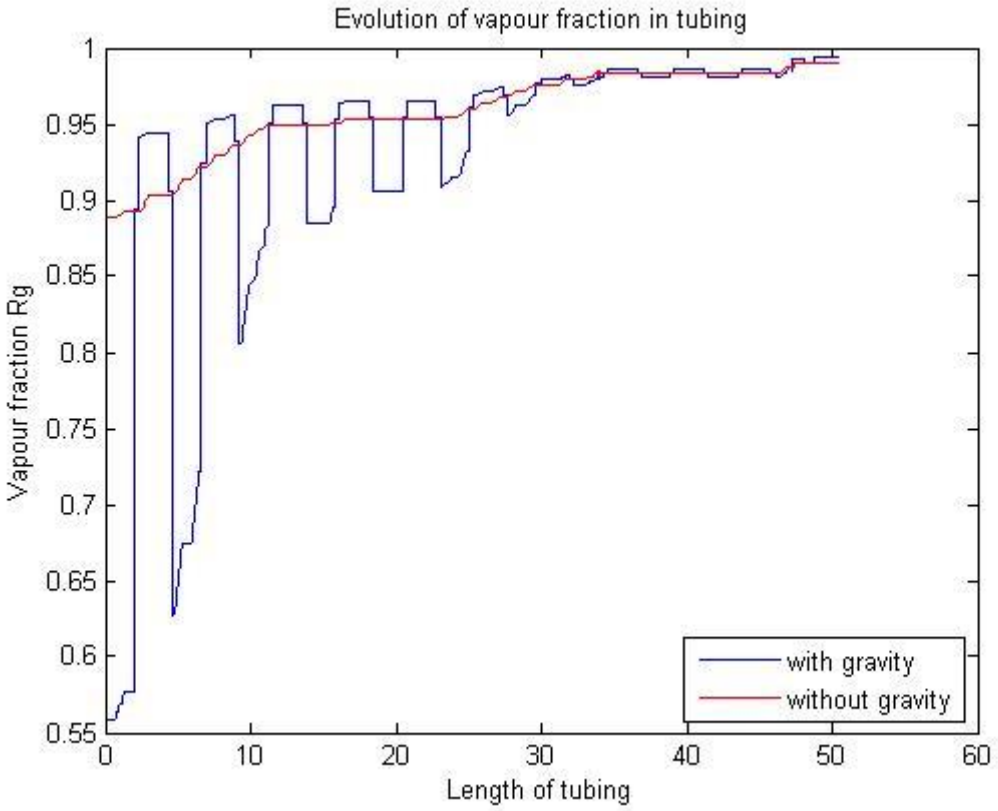
Here, for both cases, the refrigerant is totally in the form of vapor at the end. The condition is filled.

In the program, the condition x = 1 has been replaced by x = 0.99: some equations contain 1/(1-X) factor, which can lead to different problems of continuity and validity of the models. Beyond 0.99, we consider the constant title (the liquid phase is negligible).

The increase of the title corresponds here to the passage of the tube at the level of a component, because the evolution of the title means that the fluid moves from the liquid phase to that of steam.

3.2) Vacuum rate

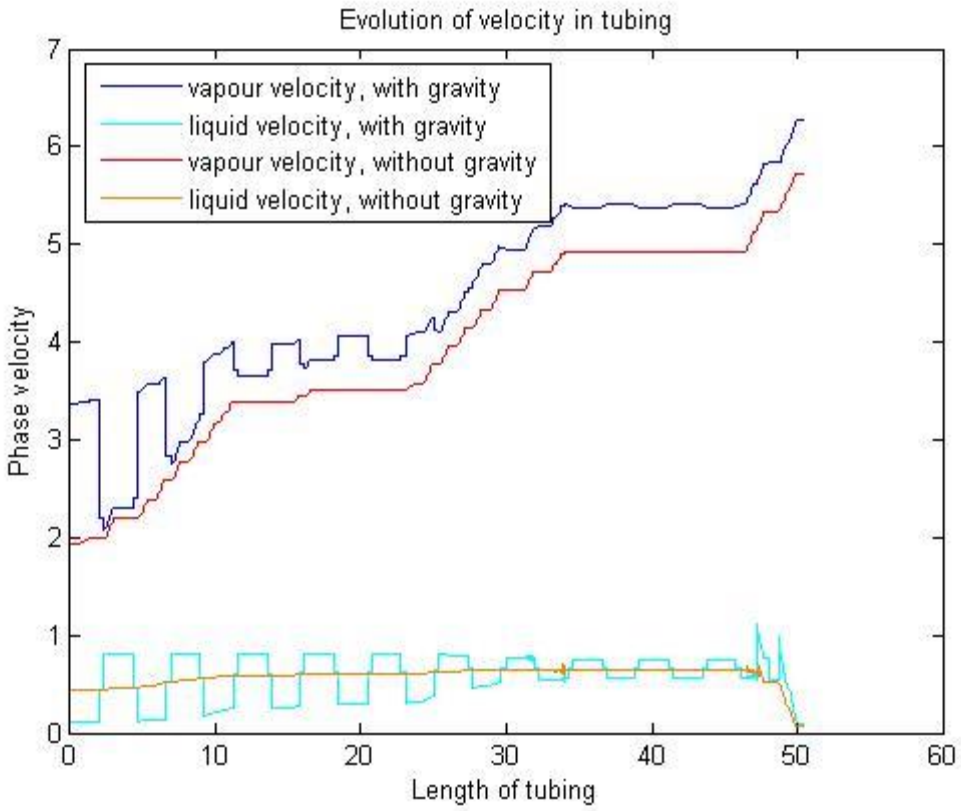
Now that we have looked at the evolution of the title, it may be interesting to compare it to that of the void rate.



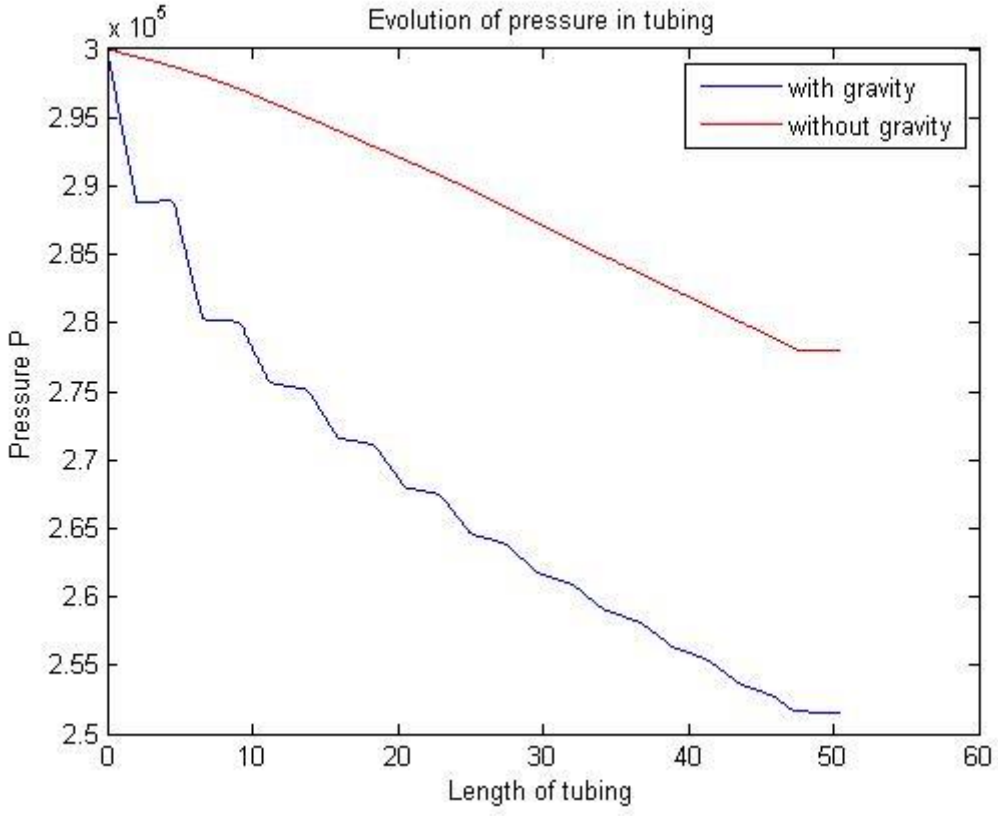
As before, the vacuum rate is limited to 0.99 to avoid different problems. The whole flow will be considered as monophasic beyond. In addition, both cases have well reached the limit of 1 at the end of the evaporator.

The difference between the two cases here is blatant. Indeed, gravity strongly influences the flow in the tubes, especially the liquid phase. The latter is accelerated in the downward tubes and slowed down in the ascending tubes. Because of flow retention, this causes a change in the thickness of the liquid layer, and therefore of the rate R_l: An increase for the ascending tubes, and a decrease for those descendants. So it is normal that all this influences the void rate: $R_g = 1 - R_l$. It is therefore normal that the curve without gravity is 'smoother ', as it is only influenced by the presence of components.

This alternation due to gravity can be observed in the velocities of the two phases.


Evolution of velocity in tubing
vapour velocity, with gravity
liquid velocity, with gravity
vapour velocity, without gravity
liquid velocity, without gravity
Phase velocity
Length of tubing
0
1
2
3
4
5
6
7
0
10
20
30
40
50
60

3.3) Pressure


Evolution of pressure in tubing
x 10^5
with gravity
without gravity
Pressure P
Length of tubing
2.5
2.55
2.6
2.65
2.7
2.75
2.8
2.85
2.9
2.95
3
0
10
20
30
40
50
60

.

Just like the previous data, the pressure inside the tubes is essential in order to calculate correctly the temperature of the components we must cool.

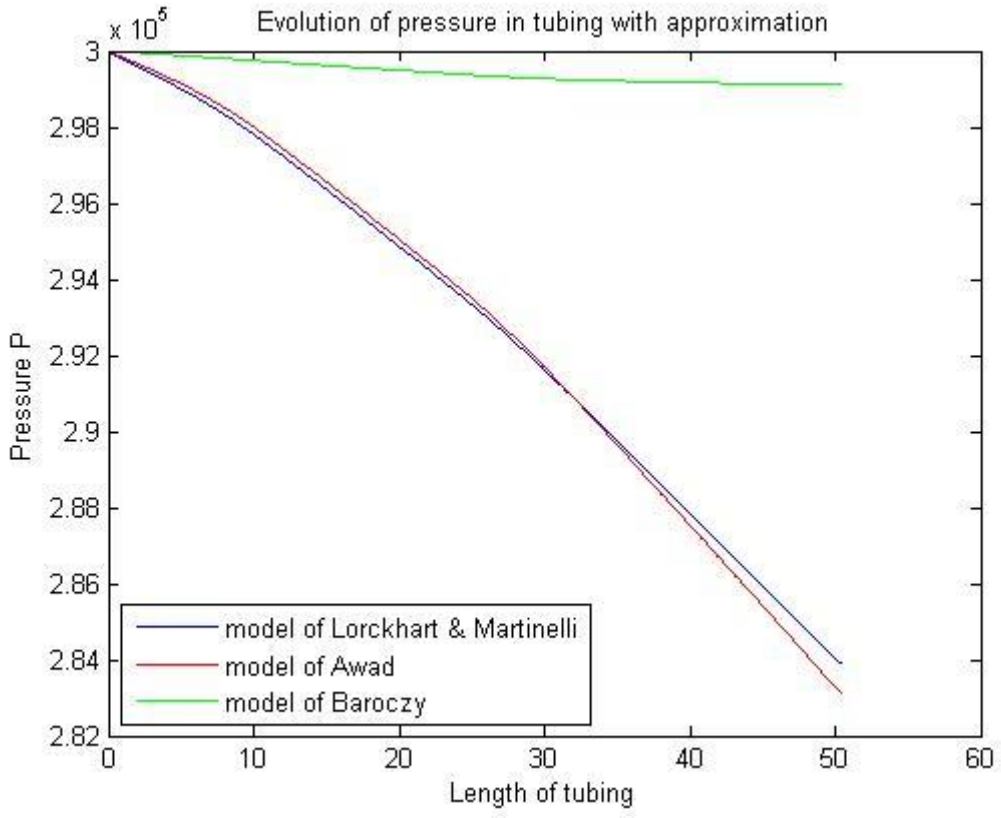
Pressure loss is an important part of our system. Indeed, the refrigerant remains at saturation temperature, even when the pressure changes: if it decreases, the temperature also decreases. However, the difference between the entry and the output for the latter shall not exceed 1 °C. So, we tried to take into account a number of phenomena to be as close to reality as possible. The evolution of pressure on this curve therefore includes the pressure loss in the elbows.

Here, the pressure losses correspond to a temperature drop of 5 °C with gravity, and 2 °C without. This difference is explained with additional pressure losses, due to the friction of the liquid caused by gravity. When switching to the monophasic case, we neglect the gas pressure loss.

The same phenomenon can be observed in the case with gravity: bearings are present, corresponding to an ascending tube, or a downward tube.

3.4) Approximation of pressure losses

The previous curve can be compared to different load loss approximations: Lockhart & Martinelli model, Awad model and Baroczy model.

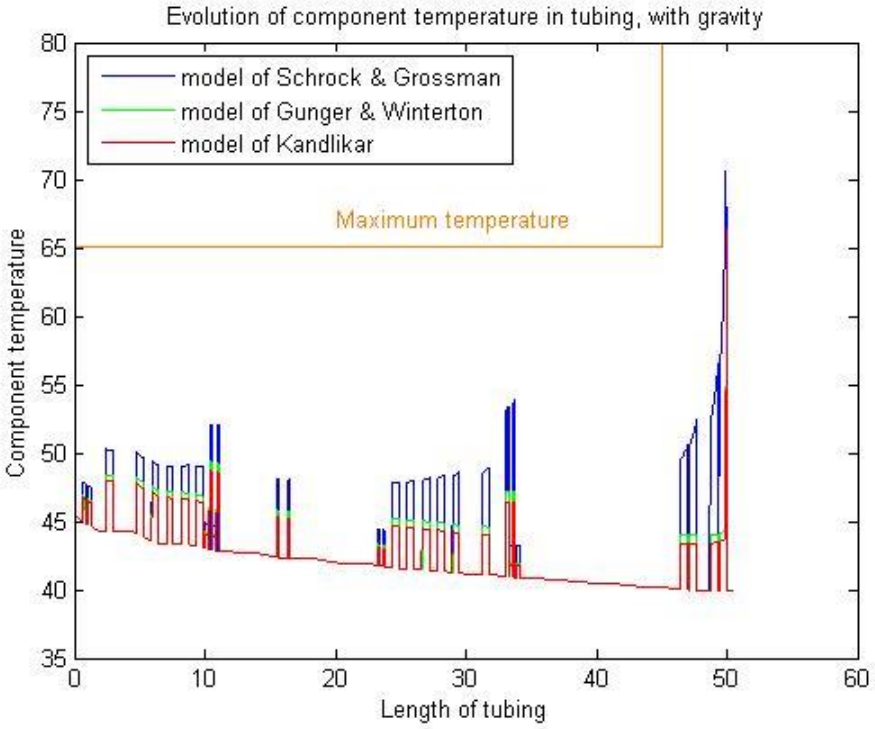


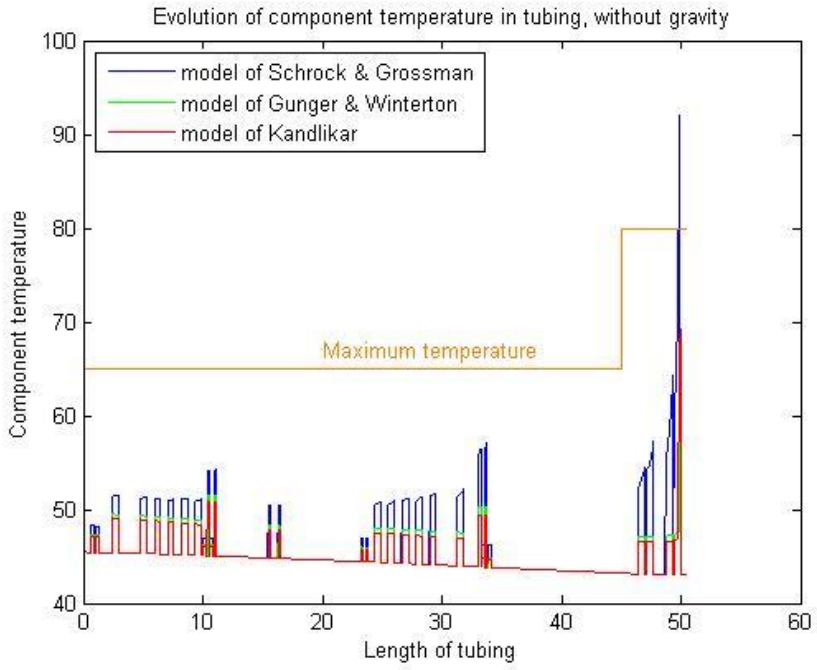
For the models of Lockhart and Martinelli and Awad, we have results fairly close to the case without gravity, which is consistent because this term does not intervene in the models. In addition, the corresponding temperature difference is about 1.6 °C, which confirms our simulations with a more precise model (the singular losses in the elbows were not considered in these approximations).

The Baroczy model does not work well. This is normal because this model is normally used for cases where the flow is more important

3.5) Temperature

The temperature was calculated for the three models: Schrock & Grossman, Gunger & Winterton and Kandlikar. These results allow us to clearly analyze the influence (and its absence) of gravity on the system.





The maximum temperature of the components has been added in order to better situate any problems.

Overall, the components do not reach their critical temperature. However, locally, at the end of the evaporator, in the case without gravity, the temperature can reach 90 °C (instead of the 80 °C supported). This phenomenon is explained by the limit we have taken for the vacuum rate and the title. This definition eventually leads to the beginning of the monophasic zone, and thus to model changes, including for temperature. This is equivalent to warming up the gas temperature locally earlier, and therefore longer, instead of evaporating the little liquid that remains.

In addition, at high vacuum rates, the models used are less effective at predicting heat transfer. By pushing the limit and refining the models, we will arrive at the best prediction of this last temperature.

The various simulations allow us to check the proper functioning of such an evaporator. All that remains is to confirm the theory with ground tests and compare the different points. Particular attention may be given to the last components of the system to check that the temperature limit is not reached.

Conclusion

In the framework this work, we have set up a MATLAB code to model the thermo-hydraulic phenomena in the evaporator of the cooling circuit of a satellite. Studies have taken into account the transition between different models as the flow regime changes in the evaporator. We compared the influence of different models at the level of interfacial friction and thermal. The studies also focused on respect for the geometry provided by the industrial partner. Our modelling of the system is in accordance with the specifications. Indeed, whether at the level of the pressure losses or for the maximum temperature of the components, the various simulations we have carried out are in accordance with the requirements.

Nevertheless, some aspects dealt with in this project can be deepened in further studies. The study of other physical models such as the annular, with consider of the grubbing could be carried out. At thermal level Chen's model could also be tested. In addition, we missed for this study the experimental data that would have allowed us to make a coherent comparison.

Bibliography

[1] Colin C., 2010, *Changement de phase – Ebullition er condensation convectives*,

Cours ENSEEIHT-INPT, Toulouse, France

[2] Collercandy N., Dupouy F. & Thomas G, *Dimensionnement d'un évaporateur pour une boucle diphasique de refroidissement d'un satellite de télécommunication*,

[3] Padilla & Revellin, 2009, *Prediction and simulation of two-phase pressure drop in return bends* ,

International journal of refrigeration 32 (2009) 1776–1783 , ELSEVIER (www.elsevier.com)

[4] P. Vassallo and K. Keller, 2004, *Two-Phase Frictional Pressure Drop Multipliers*

for SUVA R-134a Flowing in a Rectangular Duct, US government

[5] Sawant P., 2009, *Prediction of amount of entrained droplets in vertical annular two-phase flow* ,

International Journal of Heat and Fluid Flow 30 (2009) 715–728 , ELSEVIER (www.elsevier.com)

[6] Pan & Hanratty, *Correlation of entrainment for annular flow in vertical pipes* ,

International Journal of Multiphase Flow 28 (2002) 363–384 , ELSEVIER (www.elsevier.com)

[7] Belt, R.J., 2007, *On the liquid film in inclined annular flow*

[8] Awad & Muzychka , 2010, *Review and modeling of two-phase frictional pressure gradient at microgravity conditions*,

Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer International of ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th Meeting and Conference on Nanochannels, Microchannels, and Minichannels 8th International Conference on Nanochannels, Microchannels, and Minichannels Montreal, Canada