NUMERICAL STUDY OF THE THERMAL BEHAVIOR OF BIMETALLIC TUBES ON A STRATIFIED FLOW PATTERN.

LUIS FERNANDO VÉLEZ VARGAS

Tesis para optar al grado de
Magister en Ciencias de la Ingeniería

Profesor Supervisor:
RODRIGO ESCOBAR MORAGAS

Santiago de Chile, January, 2009
© 2009, Fernando Vélez Vargas
NUMERICAL STUDY OF THE THERMAL BEHAVIOR OF BIMETALLIC TUBES ON A STRATIFIED FLOW PATTERN.

LUIS FERNANDO VÉLEZ VARGAS

Tesis presentada a la Comisión integrada por los profesores:

RODRIGO ESCOBAR
DIEGO CELENTANO
AMADOR GUZMAN
SEBASTIAN RIOS

Para completar las exigencias del grado de Magíster en Ciencias de la Ingeniería

Santiago de Chile, January, 2009
A mis Padres, familia y amigos, que me apoyaron mucho.
ACKNOWLEDGEMENTS

The author greatly acknowledges support from my coworkers and friends on the Department of Mechanical Engineering especially from Alberto Ortega; thanks to his assistance this work could be completed.
# CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEDICATION</td>
<td>II</td>
</tr>
<tr>
<td>ACKNOWLEDGEMENTS</td>
<td>iii</td>
</tr>
<tr>
<td>CONTENTS</td>
<td>iv</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>vii</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>viii</td>
</tr>
<tr>
<td>ABBREVIATIONS</td>
<td>x</td>
</tr>
<tr>
<td>NOMENCLATURE</td>
<td>xi</td>
</tr>
<tr>
<td>GREEK SYMBOLS</td>
<td>xiii</td>
</tr>
<tr>
<td>Resumen</td>
<td>xiv</td>
</tr>
<tr>
<td>Abstract</td>
<td>xv</td>
</tr>
<tr>
<td>1. Introduction</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Chilean energy scenario</td>
<td>2</td>
</tr>
<tr>
<td>1.1.1 Non-renewable energy policy in Chile</td>
<td>3</td>
</tr>
<tr>
<td>1.1.2 Chilean solar energy potential</td>
<td>4</td>
</tr>
<tr>
<td>1.2 Power generating solar energy</td>
<td>5</td>
</tr>
<tr>
<td>1.2.1 Concentrated solar power technology</td>
<td>6</td>
</tr>
<tr>
<td>1.2.2 Parabolic trough based solar thermal power generation</td>
<td>10</td>
</tr>
<tr>
<td>1.3 Work motivation</td>
<td>27</td>
</tr>
<tr>
<td>1.4 Summary</td>
<td>28</td>
</tr>
<tr>
<td>2. Problem definition and thesis objectives</td>
<td>29</td>
</tr>
<tr>
<td>2.1 Approaches to the process of steam generation inside smooth pipes</td>
<td>30</td>
</tr>
<tr>
<td>2.1.1 Boiling inside horizontal pipes</td>
<td>32</td>
</tr>
<tr>
<td>2.1.2 Methods of analysis</td>
<td>32</td>
</tr>
<tr>
<td>2.2 Objectives</td>
<td>39</td>
</tr>
</tbody>
</table>
3. Methodology ......................................................................................................................... 41
   3.1 Simulation process through computational fluid dynamics ................................................. 41
   3.2 Applying CFD to steam generation inside a horizontal pipe ............................................. 44
      3.2.1 Governing equations for the single phase problem ..................................................... 44
      3.2.2 Boundary conditions for the single phase problem ..................................................... 50
      3.2.3 Equations treatment and calculation of the solution .................................................... 52
      3.2.4 Temporal dependence of the solution .......................................................................... 53
      3.2.5 Convergence criteria for the single phase problem .................................................... 54
      3.2.6 Applying CFD to the steam generation problem .......................................................... 55
   3.3 Single phase flow simulations setup ..................................................................................... 63
      3.3.1 Sub cooled liquid section ............................................................................................. 63
      3.3.2 Superheated steam section ............................................................................................ 69
      3.3.3 Convergence criteria for single phase simulations ....................................................... 69
   3.4 Two phase stratified flow simulation setup ........................................................................ 70
      3.4.1 Dynamics setup for stratified flow ............................................................................... 73
      3.4.2 Setup of boundary conditions for the fluid domain ....................................................... 76
      3.4.3 Boundary conditions setup for the solid domain .......................................................... 77
      3.4.4 Initial conditions setup ................................................................................................ 77
      3.4.5 Calculation procedure .................................................................................................. 78
      3.4.6 Convergence criteria for the multiphase simulations ................................................... 78
   3.5 Additional requirements of the simulations ......................................................................... 79
4. Simulation results and discussion ............................................................................................. 80
   4.1 Single phase simulation results – sub cooled liquid ............................................................ 80
      4.1.1 Base case analysis ......................................................................................................... 81
      4.1.2 Comparison of the bimetallic results against the base case ......................................... 85
   4.2 Single phase simulation results – superheated steam .......................................................... 92
      4.2.1 Base case analysis ......................................................................................................... 92
      4.2.2 Comparison of the bimetallic results against the base case ......................................... 95
   4.3 Two-phase simulation results – stratified flow ................................................................. 100
      4.3.1 Base case analysis ......................................................................................................... 100
      4.3.2 Comparison between base case and bimetallic absorbers ......................................... 107
   4.4 Summary of the results obtained ....................................................................................... 115
5. Conclusions ............................................................................................................................. 118
LIST OF TABLES

Table 1-1: CSP Plant capital cost breakdowns. .......................................................... 12
Table 1-2: Annual CSP O&M cost breakdowns. ......................................................... 12
Table 1-3: Collector field inlet and outlet parameters .................................................. 21
Table 3-1: Absorber tube dimensions ........................................................................... 64
Table 3-2: Mesh details for each case, single phase modeling ........................................ 65
Table 3-3: Absorber tube dimensions (two phase case) .................................................. 72
Table 3-4: Mesh details for each case two phase modeling ............................................ 73
Table 4-1: Steam volume fraction at half distance from the inlet .................................. 108
LIST OF FIGURES

Figure 1-1 Primary energy consumption in Chile, 2006 .................................................. 3
Figure 1-2: Annual average of solar irradiation for Chile .................................................... 5
Figure 1-3: Parabolic dish - Stirling engine system .......................................................... 7
Figure 1-4: Aerial view PS-10 plant, Spain ................................................................. 8
Figure 1-5: Solar parabolic trough .................................................................................. 9
Figure 1-6: Absorber tube structure .............................................................................. 9
Figure 1-7: Parabolic trough plant, aerial view, Kramer Junction ................................... 10
Figure 1-8: Solar power plant with thermal storage and fossil backup ........................... 13
Figure 1-9: Dispatched electricity using a thermal storage system .................................. 14
Figure 1-10: Combined cycle with solar field integration ............................................... 15
Figure 1-11: Direct steam generation solar power plant design ...................................... 16
Figure 1-12: Operation diagram, one through mode ..................................................... 18
Figure 1-13: Operation diagram, recirculation mode ..................................................... 19
Figure 1-14: Operation diagram, injection mode ......................................................... 19
Figure 1-15: Collectors field, simplified diagram for the INDITEP project ....................... 20
Figure 1-16: Relation between pressure and mass flow for steam-liquid flow patterns...... 23
Figure 1-17: Microchannels and porous coatings designs for absorber tubes ............... 25
Figure 1-18: Bimetallic absorber (copper-steel) prototype ........................................... 26
Figure 2-1: Flow patterns for boiling in a vertically heated tube .................................... 30
Figure 2-2: Flow patterns for a horizontal heated pipe ............................................... 32
Figure 2-3: Flow pattern map for n-butane on a horizontal pipe ................................. 38
Figure 3-1: Sketch of a control volume ....................................................................... 52
Figure 3-2: Components of a mesh element ............................................................... 53
Figure 3-3: Interphase heat transfer ............................................................................. 61
Figure 3-4: Single phase problem’s domain ............................................................... 63
Figure 3-5: Single phase case three dimensional mesh .............................................. 65
Figure 3-6: Reference angle for irradiation profile ..................................................... 66
Figure 3-7: Irradiation profile .................................................................................. 67
Figure 3-8: Boundary conditions ............................................................................. 67
Figure 3-9: Two-phase case three dimensional mesh ................................................. 72
Figure 3-10: Relation between Nusselt number and steam mass fraction ..................... 76
Figure 4-1: Heat flux profile .................................................................................. 81
Figure 4-2: Velocity vector plot (64 bar) ................................................................ 82
Figure 4-3: Curves defined for data sampling ............................................................. 82
Figure 4-4: Velocity plots for subcooled liquid (64 bar) ............................................. 83
Figure 4-5: Temperature Plots for subcooled liquid (64 bar) ..................................... 84
Figure 4-6: Temperature contour plots (98 bar and 64 bar) ....................................... 85
Figure 4-7: Maximum temperature difference, subcooled liquid ............................... 87
Figure 4-8: Cross section temperature contour plots for the 70/30 case ..................... 88
Figure 4-9: Cross section temperature contour plots for the 50/50 case ..................... 88
Figure 4-10: Cross section temperature contour plots for the 30/70 case .................... 89
Figure 4-11: Set of temperature profile plots (external surface), sub cooled liquid
Figure 4-12: Maximum temperature difference in solid domain (subcooled liquid)
Figure 4-13: Velocity profile per cross section (64 bar)
Figure 4-14: Temperature profile per cross section (64 bar)
Figure 4-15: Temperature contour plot for base case (solid domain) (98 bar and 64 bar)
Figure 4-16: Maximum temperature difference, superheated steam
Figure 4-17: Cross section temperature contour plots for the 70/30 case
Figure 4-18: Cross section temperature contour plots for the 50/50 case
Figure 4-19: Cross section temperature contour plots for the 30/70 case
Figure 4-20: Set of temperature profile plots (external surface), superheated steam
Figure 4-21: Maximum temperature difference at external pipe surface (superheated steam)
Figure 4-22: Liquid superficial velocity vector plot (at symmetry plane)
Figure 4-23: Liquid superficial velocity vector plot (at plane perpendicular to symmetry plane)
Figure 4-24: Superficial velocity vector plot (tangential projection)
Figure 4-25: Superficial velocity vector plot (at outlet, normal projection)
Figure 4-26a-b: Temperature contour plot at cross sections
Figure 4-27a-b: Temperature contour plot, internal surface with cross sections
Figure 4-28a-b: Temperature contour plot, absorber external surface (all domains)
Figure 4-29: Temperature contour plot, absorber internal surface (all domains)
Figure 4-30: Set of temperature profile plots (external surface)
Figure 4-31: Maximum temperature difference per section (external surface)
Figure 4-32: Temperature distribution on the periphery of an absorber
ABBREVIATIONS

CFD  Computational fluid dynamics
CSP  Concentrated solar power
DISS Direct solar steam
EbFVM Element-based finite volume method
FVM  Finite volume method
IAPWS International association for the properties of water and steam
RMS  Root mean square
SEGS Solar energy generating systems
NOMENCLATURE

$A_{\alpha\beta}$  Interphase area density [1/m]
$A_{EXT}$  Pipe external surface’s area [m]
$B_D$  Boiling number
$C_D$  Drag force correlation’s constant
$C_D$  Constant for Shah’s correlation
$C_{\epsilon 1}$  $k - \epsilon$ model constant
$C_{\epsilon 2}$  $k - \epsilon$ model constant
$C_\mu$  $k - \epsilon$ model constant
$c(a)$  Drag coefficient for a pair of phases $\alpha\beta$ [kg/s]
$c_{\alpha\beta}$  Drag force correlation’s constant
$c_p$  Constant pressure heat capacity [J/(kg K)]
$D$  Pipe’s diameter [m]
$D_i, d_i$  Pipe’s internal diameter [m]
$d_{\alpha\beta}$  Interphase length scale [m]
$E$  Convective factor
$e$  Internal energy [kJ/kg]
$F_S$  Shah’s constant
$F_{L_f}$  Liquid Froude’s number
$F_{D,\alpha}$  Drag force for phase $\alpha$ [N]
$G$  Mass flow velocity [kg/(m² s)]
$g$  Constant of gravity [m/s²]
$H_{tot}$  Total enthalpy [J/kg]
$H_{\alpha\delta}$  Enthalpy carried from phase $\alpha$ to interphase [J/kg]
$H_{FG}$  Enthalpy of vaporization [J/kg]
$H_{IN}$  Enthalpy at domain’s inlet [J/kg]
$H_L$  Liquid phase heat transfer coefficient [W/(m² K)]
$H_{TP}$  Lcal two-phase heat transfer coefficient [W/(m² K)]
$h_{cb}$  Convective boiling heat transfer coefficient [W/(m² K)]
$h_{nb}$  Nucleate boiling heat transfer coefficient [W/(m² K)]
$h_{\alpha}$  Heat transfer coefficient for phase $\alpha$ [W/(m² K)]
$h_{\alpha\beta}$  Heat transfer coefficient between phases $\alpha$ and $\beta$ [W/(m² K)].
$h_{\beta}$  Heat transfer coefficient for phase $\beta$ [W/(m² K)]
$h_{L_{sat}}$  Liquid’s enthalpy at saturation condition [kJ/kg]
$I$  Turbulent intensity
$k$  Turbulent kinetic energy [J/kg]
$k_{in}$  Turbulent kinetic energy at inlet [J/kg]
$L$  Latent heat of vaporization [kJ/kg]
$L_{scale}$  Length scale [m]
$M_{\alpha}$  Sum of forces on the interphase acting on the $\alpha$ phase [N]
$m_{\alpha}$  Water mass flow (liquid state) [kg/s]
mass flow at inlet [kg/s]
mass flow of water (gas state) [kg/s]
Mass flow to phase α from phase β [kg/s]
Constant for Shah’s correlation
Nusselt number for phase α
Nusselt number for phase β
Vector normal to symmetry plane
Turbulence production [J/s]
Buoyancy turbulence production [J/s]
Liquid phase Prandtl’s number
Turbulent Prandtl’s number
Pressure [kPa]
Reduced pressure
Saturation pressure [kPa]
Adective energy flux at inlet [W]
Interphase heat flux to phase alpha from other phases [W]
Local flow of heat from the wall of the tube to the fluid [W/kg]
External heat flux per area unit [W/m²]
Flow of sensible heat towards phase α from the interphase [W]
Flow of sensible heat towards phase β from the interphase [W]
Diameter based Reynolds number
Liquid phase Reynolds number
Volume fraction for phase alpha
Boiling factor
Source vector of external work [N]
Source vector of momentum [N]
Temperature [K]
Saturation temperature at local saturation pressure [K]
Temperature of the wall in a certain axial position [K]
Time [s]
Velocity vector [m/s]
Time-averaged velocity vector [m/s]
Time variable velocity vector [m/s]
Domain’s total volume [m³]
Martinelli parameter
Steam mass fraction
GREEK SYMBOLS

\( \lambda_L \)  Liquid thermal conductivity [W/(m K)]
\( \rho_{\alpha \beta} \)  Volume fraction averaged density [kg/m\(^3\)]
\( \varphi \)  Reference angle for heat flux distribution [rad]
\( \sigma \)  Surface tension coefficient [N/m]
\( \alpha_k \)  \( k - \varepsilon \) model constant
\( \varepsilon \)  Vortexes’ dissipation [J/(kg s)]
\( \alpha \)  Fluid phase descriptor (subindex)
\( \varepsilon_{in} \)  Vortexes’ dissipation at inlet [J/(kg s)]
\( \Gamma_{\alpha \beta} \)  Mass flow per unit of volume form phase \( \beta \) towards phase \( \alpha \) [kg/s]
\( \Gamma_{\beta \alpha} \)  Mass flow per unit of volume form phase \( \alpha \) towards phase \( \beta \) [kg/s]
\( \rho_{ref} \)  Reference density [kg/m\(^3\)]
\( \mu_t \)  Turbulent viscosity [Pa s]
\( \phi \)  Generic variable
\( \Gamma \)  Diffusivity
\( \Gamma_t \)  Eddy diffusivity [Pa s]
\( \delta \)  Kroenecker’s delta
\( \tau \)  Deformation stress vector [N]
\( \mu \)  Dynamic viscosity [Pa s]
\( \rho_L \)  Liquid phase density [kg/m\(^3\)]
\( \rho_G \)  Gaseous phase density [kg/m\(^3\)]
**RESUMEN**

En el diseño de plantas solares térmicas con generación directa de vapor en el tubo absorbedor del colector de canal parabólico, la eficiencia está vinculada directamente al desempeño de este componente. Optimizar la transferencia de calor en el absorbedor se traduce en estructuras más cortas, generando la misma cantidad de vapor. Por lo tanto, es posible proponer estructuras compuestas para el absorbedor tubos construidos de más de un material, donde la disposición de cada uno toma ventaja de propiedades específicas, al estudiar las características termodinámicas del flujo, especialmente los procesos de cambio de fase (la ebullición al interior del tubo del líquido que fluye a través del campo de colectores). En esta investigación se presentan los resultados de la simulación numérica de ebullición convectiva en tubos horizontales. Se prescribe un flujo de calor en la superficie externa del absorbedor generando estratificación líquido-vapor. Se aplicaron herramientas de dinámica de fluidos computacional para la modelación de la física a través del programa ANSYS CFX-11, basado en la técnica de volúmenes finitos basados en elementos. Adicionalmente a la simulación de cambio de fase, se realizaron también simulaciones para líquido sub enfriado y vapor sobrecalentado. La metodología se repitió para tubos compuestos de cobre y acero en diferentes proporciones. Los resultados muestran la ventaja de utilizar absorbedores compuestos al reducir los gradientes de temperatura en la sección transversal, un parámetro crítico del desempeño del absorbedor. Las simulaciones muestran también que la efectividad del cobre en la pared del tubo decrece con mayores proporciones, logrando menores reducciones de temperatura con adiciones de cobre superiores al 30 %. Esto ocurre a expensas de la resistencia estructural. Se considera que aunque la metodología de simulación de cambio de fase en patrón estratificado se encuentra en un primer nivel de desarrollo, la metodología aquí presentada constituye una herramienta valiosa en procesos de diseño relativos a la transferencia de calor y conversión de energía.

Palabras Claves: generación directa de vapor, dinámica de fluidos computacional, cambio de fase térmico, flujo estratificado.
ABSTRACT

In the design of solar thermal power plants with direct steam generation, the heat collecting efficiency is directly linked to the performance of the absorber tube. Optimizing the heat transfer mechanisms translates into shorter structures, while generating the same amount of steam. Thus, it is possible to propose composed configuration for the absorber tube (this is, tubes made from various materials, where the disposition takes advantage of each material properties), by studying the thermodynamic characteristics of the flow, especially the phase change processes. Herein, the first results of a numerical study of the convective boiling process in horizontal tubes are presented. The study features a prescribed heat flux on the surface of the tube, modeled with ANSYS CFX-11, which uses the element based finite volume approach for three dimensional geometries. Aside from the stratified liquid-steam region, sub cooled liquid and superheated steam segments were also simulated. The results for the three sections simulated delivered information about the advantages of using bimetallic absorbers, by reducing the maximum temperature difference on the external surface, a critical parameter for the performance of these structures. Although, the simulations showed that the effectiveness of the addition of copper decreases: the temperature reduction becomes less effective when more copper is added, at expenses of structural resistance, especially in for two-phase flow, temperature reduction wasn’t significant for others structures with a fraction of copper in the wall over 30%.

It is considered that though this in a first level of development, the methodology for simulating stratified flows due to thermal phase change here presented is a very valuable tool in design process related to heat exchange and energy conversion.

Keywords: direct steam generation, CFD, thermal phase change, stratified flow.
1. INTRODUCTION

Nowadays, mankind faces enormous challenges for development, like never before in history. The reports on global warming released by the Intergovernmental Panel on Climatic Change (IPCC, 2008) account for the impact of this phenomenon in events such as polar ice melting, increases in the ocean level and temperature rising happening all across the globe, and are related to the increase of greenhouse gases from industrial activity. Their root is the intensive use of fossil fuels in energy conversion processes.

The projected impact on ecosystems is such that, in the absence of profound and sustained measures in order to change the way society satisfies its energetic needs, we will be forced to deal with serious disorders.

From a risk assessment perspective, it is mandatory to carry out significant efforts to alleviate the potential impact of global warming on the future, according to the Stern report on economic impact of global warming (Stern et al, 2006).

At the same time, we have to deal with growing uncertainties in the energetic markets. Prices of fossil fuels such as oil and natural gas present an upward trend, jeopardizing the chances of development of poorer countries if they can’t meet their energetic demand. Alas, energy consumption, especially in developing economies it’s closely related to economic growth (Dorian, Franssen, & Simbeck, 2006).

One of the main conclusions of IPCC’s work is that in order to limit the impact of high concentration of greenhouse gases in the ecosystem, we must design a future scenario where power generation by means of renewable sources of energy has a significant role. Moreover, from an economic point of view, it is possible to diminish the relative scarcity of fossil fuels (along with the price of the energy generated) using renewable sources.
1.1 Chilean energy scenario

At present time, electricity generation in Chile is achieved mainly through the use of hydroelectric facilities and fossil fuel based thermoelectric plants (CNE, 2007): in 2007, their share of total power generated was 38 and 62 percent, respectively. Chilean indigenous fossil fuel production is practically irrelevant; because of this, economic growth increases the dependence on energetic imports from 18 percent in 1982 up to 70% in 2006.

In the Chilean energy scenario, both hydroelectric generated and fossil based power are related to a supply security uncertainty: A sufficient level of water resources depends on the amount of rain and the occurrence of non-periodic droughts. Argentina, Chile’s main natural gas supplier has reduced the amount of gas sold, forcing the power generating industry to switch to alternatives of higher costs, such as diesel.

This complex scenario emerges after a long period of cheap electricity generation, when the nation’s demand doubled in fifteen years and there was a 30 percent increase on the electric consumption per unit of gross product (CNE, 2007).

Thus, there is a need of diversifying the energetic matrix, either by balancing the share of existing technologies or by the inclusion of new ones into the mix, in order to protect the economy against fuels price fluctuations and supply shortages. It is expected that this policy (energy matrix diversification) assures energy supply.

Against this disadvantageous scenario, non conventional renewable energies sources such as wind, geothermal, solar or biofuels are included in the design of public policies tending to guarantee energetic security.
Currently, the ‘traditional’ renewable energies’ share in the Chilean energy matrix is 24 percent, comprising biomass (mainly firewood) and hydroelectricity. Renewable sources’ share is, in practice, nonexistent. In 2005, mini-hydro (under 20 megawatts), biomass and wind sources only reached 2.4 percent of a 11.9 MW total of installed capacity.

It is assumed that Chile has a relatively abundant potential of non-renewable sources, mostly geothermal, eolic and solar (CNE, 2007). Nevertheless, considering seriously these sources as an alternative for power generation, at large scales, implies a thorough cadastre of each one of the resources, which is inexistent until today.

1.1.1 Non-renewable energy policy in Chile

In regards of integrating non-renewable energy into the Chilean energy matrix, there are measures devised with attempting to promote the development and deployment of new generation projects.

President Bachelet’s government proposed that by 2010, 15 percent of the newly installed generation capacity should be based in renewable resources other than...
hydroelectric (CNE, 2007). In addition to that, a bill was passed to the parliament which stated that energy generators should provide at least 5% of the energy by renewable resources. It also proposes the removal of entry barriers for renewable resources generating plants up to 20 MW rated power (Comisión de Minería y Energía, 2006). This limit seems to be disadvantageous, because optimum plant size varies with each technology; some technologies have optimum sizes far greater than this limit, resulting in an excluding measure for some renewable resources. On the same issue, the experience in California shows limits four times the proposed, set in 80 MW since the 80’s decade (Aabakken, 2006).

There is not a single answer for the solution of energy supply through renewable resources. They are inherently irregular; which implies that we can’t rely in one source only. The concept behind a sustainable energy matrix states a sources and conversion technologies portfolio such that energy supply capabilities will not be compromised in the future. Therefore, planning strategies for a transition towards a sustainable energy matrix consider a mix of technologies, rather than a ‘unique source’ solution.

1.1.2 Chilean solar energy potential

Chile is located in the so called ‘solar belt’, an Earth’s region where there are the higher levels of solar radiation, between latitudes 15°N, and 35°N (Acra, Raffoul, & Karahagopian, 1984). The magnitude can be as high as 6 kWh/m² for the annual average (Ortega, Escobar, & Colle, 2008). Integration of solar energy into the matrix shows very promising, given that generating costs for this kind of technology decrease with higher radiation levels (Price et al, 2002). This resource can be used in order to satisfy the growing demand for electricity at most northern areas of the country, mainly because of the abundance both the solar resource (as depicted in Figure 1-2) and the usable land for the deployment of the plants. At the same time, this area concentrates large mining operations, demanding the greatest share of the power generated at SING (northern interconnected system).
Solar thermal technology does not compete for land use if it is applied to the generation of power; regions suitable for installation are often unsuitable for crops or high population density settlements since usually they are located in deserted or arid regions, as it has been for such operating facilities in the United States for over 20 years (Stoddard, Abiecunas, & O'Connell, 2006).

### 1.2 Power generating solar energy

Solar energy is especially appealing for meeting a growing energy demand, mainly due to the inexhaustible nature of its source, the Sun. At the same time, during operation, solar technology does not pollute the environment. As of today, there are several applications for solar energy. Photovoltaic technology allows the direct conversion from solar radiation into electric energy by means of materials excited by photons, generating...
a flow of electrons. In thermal solar plants, solar radiation is absorbed as thermal energy and finally converted into electricity, usually working on the base of traditional thermodynamic cycles.

1.2.1 Concentrated solar power technology

This technology is based in the concentration of solar radiation over a particular region (such as a plate or a tube) by factors between 50 and 10,000 times the direct radiation, in order to produce high temperature thermal energy, which is later transformed into electric power (Aabakken, 2006).

In Concentrated Solar Power systems (CSP), mirrors focus radiation on a receiver, which carries a working fluid on the inside that raises its temperature in order to use that thermal energy to power conversion machines (vapor based turbines, Stirling engines). Alternatively, concentrated radiation can also be converted directly to electricity through photovoltaic cell arrays. Other high temperature heat industrial process can take advantage of these technologies also. The operation limit of these systems is defined by the maximum operating temperature at the solar energy conversion device (European Commission, 2004). In particular, heat for generating electric power is about 300°C. In order to achieve high temperatures, the most suitable technologies are: concentrating dish, solar tower and parabolic trough.

a) Stirling Dish system

A parabolic dish-shaped reflector powers a Stirling engine mounted at the focal point of the dish. Modular units, with outputs ranged between 7 and 25 kW are designed to deliver electricity particularly in remote zones. These systems have the highest solar-to-electric efficiency of all solar conversion technologies, over 30%. However, its cons are a low reliability and high cost per kilowatt: the levelized cost estimation is 54 dollar cents per electric kilowatt according to short term estimations (Bemis & DoAngelis, 2007)
b) Solar Tower

Through multiple heliostats, solar radiation is focused on a central receiver installed at the top of a tower, where the thermal energy collected is absorbed by the working fluid. Later on, that energy is used in a thermodynamic cycle. Given the attributes of the fluid (high heat capacity), overnight thermal storage is feasible. This kind of design rated electric power varies between 30 and 200 MW of electricity. Along with parabolic trough, solar tower is the only technology that has been commercially deployed at PS10 plant, opened 2007 in Andalucía, Spain. In 55 ha, 624 heliostats concentrate radiation into a water steam boiler placed at the central tower’s top. It also counts with steam thermal storage systems, capable of generating 11 MW of electricity (European Commission, 2005).
c) Solar Parabolic Trough

Solar radiation is focused on a linear tubular receiver, coated with minimized emissivity materials and encapsulated in a glass envelope, in order to reduce radiation and convective losses. On the interior of the absorber tube flows a high heat capacity working fluid, which thanks to concentrating factors of about 80 times (Quaschning, 2003) reaches temperatures over 400 °C. Afterwards, the collected thermal energy is transferred to water in heat exchangers. Finally, the generated steam drives a turbine-generator set. These plants capacities range between 10 and 280 MW of electric power (Aabakken, 2006, Abengoa Solar, 2008).
These systems share the same design principle of traditional alternatives for thermal based electricity generation. However, for solar power technologies, fossil fueled steam boilers are switched over a heat exchanger where the working fluid in the solar structure delivers heat gained through collection of solar radiation.
1.2.2 Parabolic trough based solar thermal power generation

First parabolic trough solar fields were used for supplying heat for thermal processes in the decades of 1970 and 1980. Chile was one of the first countries that adopted this technology, at a plant located in Antofagasta in 1972 (CNE, 2000).

Later on, in 1983 in California, South California Edison Company signs the first agreement with Luz International Limited with the purpose of buying power from their solar power plants (SEGS I & II), where all the heat collected in the solar field is used in a Rankine power cycle (Price & Kearney, 1999). The legislation of those times made possible the construction of nine more plants, reaching a total power of 354 MW, efficiently competing in power market between 1974 and 1990. Eventually, Luz would declare bankruptcy due to the failure in getting financial support for the construction of a tenth plant, due basically to legislation changes and a decreasing costs electric power market along with incentives for traditional sources (such as oil and gas). Despite Luz’s demise, existing plants keep on functioning up until today as independent endeavors, managed independently.

Figure 1-7: Parabolic trough plant, aerial view. Kramer Junction. (Solar Millenium AG, 2007)
The experience gained through the SEGS plants (solar electric generating systems) demonstrated the viability of solar thermal technology, reducing the risk of such investments (Price & Kearney, 1999). Thanks to cost reduction potential and technology improvements, in 1992 Kramer Junction (SEGS plant III through VII) joined Sandia National Labs in a research and development project aimed to reduce operating and labor costs. The overall drop in costs (taking the original operating scheme of SEGS plants as a reference was of 40 %), saving the operators 50 million dollars (Aabakken, 2006).

There are several projects of this kind at various stages of deployment in Europe (Spain, Greece, and Italy), Africa (Morocco, Egypt, and South Africa), India (funding for solar thermal power plants were approved in 1996; still in a experimental stage) and in the United States; often supported by international programs of development, such as the International Monetary Fund (German Aerospace Center, 2005, Stoddard et al, 2006).

In 1998 a 'roadmap' was outlined to ensure the competitiveness of solar thermal technology compared to traditional fossil power generation It identified areas where research efforts should be placed, on the attempt of reducing generating costs and increasing reliability of the integrated plant systems (Price & Kearney, 1999).

At present, SEGS plants generating costs are about 10-12 dollar cents per kWh (Aabakken, 2006). It has been anticipated that this value should be lower in next generation plants, reaching 6 dollar cents per kWh within a decade.

While the increase on the size of the plant represents the most direct cost reduction, optimization of its components, such as the support structure of the pipeline, increased reflectivity materials for the mirror, development of surface coatings resistant to higher temperatures and improving the properties of the working fluid are also key issues to make this technology cost competitive with traditional alternatives.
Table 1-1: CSP Plant capital cost breakdowns projected for 2007 (thousands of 2005 dollars) (Stoddard et al, 2006).

<table>
<thead>
<tr>
<th>Item</th>
<th>Cost</th>
</tr>
</thead>
<tbody>
<tr>
<td>Site Work and Infrastructure</td>
<td>2.455</td>
</tr>
<tr>
<td>Solar Field</td>
<td>205.109</td>
</tr>
<tr>
<td>HTF System</td>
<td>9.895</td>
</tr>
<tr>
<td>Thermal Energy Storage</td>
<td>57.957</td>
</tr>
<tr>
<td>Power Block</td>
<td>38.754</td>
</tr>
<tr>
<td>Balance of Plant</td>
<td>22.533</td>
</tr>
<tr>
<td>Contingency</td>
<td>33.742</td>
</tr>
<tr>
<td><strong>Total Direct Costs</strong></td>
<td>393.28</td>
</tr>
<tr>
<td>Indirects</td>
<td>113.469</td>
</tr>
<tr>
<td><strong>Total Installed Cost</strong></td>
<td>494.386</td>
</tr>
</tbody>
</table>

Table 1-2: Annual CSP O&M cost breakdowns projected for 2007 (thousands of 2005 dollars) (Stoddard et al, 2006).

<table>
<thead>
<tr>
<th>Item</th>
<th>Cost</th>
</tr>
</thead>
<tbody>
<tr>
<td>Labor</td>
<td>528</td>
</tr>
<tr>
<td>Administration</td>
<td>979</td>
</tr>
<tr>
<td>Operations</td>
<td>633</td>
</tr>
<tr>
<td>Maintenance</td>
<td>3.018</td>
</tr>
<tr>
<td><strong>Total Labor</strong></td>
<td>419</td>
</tr>
<tr>
<td>Miscellaneous</td>
<td>263</td>
</tr>
<tr>
<td>Service Contracts</td>
<td>260</td>
</tr>
<tr>
<td>Water Treatment</td>
<td>669</td>
</tr>
<tr>
<td>Spares and Equipment</td>
<td>1.859</td>
</tr>
<tr>
<td>Solar Field Parts and Materials</td>
<td>226</td>
</tr>
<tr>
<td>Annual Capital Equipment</td>
<td>3.695</td>
</tr>
<tr>
<td><strong>Subtotal</strong></td>
<td>6.713</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td>528</td>
</tr>
</tbody>
</table>
a) CSP plant characteristics

The solar field is made from multiple collectors arranged in parallel rows, aligned in north south direction. A monitoring system consisting of automatically-controlled engines are responsible for maintaining the radiation focused on the absorber element throughout the day, because in these kind of systems only the direct component of solar radiation can be concentrated.

![Diagram of solar power plant with thermal storage and fossil backup](image)

Figure 1-8: Solar power plant with thermal storage and fossil backup (adapted from Flagg Solar International)

The typical structure of a solar power generation plant includes the collectors’ field, where the outflow of the collector rows is connected to heat exchangers that deliver the energy of the solar circuit to the water, reaching states of superheated steam. The water circuit is a traditional Rankine cycle, where after expanding into the turbine, steam is condensed and cooled through cooling towers, and then goes back through the pumps segment of heat exchangers.

Solar thermal power plant present multiple modes and operating schemes, detailed below.
i) Operating modes

It is possible to design a continuously generating solar thermal power plants, but in order to achieve this it requires thermal storage systems to extend the operating hours at times of the day when solar radiation is not enough to generate power by itself (Stoddard et al., 2006). By means of a pair of molten salts storage tanks, heat is stored during high radiation hours, releasing it during moments where solar energy doesn’t amount for enough heat output. Alternatively, these systems extend the operation of the power cycle up to night hours, dispatching energy when the demand peaks don’t match the period of maximum radiation, as depicted in Figure 1-9.

The Andasol plant, in Spain is an example of this array. With 50 MW of installed power, it has a 7.5 hours storage capability. Thanks to that, operating hours are increased at full rated capacity from 2.000 to 3.500 hours a year (European Commission - Directorate General for Energy and Transport, 2004).

Throughout the day, not only because of terrestrial rotation, but for cloudiness also (affecting the direct fraction of solar radiation), the fluid’s temperature at the outlet of the solar field is variable. Hence, the amount of steam generated also varies. Due to a
limited range for the amount of steam feeding the turbine, this issue is often solved by means of a fossil boiler installed as a backup system, delivering the amount of heat lacking at the solar field for a proper functioning of the turbine (Price & Carpenter, 1999).

ii) Integrated solar combined cycle

Another mode of operation consists on integrating the collectors’ field to a traditional gas turbine power cycle. With this scheme, heat rejected by the Brayton cycle’s turbine is added to the heat gathered at the solar field, reducing costs in about 22% versus a solar thermal power plant (steam cycle) such as California’s (Maryippan, 2001).

![Combined cycle with solar field integration](image)

**Figure 1-10:** Combined cycle with solar field integration (adapted from Greenpeace Internacional, European Solar Thermal Industry Association, 2005)

Direct steam generation on the solar field is another possible design.

b) Direct Steam Generation

Traditional designs of parabolic trough based solar power plants uses synthetic mineral oils of high heat capacity flowing inside the collectors. However, due to elevated costs,
this material carries additional operation difficulties, such as toxicity, flammability (which implies additional expenses for fire control systems) and degradation of its properties over a certain temperature limit. Because of this last disadvantage, such oils impose a limit to the maximum operating temperature on the collector’s field (Odeh, Morrison, & Behnia, 1998).

By generating steam directly on the absorber, the need for a special work fluid is eliminated. The water heating process (warming, boiling and steam overheating) is done completely inside the tubes. Accordingly, the working pressure at the collectors’ field increases significantly reaching values over 100 bar due to steam generation. Nonetheless, this process brings several advantages by dispensing the need for a heat exchanger, a source of heat and pressure losses.

Investment reduction by avoiding heat exchangers, reducing pumping loads on one hand, and increasing operating pressure and temperature of the cycle translate into a final levelized energy cost reduction of 26% compared to the plant’s original design with heat transfer fluid (Langenkamp, 1998).

Figure 1-11: Direct steam generation solar power plant design (Eck & Hirsch, 2007)
In 1996, in order to demonstrate feasibility of direct steam technology, was launched the DISS project (Direct Solar Steam), located at the Plataforma Solar de Almería, Spain (Price et al, 2002). In a first stage, the project defined specific targets, until 1998. It was especially important to describe the behavior of the system in both starting and stopping procedures, and also the response to variable radiation conditions. This information would be useful to know the mechanical stress on the absorber tubes (Eck & Steinmann, 2002).

The designed steam generator is 500 meter long, with a maximum thermal output power of 2 MW (Eck et al, 2003), with steam properties given at 400°C and 100 bar. Being an experimental facility, there is no generation of electric power; the flow of steam (after its characterization) is condensed and then returned to the inlet of the solar field.

Although it would be desirable to obtain higher temperatures, delivering higher enthalpy steam and ultimately, increasing the efficiency of the whole process, working conditions are determined by the thermal stability of the selective coating at the external surface of the tubes set a limit to maximum temperature at the solar field. Not only because of the degradation of the coating; higher temperatures generate steep gradients on the tube material, causing stresses that could risk the integrity of the structure, mainly because of the breakage of the glass envelope surrounding the tube and defocusing of the receiver. Knowing these limitations, tests were performed at pressures of 30, 60 and 100 bar (Eck, et al, 2003).

Thanks to the experiments performed at this stage, it was concluded that structures which resist higher temperatures could make possible a more efficient absorption of solar irradiation on the tubes, thus raising the overall process efficiency.


Three designs were tested for based on steam production and process control criteria. The choice depends on the capability for delivering constant properties (pressure,
temperature) steam flow on the turbine’s inlet. Given that the collectors’ field is coupled directly to the power section (power conversion systems), the steam flow properties at the outlet of the field are the same of those at the turbine’s inlet. As the flow of steam depends on the amount of solar radiation, it is not possible to ensure a constant flow of steam at the outlet without means of thermal storage.

The ‘one-through’ mode is the simplest configuration, where all the water flow is preheated, evaporated and overheated in one pass through the solar field. However simple this configuration, the region where evaporation ends may vary. Thus, along the day the point where saturation is reached changes, causing high temperature gradients on the tube, risking the stability of the structure.

![Figure 1-12: Operation diagram, one through mode (Price et al, 2002)](image)

In the recirculation mode, the water flow entering the solar field is more than the maximum capable of completely change to steam. Due to the excess of water, the field is divided in a vapor generating section ending in a phase separator. On one end of the separator, steam flows towards an overheating section, while the excess water is returned at the field’s inlet, mixing it with the ‘new’ flow. There are difficulties associated to the control of the key parameter: the recirculation ratio; while a high ratio guarantees enough cooling at the absorbers, at the same time it increases the head loss.
A third mode is the injection scheme, considering different collectors modules, where the mix’s quality is controlled by adding liquid water in order to stabilize the boiling and overheating regions. Nonetheless, it has proven very difficult to control due to the current imprecision of the vapor fraction sensors (Eck & Steinmann, 2002).

DISS project demonstrated the feasibility of direct steam generation at the solar field, being the recirculation mode the design with better results, in terms of reduced thermal stress on the tubes and steam flow controllability. It was also proposed the integration of these solar fields with traditional Rankine cycles. Due to the optimum operating conditions at the turbine is around 550 °C, it was identified a need for high temperature resistant materials (Leon et al, 2002).
By 2003, facilities summed up a total of 4,000 operating hours. Afterwards, the INDITEP project continued with the advances achieved by DISS, succeeding on the generation of 5 MW of electricity. Although being a research project, it was decided to assure the robustness and reliability of this generation strategy, in order to give the first steps towards a pre-commercial plant. Therefore, plant size was reduced in order to limit financial risks.

For electricity generation purposes, an overheated steam turbine was used. On the other hand, collectors’ field was operated accordingly the recirculation scheme. This operation mode proved to be the safer and controllable one (Zarza et al, 2006).

As depicted on Figure 1-15, each group of collectors is composed of 3 pre heaters, 5 evaporators and 2 over heaters.

Despite the fact that project DISS carried out tests at pressures up to 100 bar, after the good results obtained at lower pressure, avoiding the risk of leakage, the field was designed to operate according to the parameters in Table 1-3.
Table 1-3: Collector field inlet and outlet parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water temperature at inlet</td>
<td>115 °C</td>
</tr>
<tr>
<td>Inlet water pressure</td>
<td>80 bar</td>
</tr>
<tr>
<td>Water temperature at outlet</td>
<td>410 °C</td>
</tr>
<tr>
<td>Outlet steam pressure</td>
<td>70 bar</td>
</tr>
</tbody>
</table>

In order to determine the vapor flow in each row of collectors, there are several facts that had to be accounted for: a flow too big would make necessary more collectors to achieve the same vapor generation (besides a greater head loss); lower flows generate liquid-gas stratification, with the resulting thermal stresses on the tube, due to different conditions for the heat transfer. Therefore, the choice of the mass flow commits these two aspects. In the case of INDITEP, a flow of 1.42 kg / s per row ensures 1.17 kg of superheated steam, meaning quality 0.82 at the end of the evaporating section (Zarza et al, 2006). Additionally, there is water injection into the entrance of the second overheated, thus controlling the temperature of generated steam.

All in all, the INDITEP project made use of the experience gained in the two phases of DISS to demonstrate the strategy of direct steam generation, focusing on the system’s performance reliability in real conditions of operation, leaving the overall efficiency maximization for future studies (Zarza et al, 2006).

Following this line of research, the design of a saturated steam solar power plant has been studied (Eck & Zarza, 2006), which would eliminate the need of a overheating section by using a special model of steam turbine, designed to operate in saturation conditions (allowing a steam fraction as low as 0.85). The idea behind this concept is to create smaller plants, attracting more investors. The current research results point that, whether it is possible to increase the amount of generated steam, the initial investment would be greater respect a overheated steam plant, mainly due to the complex operation of the turbine, which is not suited for repeated shut downs and starting procedures on a daily basis.
The studies for increasing the global efficiency of the direct steam generation process have focused on the performance of the absorber, both experimental and theoretical. The proper design of this component impacts directly on the plant’s operation, especially at the phase change region.

The absorber element’s performance impacts on two fronts: a heat transfer optimizing component increases the collection efficiency; at the same time it would mean saving materials generating the same amount of steam in a reduced length. Also, the component must sustain high pressures joined to the thermal stress caused by a non-uniform heat flow on the exterior (Benz et al, 2006). Early experimental work already showed deflections of the steel pipes while low flow of water circulation (1 l / min), especially during periods of start-up (thermal transient) (Almanza et al, 2002). Upward deflection of the tube hurts the process, by defocusing the receiver from the parabola’s focal line (up to 50 mm) where solar radiation is concentrated. In addition, there is the issue of the solar envelope, which could break due to the deflections. A natural choice in order to reduce temperature differences that caused the deflections was to replace the steel tube for one made of copper, a material of much higher thermal conductivity than steel. However, the continuous thermal cycling affects the mechanic properties of copper, (a process similar to soft-annealing); moreover, the material by itself imposes a pressure limit much lower than steel (1.7 MPa), a variable that determines the performance of the whole system, by imposing a limit on the properties of the steam generated.

Experimental studies have already verified a relation between working pressure and fluid flow with the flow pattern produced in presence of two phases inside the absorber (Eck et al, 2004). At lower pressures (30 bar), annular flow can be expected; intermediate pressures (60 bar) present both annular and stratified flow. Ever higher pressure levels (100 bar) present stratification for reduced flows, taking the experience gathered at INDITEP as a reference.
Figure 1-16: Relation between pressure and mass flow for steam-liquid flow patterns (Eck et al, 2004)

Figure 1-16 depicts a diagram obtained by testing the thermal behavior of horizontal absorbers where data (represented by circles) is grouped into three columns, according to the working pressure at which the tests were carried out.

As it is not possible to observe directly the flow patterns inside the absorber, they are deduced from the temperature distribution in the section of pipe. For this reason, several sensors monitor the maximum temperature difference in the profile through thermocouples installed on the surface of the tube. In this way, an annular profile is characterized by small angular temperature gradients, while at the presence of stratification; the upper region may present differences in temperature of several tens of degrees compared to the bottom of the tube. If there is drying, or complete saturation, temperature differences can be made more pronounced by the low coefficient of heat transfer of steam (Eck et al, 2004).

Therefore, stratification is considered as an undesirable phenomenon, and techniques are designed in order to either avoid it completely or deal with it without the harmful effects on the tube.
In most cases, low mass flows imply stratification. Recent studies suggest strategies to deal with this pattern (Rojas, Andrés, & González, 2008). A straightforward technique consists of preventing the formation of stratified flow, increasing the mass flow, forcing an annular pattern instead. Such flow depends on both working pressure and external heat flow. This measure impacts augmenting head loss, given that in order to generate the same amount of steam the length of the absorber must also be increased. Another way is to deal directly with stratified flow. An immediate advantage is that by not limiting the magnitude of the water flow, it is possible to increase the working hours of the plant, allowing it to function at lower radiation conditions. One of the direct strategies is the use of tubes with greater thermal conductivity, but at the same time, the material must have suitable mechanic properties in order to endure the working pressure.

A third approach is using capillary systems to aid in keeping the internal wall wet at all times, as occurs with the annular pattern. Current investigation points to the use of microchannels and porous coatings as depicted in Figure 1-17. The increased wall fraction covered by water enhances the overall heat transfer coefficient.

There are both theoretical and experimental studies showing that by using bi-metallic absorbers it is possible to solve the issue of high temperature gradients (Flores & Almanza, 2004). It is proposed a configuration consisting of two concentric tubes: an external steel tube and an internal copper tube. It has been shown that for low mass flows (between 60 and 150 kg/h) where stratification cannot be avoided, this configuration mitigates the temperature differences on the tube section. By doing this, stratification induced deflections are reduced.
Later on, research has focused on the bimetallic tubes with different proportions of materials and several operating regimes. In Torres’ research (Torres et al, 2006), a static heat transfer problem is solved for a fixed level of stratification. An 80% steel-20% copper mix is assessed, at high pressure conditions (165 bar). By solving the model for in transient state, it is concluded that compared to a 100% steel tube, thermal gradients on a section of the tube are lower.
Such researches point out the importance of a proper design of the absorber, a key element of the solar thermal plant. It is the phase change phenomenon which determines the heat transfer characteristics and its effects on the metallic structure. As it has been shown before, phase stratification is a critical event because it generates elevated temperature differences at the top and bottom of the tube. In order to avoid it, mass flow can be raised to assure annular flow, but at the same time, more collectors are needed. On the other hand, higher enthalpy steam requires higher pressures. And given that one of the consequences of high pressure is stratification, another limit is imposed to the plants, in order to avoid such pattern.

Solar field optimization requires the use of high pressure collectors (typically near 100 bar) on the smallest possible area. Such field would have to deal with stratification inside the tubes, and thus high thermal gradients, deforming the absorbers. For this reasons, absorber design studies arise, in particular the bimetallic approach.

However, existing studies discuss particular situations: for a given plant a design, the effect of a singular bimetallic configuration is studied. When modeling, only the heat transfer problem is addressed, not the thermally driven mass transfer processes, for example. Existing plant simulations are one dimensional, not considering the temperature distribution at the cross-section.
The present work discusses the behavior of proposed bimetallic structures for several operating conditions, under both single-phase and multi-phase regimes. To do so, relationships between inlet flow conditions and incident energy are established for different materials proportions. Accordingly, mass, momentum and heat transfer are modeled and solved by means of computational simulation.

1.3 Work motivation

There are several motives why the optimization of components of solar thermal power plants is a matter of interest, in particular the direct steam generation strategy.

At a global scale, solar concentrating plants for electric power generation do not emit greenhouse gases through their operation. Thus, this technology has a very limited impact on global warming. At a local level, Chile needs to diversify its energy matrix to be able to assure supply; given that doesn’t count with indigenous production of fossil fuels, traditional generation energy costs are threatened by volatile fossil fuels prices, jeopardizing the economy’s growth. Moreover, Chile is located on the region called ‘the solar belt’, the highest solar radiation area of Earth, optimum in order to the deployment of solar energy conversion technologies.

Concentrated solar power is the only technology tested at commercial scale for over 20 years. It has a broad capacity range (up to 200 MW are possible through solar tower facilities) and it’s supported by organizations such as the World Bank and the European Union, relying on an abundant potential for costs reduction in the upcoming years.

Several studies validate direct steam generation in solar power plants as an alternative for reducing generating costs when compared to traditional solar thermal power plants, demonstrated up to a pre-commercial level, thanks to the INDITEP experience. Moreover, it is possible to reduce costs even further by optimizing each plant components. The absorber is of critical importance: an optimum design allows higher pressures and ideally a reduction on the size of the solar field. Lowering the generating
costs of this solar generation strategy it is possible to integrate it to the Chilean energy matrix.

1.4 Summary

The growing energy demand drives the use of new power generating technologies. In particular, in the case of Chile, solar energy is emerging as an appealing alternative, given that the country possesses regions with high levels of solar radiation. In order to make this a competitive alternative, the use of thermal plants with parabolic channel is proposed. Such plants present opportunities for improvement in its design, generating steam directly in the collectors. The design of these elements can be optimized by improving the heat transfer from the tube to the fluid, using for example, more than one material on the wall depending on the characteristics of the flow. At present, existing studies are scarce, and mostly experimental, there are currently no theoretical studies that address the overall problem of heat transfer between a bimetallic absorber and the fluid with mass transfer promoted by thermal phase change. It is then the purpose of this thesis propose the optimum configuration of materials in the collector through computer simulation, so as to minimize the thermal gradients in the section, both in situations of single-phase sub cooled liquid and superheated steam as in the intermediate: analysis of stratification pattern. By eliminating temperature gradients and allowing stratification, it is possible to expand the operating range of the solar field, both in mass flow as pressure operation, allowing the deployment of small plants. This will establish relations between the conditions of entry and flow of incident energy with the materials of the tube. In order to comply such objective, several equations representing the problem’s physics must be resolved, such as: momentum, energy transfer, and temperature-driven mass transfer (thermal phase change from liquid to vapor) at a three-dimensional level.

On the following chapter, the problem to be addressed is stated, along with the approaches found in literature in order to solve it. Then, the objectives of this thesis are enounced, along with its expected contribution
2. PROBLEM DEFINITION AND THESIS OBJECTIVES

We attempt to find the best configuration of materials in an absorber tube of a parabolic solar collector channel such that during steam generation in a stratification pattern, the thermal gradients in the cross-section should be minimized. Stratification occurs in relatively low mass fluxes (lower than 1 kg/s), specially at elevated working pressure (Eck et al, 2004). It can also be generated due to sudden changes in radiation conditions (cloudiness). But, even at large scales thermal plants, working at high mass flow rates to assure an annular pattern before complete internal wall drying a so called open-annular pattern is formed in which the upper part of the liquid annulus ‘breaks’, drying that region of the tube. This situation causes a steep rise of the temperature at dried zones, exerting thermal stresses on the absorber, defocusing it and even causing the breakage of the glass envelope, in charge of maintain vacuum around the tube, to reduce convective losses with the environment.

In the literature, the progress of proposed alternative configurations for the materials in the absorber is discreet. Researches account for experimental studies taking into account only a couple of material combinations. Typically, the problem has been addressed through correlations in order to solve heat transfer inside the absorber, a methodology that cannot be generalized, failing at cases of phase’s stratification (Thome, 2007). Another approach to the problem has been solving only heat transfer processes between the tube and fluids, for a certain level of prescribed stratification without taking into account the phase change phenomenon, and the resulting effects of changes of vapor level on the tube’s wall temperature. These researches point out to the difference in the temperature fields obtained for the absorbers calculated.

Despite the existence of studies for bimetallic absorbers, this research presents advances of results on the behavior of such devices under the evolution of a two-phase scheme by means of simulating thermal phase change, on a three-dimensional approach.
2.1 Approaches to the process of steam generation inside smooth pipes

Steam generation inside the absorber presents several physical phenomena, representing both convective and conductive heat transfer, gas phase nucleation at internal walls, mass and energy transfer between phases, forming patterns as mass fraction approaches one.

![Figure 2-1: Flow patterns for boiling in a vertically heated tube (Collier & Thome, 2001)](image)

As depicted in Figure 2-1, let’s consider a vertical oriented pipe, with a constant heat flux on its external surface. On the bottom end, sub cooled liquid (meaning that for a
certain operating pressure, fluid temperature is below saturation) enters, while at the top, the outflow consists of pure vapor. In a first segment, we have single-phase convective heat transfer to liquid, laminar or turbulent (zone A on Figure 2-1). This mechanism endures until a certain distance where exists enough overheating at the wall, such as little vapor bubbles are generated on the surface, on spots called nucleation sites (zone B on Figure 2-1). Due to local overheating, generated bubbles are surrounded by subcooled liquid, collapsing as they approach the center of the tube. At this stage, some overheating at the wall can be observed, needed to maintain nucleation sites active. At the same time, bulk temperature steadily approaches saturation as the heat flux on the external surface is maintained.

Thermodynamically, there is a clearly defined boundary for the transition from subcooled nucleated boiling towards saturated nucleate d boiling. At such point, a thermodynamic balance yields saturation temperature for the fluid. Nonetheless, it’s still possible to found locally subcooled liquid. Then, the balance results are explained thanks to the presence of the vapor generated. Thus, we have bubbly flow in a liquid phase (zone C). Higher vapor fractions show a slug flow, where bubble coalescence forms a kind of vapor piston (zone D), and before the vaporization is completed, an annular flow is presented. At this stage, the liquid film’s effective conductivity proves to be high enough to avoid bubble formation, dragging heat from the wall to the interphase, where phase change occurs, in fact. Advancing further towards the outlet, the liquid annulus will eventually ‘break’, leaving the pipe internal wall dry (zone E-F). After this point, the pattern is called ‘fog flow’, where liquid droplets may persist on the core flow (zone G). A characteristic sign of wall drying is a steep rise of the tube’s temperature, because of the lower heat transfer coefficient of the gas phase (Collier & Thome, 2001).

At the same time, as the results of DSG showed, the patterns generated inside the tube are influenced also by the working pressure, the amount of water flow and the magnitude of the heat flux. Whether low heat flux could produce drying only close of the outlet (if any), a flux big enough could cause film boiling, still in presence of core
sub cooling, affecting severely the structure of the tube because of high temperature differences.

2.1.1 Boiling inside horizontal pipes

The most distinctive feature of this kind of flow is the lack of symmetry, as opposed to the vertical orientation, due to the effects of gravity. This difference is depicted in Figure 2-2. Even for a uniform heat flux, density difference drives a distinct evolution of flow patterns. An especially dire consequence is the drying (permanent or intermittent) at early stages of the heating process, as observed for slug flow, and so on. At higher mass velocities, the influence of gravity may decrease, tending to a more symmetric phases distribution as in the vertical case.

![Figure 2-2: Flow patterns for a horizontal heated pipe (Collier & Thome, 2001)](image)

2.1.2 Methods of analysis

One of the approaches to the problem of heat transfer is through correlations. They deliver information about the temperatures, heat transfer coefficients and amount of steam generated at a one-dimensional level.

On the following, the most used correlations for horizontal flow are briefly presented.

a) Correlation methods
In order to determine the temperature along the tube, heat transfer is determined by local two-phase heat transfer coefficient \( h_{TP} \), defined as:

\[
h_{TP} = \frac{q}{(T_{wall} - T_{sat})}
\]  

(2.1)

Where \( q \) represents the local flow of heat from the wall of the tube to the fluid, \( T_{wall} \) is the temperature of the wall in a certain axial position, \( T_{sat} \) is the saturation temperature at local saturation pressure \( p_{sat} \). The determination of \( h_{TP} \) generally involves the overlap of two effects: the transfer of heat by nucleated boiling \( h_{nb} \) and heat transfer by convective boiling \( h_{cb} \). The latter refers to the transfer of heat convection between the wall and the heated liquid phase.

In the case of horizontal tubes, the method for determining the ratio of two phases is usually an adaptation of procedures designed for vertical tubes, such as the correlation of Shah, one of the most commonly used in the literature (Thome, 2007).

i) Shah’s Correlation

Froude’s number defines a threshold for stratified flow:

\[
Fr_L = \frac{\dot{m}}{(\rho_L g d_i)}
\]  

(2.2)

Where \( \dot{m} \) stands for mass flow, \( \rho_L \) liquid density, \( g \) gravity and \( d_i \) internal tube diameter. For \( Fr_L < 0.04 \), the pattern is stratified.

The convective coefficient \( h_{cb} \) is calculated by:

\[
\frac{h_{cb}}{h_L} = \frac{1.8}{N^{0.8}}
\]  

(2.3)
Where $N$ is a factor whose value depends on $Fr_L$ and $h_L$ (liquid heat transfer coefficient) and is calculated based on the Dittus-Boelter correlation, with the liquid fraction of total flow (calculated as $\dot{m}(1 - x)$, with $x$, steam mass fraction):

$$h_L = 0.023 \, Re_L^{0.8} \, Pr_L^{0.4} \left(\frac{\lambda_L}{\mu_L}\right)$$

(2.4)

Where the liquid’s Reynolds number is defined for:

$$Re_L = \frac{\dot{m}(1 - x) \, d_i}{\mu_L}$$

(2.5)

with $\mu_L$ as dynamic viscosity of the fluid. Prandtl number is defined as:

$$Pr_L = \frac{c_{pl} \, \mu_L}{\lambda_L}$$

(2.6)

c$_{pl}$ as the liquid’s constant pressure heat capacity and $\lambda_L$ the fluid’s thermal conductivity.

If $Fr_L > 0.04$, this method applies without changes regarding the vertical oriented case, where $N = C_O$, with $C_O$ defined as:

$$C_O = \left(\frac{1-x}{x}\right)^{0.8} \left(\frac{\rho_g}{\rho_L}\right)^{0.5}$$

(2.7)

where $\rho_g$ represents the steam’s density.

If the flow is stratified, $N$ is corrected by:

$$N = 0.38 \, Fr_L^{-0.3} \, C_O$$

(2.8)

This relation’s effect is to lower the magnitude of $h_{tp}$ for low mass velocities.

Nucleate boiling heat transfer coefficient $h_{nb}$ is calculated as:

If $N > 1$ and $Bo > 0.0003$:
\( h_{nb}/h_L = 230 \text{Bo}^{0.5} \)  \hspace{1cm} (2.9)

With Bo, boiling number, representing the true fraction of heat flux over that reachable if total liquid evaporation occurred:

\[ Bo = \frac{q}{(m \cdot h_{FG})} \]  \hspace{1cm} (2.10)

If \( N > 1 \) and \( Bo < 0,0003 \):

\[ \frac{h_{nb}}{h_L} = 1 + 46 \text{Bo}^{0.5} \]  \hspace{1cm} (2.11)

If \( 1 > N > 0,1 \):

\[ \frac{h_{nb}}{h_L} = F_S \text{Bo}^{0.5} \exp(2,74N - 0.1) \]  \hspace{1cm} (2.12)

Shah’s constant, \( F_S = 14,7 \) if \( Bo > 0,0011 \), while if \( Bo < 0,0011, F_S = 15,43 \).

Where \( N < 0,1 \), the coefficient is calculated as:

\[ \frac{h_{nb}}{h_L} = F_S \text{Bo}^{0.5} \exp(2,74N - 0.15) \]  \hspace{1cm} (2.13)

Finally, \( h_{TP} \) is determined by taking the maximum between the convective and boiling coefficients.

ii) Gungor-Winterton correlation

Just as in Shah’s, there are several corrections to be made from the original method, also designed for vertical tubes (Thome, 2007). Here, the overlapping is given through:

\[ h_{TP} = E \cdot h_L + S \cdot h_{nb} \]  \hspace{1cm} (2.14)

This method also sets a threshold for stratification, defined for values of \( Fr_L \) smaller than 0,05.
Like before, $h_L$ is calculated as in Shah’s method, meanwhile nucleated boiling coefficient is obtained through Cooper’s relation:

$$h_{nb} = 55p_r^{-0.12}(-0.4343 \ln p_r)^{-0.55}M^{-0.5}q^{0.67} \hspace{1cm} (2.15)$$

Where heat flux must be entered as $W/m^2$. $M$ stands for molecular weight and $p_r$ reduced pressure, defined as the ratio between saturation pressure and critical pressure. Convective factor $E$ is defined as:

$$E = 1 + 24.000Bo^{1.16} + 1.37 \left(\frac{1}{X_{tt}}\right)^{0.86} \hspace{1cm} (2.16)$$

where

$$X_{tt} = \left(\frac{1-x}{x}\right)^{0.9} \left(\frac{\rho_l}{\rho_g}\right)^{0.5} \left(\frac{\mu_L}{\mu_g}\right)^{0.1} \hspace{1cm} (2.17)$$

$\mu_L$, the fluid’s dynamic viscosity.

$E$ is corrected by a $Fr_L^{(0.1-2Fr_L)}$ factor when below the threshold for Froude’s number.

It should be noted though that if $1/X_{tt} \leq 0.1$, $F$ equals 1. Boiling factor is:

$$S = \left[1 + 0.00000115 E^2 Re_L^{1.17}\right]^{-1} \hspace{1cm} (2.18)$$

which, if under Froude’s threshold, should be corrected by a factor equal to $Fr_L^{0.5}$.

Despite of being relatively easy to implement, the above cannot be validated as general methods for the resolution of convective boiling problems. For instance, Froude number’s threshold has proven very unreliable, being underestimated by a factor of 10 in most cases, especially those of stratified flow.

A second approach is direct resolution of the thermodynamic problem, including the mass transfer mechanisms.

The method of analysis for steam generation inside tubes is an extension of the methodology used in single-phase cases: a domain is to be defined, along with the governing equations of relevant physics (mass, momentum and energy conservation),
later solved thanks to simplifications which depend on each case. The approaches found in literature are:

b) Homogeneous flow model

Two-phase flow is assumed as single phase flow of a material which has properties obtained through weighting each phases’ properties. The assumptions of this model are, chiefly: same velocity field for both phases; thermal equilibrium between phases and use of a suitable friction factor. From Figure 2-2, one would expect a homogeneous model assumption only at early stages of the heating process. However, it has been widely used even in high gas fraction patterns.

c) Separated flow model

The number of equations is doubled, with for each phase having its own set. Additionally, information must be provided on the relationship between each phase and the tube wall, as well as the interaction between the phases, either through empirical relationships or particular simplifications, which may complicate the calculations.

d) Flow patterns

In the form of maps, regions are graphically described, where the flow will take the form of a pattern determined by variables used as coordinates on the map. Then, basic equations are solved in that frame of reference, as the example in Figure 2-3.
Thus, the use of computer simulation to resolve on detail the characteristic thermodynamic processes of steam generation inside horizontal tubes (Chull & Jo Kim, 2004) is required. This involves choosing the most suitable models for carrying out calculations, including limitations on the physics of the problem and the simplifications of each case. The heat transfer in a horizontal tube composed of different materials are resolved for both single-phase liquid and gas flow, and flow with two-phases steam generation at different levels of stratification. This is intended with the purpose of generating a criterion of selection of materials and configurations of tubes, according to the different operation conditions at the fields, based on the relationship between the thermal behavior of the solution and the types and proportions of the materials used. Finally, the results are validated by comparison with experimental and analytical results from research found in the literature.

Then, the contribution of this thesis is:

- To present the application of computer simulation to the problem of heat transfer in stratified flow as an effective approach for the treatment of problems involving phase change in these conditions. The method allows testing multiple designs under different
operating conditions without the need to build prototypes. By doing so, a methodology for simulating these kinds of flows is established.

- To develop a selection criterion for materials to be used in the absorber under the present flow pattern. Thanks to this, the possibilities for design of solar thermal plants are expanded, by making possible to work in conditions where the consequences of stratified flow limit the mass flow or working pressure pressures on which they operate.

### 2.2 Objectives

a) Definition of the flow to be modeled

In order to describe the flow in the most accurately way, a selection of most suitable models for simulating single phase and two phase flow with mass and heat transfer. Both the focus on how to deal with each one of the phases (either homogeneous or separated flow models) and the phase change treatment are to be adjusted according to the boundary conditions (geometric and temporal) of the problem.

b) Application of stratified flow model to heat transfer inside metallic pipe

Stratified flow model, once defined, is applied to the case of the absorber present on solar collectors on a direct steam generation solar power plant. The results obtained will show the temperature field product of the flow pattern and the external heat flux.

c) Establish the relation between stratification and thermal gradients for different tube arrangements

The defined model is then applied to several bimetallic tubes, with different proportions on the materials the tube is made of.

d) Define a choosing criterion for the absorber type
Once a relation between the mix of materials the tube is made of has been determined, and the thermal response has been characterized for both single phase (sub cooled liquid segment and overheated steam segment) and two-phase stratified flow, a minimum cross section temperature gradient criterion is established.

This attainment of the proposed objectives implies solving (on a three dimensional level) the equations governing the physics of the absorber’s flow, expressed as mass, momentum and energy conservation, along with additional closure relations needed to model specific phenomena (when applies) as interphase mass and energy transfer due to thermal phase change occurring inside the pipe. These equations are defined on a discretized domain where the inlet flow of water is prescribed, as the flux of heat on the external surface of absorber’s model. Solving the coupled problem of heat transfer with the pipe’s materials yields the temperature fields needed to analyze the proposed configurations.
3. METHODOLOGY

On the present chapter, the methodology employed to deal with heat transfer on the parabolic trough’s absorber tube is discussed. Regions of sub cooled liquid, superheated steam and intermediate stratified flow are simulated by means of computational fluid dynamics. On the second part, the methodology is described in terms of the governing equations and the assumptions made in order to apply them to a particular case: the study of thermal behavior for different combinations of materials in a bimetallic absorber, versus the results obtained for the typical steel absorber used today, under stratification conditions.

Computational fluid dynamics is an engineering tool used mostly to simulate the behavior of systems where detailed features of the fluid flow are of interest, including advection, heat transfer, radiation or mass transport effects, just to mention some. Through the application of this tool, it is possible to contribute with crucial information for the design process, accelerating it and, at the same time, reducing cost by eliminating the need for prototype building which can be highly time-consuming, besides expensive. Once this method generates results, modifying the models is fairly easy, thus it is possible to get information for several conditions influencing the system without having to start the process from scratch.

3.1 Simulation process through computational fluid dynamics

Computational fluid dynamics (CFD) is based on the numerical solution of mass, momentum and energy conservation. Additionally to these equations, specific relations describing more complex physics, equations for each domain’s boundary and data for initial conditions are needed for closure. Appendix B develops a example case to validate

In order to solve such differential equations system, a numerical method has to be used. In the finite volume method (FVM) the geometric space defined by the problem is
divided in little regions or control volumes, forming a two-dimensional or three-dimensional mesh. Once the domain has been discretized, equations are applied to each control volume, given that physical variables are conserved for all elements. Because of this feature, a linear system of equations is generated due to discretization of governing equations also. Such system is then solved through iterative solution methods.

The difference between current and previous iteration’s results is called residual, or error. This amount is a measure of conservation of the properties of the flow. Thus, as the residue approaches zero, a converged solution is obtained. The difference between this solution and the problem’s exact solution depends on both physics and the domain’s discretization. Additional to this, since many physical processes are represented by empirical relationships, there may be a greater difference to the behavior of the actual system.

Consequently, the method yields an approximation of the fields of variables relevant for a given problem, through a limited number of points on the domain (due to space discretization), where the balances are calculated. By analyzing these sets of physical variables, it is possible to represent the system behavior and evolution according to the initial and boundary conditions.

The process itself has a well defined step sequence; what follows is the description of those stages:

a) Geometry definition: problem’s domains.

The domain is defined by its boundary conditions; in turn, they depend on each problem’s physics and dimensions, representing regions such as inlets, outlet and walls. After defining these regions, the whole domain’s geometry is transformed into a mesh made up of tiny control volumes. The shape of the elements can vary, depending on their location. Specifically, is very important to create reduced elements near the domain’s boundaries, in order to represent accurately the boundary layer phenomena through an increased number of calculation points per length unit. The immediate consequence of
Mesh generation is the time required for solving the equations generated, related to the number of control volume’s balances. There’s a trade-off between the need of a fine mesh in which results are more accurate and processing capacity and available time for the calculations.

b) Physics definition

For each problem it is necessary to describe the relevant models, meanwhile others are discarded if their influence on the flow is negligible. For this reason, when defining the simulation, it should be decided what physical models take part in the calculation, whether representing a variable’s conservation equation or a closure relationship (like interphase terms or chemical processes, for example). In the same way, it is possible to add terms as sources or sinks of mass, momentum or energy. To completely define the problem are also necessary equations to represent boundary conditions as inputs, output regions, the relationship of the walls with the flow, and so on. Last but not least, the properties of the materials presents in the domain must also be defined.

c) Time dependency of the solution

According to the problem and the research purposes, if the evolution of the variables along time is a matter of interest, the problem can be modeled on a transient state. In that case, several solutions stages are generated given a certain time step being necessary to define the initial time, final time and the magnitude of each time step. On the other hand, a steady state analysis yields a single solution where the values of the variables are not time dependant. The need for a transient simulation is in close relation with the problem’s physics; there are for example, periodic physical phenomena that can’t be modeled as time-independent.

Appendix B shows a brief case of CFD use as a starting point conceptualization, by comparing results of a classic type of heat transfer, solved by analytic methods and CFD tools.
3.2 Applying CFD to steam generation inside a horizontal pipe

The problem of steam generation comprises several physical mechanisms, modeled through CFD, by means of their governing equations and suitable empiric relations.

As it has been shown in previous chapters, the problem is a case of phase change in presence of a convective flow with a prescribed heat flux addition. Therefore, both velocity and temperature fields are to be solved, along with the variables related to the phase change. At the same time, the heat transfer within a solid domain (i.e. the pipe structure itself) is addressed at the same time, having to solve that system also.

The solution of the heat transfer inside the fluid domain combines both convective and thermal phase change effects. Those mechanisms rise yet at present, numerous challenges for simulation; usually the obtained results are by means of correlations, which can’t be generalized to every case. Phase change is represented on the relations as a mass transfer term between phases because of thermal effects, thus there’s a coupling between all the simulated events occurring inside the pipe.

In the following section, the single phase problem is addressed. On one hand, it represents the physics for the first and last section of the absorber, meaning sub cooled liquid and superheated steam flow; on the other, the physics for the simulation of the steam generating section are based on the single phase problem, to which several closure equations are added in order to simulate phase change.

3.2.1 Governing equations for the single phase problem

As described in Chapter 1, the evolution of the flow pattern for the problem at hand begins with only one phase: sub cooled water. At this stage, the fluid domain solution fields are obtained through solving Navier-Stokes system along with energy conservation, coupled with an energy equation for the solid domain. Basically, a forced convective problem with conjugate heat transfer in solid domains must be solved for a single phase fluid field.
In what follows, the governing equations of the solved physics are described.

a) Mass conservation equation

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \]  
(3.1)

where \( \rho \) and \( \mathbf{U} \) stand for density and velocity vector, respectively.

b) Momentum conservation equation

\[ \frac{\partial}{\partial t} (\rho \mathbf{U}) + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla p + \nabla \cdot \mathbf{\tau} + S_M \]  
(3.2)

With \( \mathbf{\tau} \) representing the deformation related stresses in the expression:

\[
\mathbf{\tau} = \left( \mu (\nabla \mathbf{U} + (\nabla \mathbf{U})^T) - \frac{2}{3} \delta \nabla \cdot \mathbf{U} \right)
\]  
(3.3)

where \( \delta \) is Kroenecker’s delta, \( \mu \) dynamic viscosity and, \( S_M \) the source (or sink) of momentum.

c) Energy conservation equation

Total energy equation for a control volume is defined by:

\[ \frac{\partial (\rho H_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{U} \cdot \mathbf{H}_{tot}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (\mathbf{U} \cdot \mathbf{\tau}) + \mathbf{U} \cdot \mathbf{S}_M + \mathbf{S}_E \]  
(3.4)

Where \( H_{tot} \) represents total enthalpy:

\[ H_{tot} = H + \frac{1}{2} \mathbf{U}^2 \]  
(3.5)

\( \nabla \cdot (\mathbf{U} \cdot \mathbf{\tau}) \) represents the contribution due to viscous stress.
The term $U \cdot S_m$ accounts for the work exerted by external momentum sources. Finally, $S_E$ represents a source term for external work sources.

It is possible, though, to perform the following simplifications:

For low velocity and constant density flows, the contribution of mechanic energy can be disregarded, only dealing with thermal energy conservation:

$$\frac{\partial (\rho H)}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho \, U H) = \nabla \cdot (\lambda \nabla T) + U \cdot \nabla p + (\nabla \cdot (U \cdot \tau) - U \cdot (\nabla \cdot \tau)) + S_E \tag{3.6}$$

Where the term $(\nabla \cdot (U \cdot \tau) - U \cdot (\nabla \cdot \tau))$ represents viscous dissipation and it is always negative. Applying a last relation, between enthalpy and internal energy:

$$H = e + \frac{p}{\rho} \tag{3.7}$$

with $e$, the internal energy. Thus, a thermal energy conservation equation is obtained:

$$\frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho \, U e) = \nabla \cdot (\lambda \nabla T) + p \nabla \cdot U + (\nabla \cdot (U \cdot \tau) - U \cdot (\nabla \cdot \tau)) + S_E \tag{3.8}$$

d) Energy conservation in solid domains

For heat transfer in a solid, energy conservation is simpler, in absence of convective effects.

$$\frac{\partial (\rho c_p T)}{\partial t} = \nabla \cdot (\lambda \nabla T) + S_E \tag{3.9}$$

where $\rho$, $c_p$ and $\lambda$ are the solid’s density, heat capacity and thermal conductivity.

Besides these basic equations, turbulent effects must also be addressed. Turbulent consists on variations on the velocity field in time and space. It occurs in multiple length scales, and usually is an unstable and three dimensional phenomena. Turbulence is often
related to high order Reynolds number, such as the observed in this problem, when inertial effects become comparable to viscous stress. Reynolds number \(Re_D\) is obtained as:

\[
Re_D = \frac{\rho U D}{\mu}
\]  

(3.10)

In this case, \(D\) represents the pipe’s diameter, where a fluid of mean velocity \(U\) flows.

While the physics of flow are fully represented by the Navier-Stokes equations, the scales at which turbulent effects occur would require of a discretization level such that their solution would not be feasible. For this reason, statistics-based turbulence models are employed to quantify the impact due to these variations.

In order to model turbulent effects, fluid velocity is separated in two: a time-averaged component \(\bar{U}\) and a time-variable one \(u\). Then:

\[
U = \bar{U} + u
\]  

(3.11)

The result of this formulation applied to the momentum transport equation is:

\[
\frac{\partial}{\partial t} \left( \rho U \right) + \nabla \cdot \{ \rho U \otimes U \} = \nabla \cdot \{ \tau - \rho \bar{u} \otimes \bar{u} \} + S_M
\]  

(3.12)

The term \(\rho \bar{u} \otimes \bar{u}\) is called ‘Reynolds’ stress’.

In energy equation, applying equation (3.11) at enthalpy definition yields:

\[
H_{tot} = H + \frac{1}{2} U^2 + k
\]  

(3.13)

With \(k\), the turbulence contribution to enthalpy:

\[
k = \frac{1}{2} \bar{u}^2
\]  

(3.14)
In general, for transport of a given property $\phi$, turbulence’s contribution $\rho \overline{u\phi}$ causes the following change:

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\rho \mathbf{U}\phi) = \nabla \cdot (\Gamma \nabla \phi - \rho \overline{u\phi}) + \mathbf{S}_\phi$$  \hspace{1cm} (3.15)

The technique chosen for modeling turbulent effects is called “eddy viscosity model”. Its main hypothesis is that turbulence is made of little vortexes (eddies) in permanent formation and dissipation, and that the effects on turbulence are proportional to velocity gradients.

According to this model, Reynolds’ stress is defined as:

$$-\rho \overline{u \otimes u} = \mu_t (\nabla \mathbf{U} + (\nabla \mathbf{U})^T) - \frac{2}{3} \delta (\rho k + \mu_t \nabla \cdot \mathbf{U})$$ \hspace{1cm} (3.16)

The term $\mu_t$ is call turbulent viscosity, and it also must be modeled. In the same way, turbulent contribution to the diffusion of a certain property is related to its gradient by the equation:

$$-\rho \overline{u\phi} = \Gamma_t \nabla \phi$$ \hspace{1cm} (3.17)

Where eddy diffusivity $\Gamma_t$ is defined. The relation for $\Gamma_t$ is:

$$\Gamma_t = \frac{\mu_t}{Pr_t}$$ \hspace{1cm} (3.18)

$Pr_t$ is the turbulent Prandtl’s number.

Hence, closure equations are needed for $\mu_t$. For single phase cases the $k-\varepsilon$ model has been chosen. This model relates turbulent viscosity as the product between turbulent velocity and a given turbulence length scale. Turbulent velocity is described in terms of the turbulent kinetic energy $k$, obtained through solving its own transport equation. The length scale on the other hand is estimated through two properties of the flow: turbulent kinetic energy and its dissipation rate, also obtained by solving its transport equation.
The model states the following equations:

$$
\mu_\varepsilon = C_\mu \frac{k^2}{\varepsilon}
$$

where $\varepsilon$ represents the vortexes’ dissipation. $C_\mu$ is a constant value equal to 0.09.

Transport equations for $k$ and $\varepsilon$ are:

$$
\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho \mathbf{U} k) = \nabla \cdot \left[ \left( \mu + \frac{\mu_\varepsilon}{\sigma_k} \right) \nabla k \right] + P_k - \rho \varepsilon
$$

$$
\frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \mathbf{U} \varepsilon) = \nabla \cdot \left[ \left( \mu + \frac{\mu_\varepsilon}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{\kappa} (C_{e1} P_k - C_{e2} \rho \varepsilon)
$$

where the constants $C_{e1}, C_{e2}, \sigma_k$, and $\sigma_\varepsilon$ take values of 1.44, 1.92, 1, and 1.3, respectively (ANSYS Inc, 2006).

$P_k$ is the turbulence production due to viscous forces and by density difference on the flow, modeled as:

$$
P_k = \mu_\varepsilon \nabla \mathbf{U} \cdot (\nabla \mathbf{U} + \nabla \mathbf{U}^T) - \frac{2}{3} \nabla \cdot \mathbf{U} (3 \mu_\varepsilon \nabla \cdot \mathbf{U} + \rho \ k) + P_{kb}
$$

Where $P_{kb}$ is the buoyancy turbulence production and it’s given as:

$$
P_{kb} = \frac{\mu_\varepsilon}{\nu_{rt}} \mathbf{g} \cdot \nabla \rho
$$

Flow simulations must also be capable of taking into account the effects of boundary layer, both hydrodynamic and thermal.

Closer to the wall, viscous effects become important and CFD uses the relation between the distance to the wall and velocity in the case of hydrodynamic layer, approximated by a logarithmic function. By doing this, there’s no need to excessively refine the mesh near the walls to model gradients. There’s a direct impact on the processing cost due to less calculation points.
The set of equations needed to simulate properly the system is completed with those representing boundary conditions. These are the relations that produce specific results for a problem’s given geometry and physic conditions.

### 3.2.2 Boundary conditions for the single phase problem

#### a) Fluid Inlet

An entry condition assumes fluid flowing inside the domain, specified by a velocity profile or a mass flow normal to the cross area at the inlet. In addition, if the flow is modeled as turbulent, turbulent energy and dissipation rate must be provided, for the $k - \varepsilon$ model, through the relation:

$$I = \frac{u}{\bar{u}}$$

(3.24)

Where $I$, the turbulent intensity is known and takes a value between 1 and 10%. Values for $k$ and $\varepsilon$ are determined by:

$$k_{in} = \frac{3}{2} I^2 U^2$$

(3.25)

$$\varepsilon_{in} = \frac{3}{2} I^2 U^2$$

(3.26)

$$\varepsilon_{in} = \rho C_p \frac{k^2}{\mu_t}$$

(3.27)

Finally, the turbulent viscosity:

$$\mu_t = 1.000 \, I \mu$$

(3.28)

The incoming flow’s temperature is prescribed directly. Consequently, this defines an advective energy flux through total enthalpy:

$$Q_{in} = m_{in} H_{tot}$$

(3.29)

#### b) Fluid outlet
The outlet boundary condition for a certain region is based on the assumption that flow only can leave the domain. A well defined problem requires the static pressure to be described. However, local variations on the area are allowed, fixing the area-averaged pressure. Thus, the velocity profile on the outlet is a result of the simulation.

c) Domain walls

Solid domain’s inner walls, in contact with the fluid are modeled as motionless, thus the fluid velocity in contact with the wall is zero. In addition, the tube’s influence on the flow regarding roughness-induced turbulence is neglected. In the case of interaction between solid and fluid domains, the interfacial boundary condition on the internal tube is such that allows conservation of the heat flow between domains (called interphase conservative flux).

Meanwhile, at tube’s external wall, heat transfer is modeled by setting a wall heat flux on the external wall per surface unit.

d) Symmetry planes

Depending on the domain’s geometry and flow characteristics, there might be symmetry planes that allow simulating a section of the domain bounded by one or several symmetry planes. Thus, it is possible to save significant time and resources, with a fraction of the calculation points needed for a complete set of results. This condition is modeled by defining a region (i.e. a plane) where normal velocity field equals zero and the gradient of all other variables it is also cancelled, i.e.:

$$\frac{\partial \phi}{\partial n} = 0$$

(3.30)

with \( n \) a vector normal to the symmetry plane.
Later, it will be analyzed the case of the horizontal pipe, where there’s symmetry condition represented by a vertical plane crossing the center of the pipe, parallel to the axis.

### 3.2.3 Equations treatment and calculation of the solution

Navier-Stokes system, turbulence quantities equation, along with the rest of transport equations and the closure relations are integrated over each control volume comprising the domain. Then, Gauss’ Divergence Theorem is applied to transform volume integrals into surface integrals. Figure 3-1 depicts a two-dimensional model of a mesh, with the surface of the control volume characterized by the dashed area.

Figure 3-1: Sketch of a control volume (ANSYS Inc, 2006).

Applying the element-based finite volumes method, equations are discretized and then a linear system is obtained. For instance, accumulation terms are discretized by approximation of values on each sector comprising the control volume, whereas the surface flow terms are approximated at integration points, (ip), showed in Figure 3-2. Then the flows are obtained through integration of the fluxes over the surface contributing to a control volume.
Second order accuracy schemes are applied to the advective terms (properties transported due to the movement of a fluid particle) in each transport equation.

The problem’s solution variables are stored at the mesh’s nodes. However, several terms in the equations require approximated solutions, or gradients of these variables in places other than the nodes.

Shape functions (from the finite elements method) are used for evaluating the variables (and their changes) inside the mesh’s elements. Shape functions (interpolating functions) used in ANSYS CFX-11 are linear in terms of parametric coordinates defined for a type of element, forming third grade polynomials.

3.2.4 Temporal dependence of the solution

For single phase flow, the solution is obtained after reaching a steady state. Nevertheless, the time-dependent term present in conservation equations is not cancelled. Instead, a false time transient is applied in order to aid solution convergence. Hence it is possible to accelerate system convergence toward a steady state if a suitable time step is chosen.
In particular, transient terms are solved by means of a first order Euler’s numeric scheme which is robust and totally implicit. This discretization though has first order precision in time and induces discretization errors by smearing of steep time gradients. This behavior is similar to numerical diffusion experimented with upwind scheme for advective terms.

In the case of single phase flow inside a pipe, the time step is calculated in function of the domain’s dimensions and initial conditions.

The solid domain has its own time step, usually greater than the fluid’s. The estimation is done calculating an average diffusion coefficient over the whole domain:

\[
\bar{\alpha} = \frac{\lambda}{\bar{\rho} \bar{c}_p}
\]  

then the time step is given by:

\[
\Delta t = \frac{l_{scale}^2}{\bar{\alpha}}
\]  

where \( l_{scale} \) represents a representative length scale, which, as with the fluid’s timestep, varies between the domain’s maximum length and \( \sqrt[3]{V} \), where \( V \) is the domain’s total volume.

### 3.2.5 Convergence criteria for the single phase problem

Given that the calculation procedure is done as an iterating procedure, a finishing condition has to be defined.

Basic convergence criterion is created through the definition of a maximum value for the residuals’ norm, a measure of the quality of the balance of each equation on every control volume the domain is made of. The value is determinate through the root mean square of the residual vector.
Because it is possible that numerically the solution doesn’t vary between iterations even when the residuals are above the proposed limit, additional criteria are proposed: such as global balance of the variables of interest of the problem, assuming that their stabilization implies achieving steady state.

In the single-phase problem at hand, the temperature on the pipe’s outlet is expected to stabilize when steady state has been attained.

3.2.6 Applying CFD to the steam generation problem

In this section, the features of multiphase simulation are presented. The problem of steam generation in stratification conditions in which two fluids are in contact with mass transfer between them, while separated by a defined interphase.

Therefore, there is a distinct set of conservation equations for each phase present on the domain. Moreover, the relationship between the phases must be modeled by means of additional equations applied on the interphase, representing momentum, mass and/or heat transfer (such is the case of thermal phase change).

Below, the approximations used for a methodology for dealing with the absorber tube physics are presented. Thereafter, the dynamic and thermal models chosen are discussed, to finally treat the phase change applied to the problem’s particular conditions.

From now on, Greek letter χ is used as a sub index to name the different phases to be modeled (for a two phase problem, $\chi = \text{steam, liquid}$).

Given the expected phases’ disposition within the domain, several parameters must be set according to the simulated case, from mesh discretization level up to the forces exerted between phases. Understanding that the phases’ mixing can occur in scales smaller than the element’s size, it is assumed that both phases are present on each control volume, quantified through a volume fraction $r_\chi$. 

55
In the following, both approaches on multi phase flows: homogeneous fluid and separated flow are now discussed and theirs equations described. Also, the free surface flow is defined and the equations for the thermal phase model are discussed.

The total pressure for multiphase simulations is determined as:

\[ p_{tot} = p_{stat} + \sum_{a} \frac{1}{2} \rho_{a} U_{a}^{2} \tag{3.33} \]

Multiphase flow is modeled under the assumption that each phase shares the same pressure field, meaning:

\[ p_{\alpha} = p \text{ for every } \alpha. \]

a) Non-homogeneous multiphase flow

Each fluid is modeled separately, thus doubling the continuity, momentum and energy conservation equations. The interaction between phases is accounted through closure relations applied on the interphase. Accordingly, the calculation of these terms is strongly conditioned by the phases’ morphology, particular to every problem (basically, it depends if there is a phase dispersed into the other, or whether each phase can be considered as continuous).

i) Mass conservation for the \( \alpha \) phase

\[ \frac{\partial}{\partial t} (r_{\alpha} \rho_{\alpha}) + \nabla \cdot (r_{\alpha} \rho_{\alpha} U_{\alpha}) = \sum_{\beta=1}^{N} (r_{\alpha}^{\beta} - r_{\beta}^{\alpha}) \tag{3.34} \]

\( N \) represents the total number of phases other than \( \alpha \) and \( r_{\alpha}^{\beta} \) stands for positive mass flow per unit of volume form phase \( \beta \) towards phase \( \alpha \). The sum of volume fractions over all phases must be conserved, meaning:

\[ \sum_{\alpha=1}^{N} r_{\alpha} = 1 \tag{3.35} \]

Similarly, each phase’s momentum equation differs from the single phase case due to phase interaction.
ii) Momentum equation for the $\alpha$ phase

It should be noted that the equation hereby presented includes the terms correspondent to the application of a turbulence model, such as the $k - \varepsilon$ model.

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha U_\alpha) + \nabla \cdot (r_\alpha \rho_\alpha U_\alpha \otimes U_\alpha) - \nabla \cdot \left( r_\alpha \mu_{\alpha \text{eff}}(\nabla U_\alpha + (\nabla U_\alpha)^T) \right) =$$

$$-r_\alpha \nabla \rho'_\alpha + r_\alpha \left( \rho_\alpha - \rho_{\text{ref}} \right) g + q$$

(3.36)

Where:

$$p'_\alpha = p_\alpha + \frac{2}{3} \rho_\alpha k_\alpha$$

(3.37)

$$\mu_{\alpha \text{eff}} = \mu_\alpha + \mu_{\text{ta}}$$

(3.38)

$$\mu_{\text{ta}} = C_\mu \rho_\alpha \left( \frac{k_\alpha}{\varepsilon_\alpha} \right)$$

(3.39)

$\mu$, $p$, $k$ and $\varepsilon$ represent viscosity, pressure, turbulent kinetic energy and turbulent dissipation rate for $\alpha$ phase, respectively. $C_\mu$ is a constant value equals 0,09. Operator $\otimes$ represents tensor product and $(\nabla U_\alpha)^T$ the transposed matrix of the term $\nabla U_\alpha$. $M_\alpha$ denotes the sum of forces on the interphase acting on the $\alpha$ phase due to the presence of other phases, such as drag and momentum transfer associated to mass transfer on the interphase.

$$M_\alpha = \Sigma_{\beta \neq \alpha} M_{\alpha \beta} = \Sigma_{\beta = 1}^{N} c^{(d)}_{\alpha \beta} (U_\beta - U_\alpha) + \Sigma_{\beta = 1}^{N} (\Gamma^+_{\alpha \beta} U_\beta - \Gamma^+_{\beta \alpha} U_\alpha)$$

(3.40)

To define the drag coefficient for a pair of phases $c^{(d)}_{\alpha \beta}$, experimental relations are used, depending on the phases’ interaction.

iii) Energy conservation for phase $\alpha$
\[
\frac{\partial}{\partial t}(r_{\alpha} \rho_{\alpha} H_{\alpha}) + \nabla \cdot (r_{\alpha} (\rho_{\alpha} u_{a} H_{\alpha} - \lambda_{a} \nabla T_{\alpha})) = \sum_{\beta=1}^{N} (\Gamma_{\alpha \beta}^{+} H_{\beta s} - \Gamma_{\beta \alpha}^{+} H_{\alpha s}) + Q_{\alpha}
\]  

(3.41)

Where \( H, T \) and \( \lambda \) symbolize enthalpy, temperature and thermal conductivity for phase \( \alpha \), respectively. \( Q_{\alpha} \) denotes the interphase heat per unit of time and volume to phase alpha from other phases and is modeled as:

\[
Q_{\alpha} = \sum_{\beta \neq \alpha} Q_{\alpha \beta} = h_{\alpha \beta} A_{\alpha \beta} (T_{\beta} - T_{\alpha})
\]  

(3.42)

And the term \( (\Gamma_{\alpha \beta}^{+} H_{\beta s} - \Gamma_{\beta \alpha}^{+} H_{\alpha s}) \) represents the heat transfer induced by mass transfer on the interphase.

For momentum, heat or mass transfer between phases the variable \( A_{\alpha \beta} \) (dimensions of \( m^2/m^3 \)); interphase area density is of great importance, depending on how phases are modeled. For the simulation of stratified flow, the resolution of the interphase between fluids is attempted (ANSYS Inc., 2006). For such two-phase case, the area density is defined by:

\[
A_{\alpha \beta} = |\nabla r_{\alpha}|
\]  

(3.43)

b) Homogeneous Multiphase Flow

The case of homogeneous multiphase flow is considered by literature as the limit of multiphase flow in which the rate of momentum transfer becomes very large. As a consequence, every present fluid shares the same velocity field. It is possible to extend this concept, modeling turbulence as homogeneous, also. Hence, this model gets rid of equations needed for accounting momentum transfer between phases.

Therefore:

\[
U_{\alpha} = U, \text{ for every phase } \alpha.
\]
Consequently, for each velocity component (super index $i$) a single velocity on each point of space:

$$U^i = U^i = \sum r^i U^i = U^i$$  \hspace{1cm} (3.44)

i) Mass conservation for phase $\alpha$

$$\frac{\partial}{\partial t} (r^\alpha \rho^\alpha) + \nabla \cdot (r^\alpha \rho^\alpha \mathbf{U}) = \sum_{\beta=1}^{N} (\Gamma^+_{\alpha\beta} - \Gamma^+_{\beta\alpha})$$  \hspace{1cm} (3.45)

ii) Single momentum conservation equation

Likewise the single phase case, the equations takes the following form:

$$\frac{\partial}{\partial t} (\rho \mathbf{U}) + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U} - \mu (\nabla \mathbf{U} + (\nabla \mathbf{U})^T)) = \mathbf{S}_M - \nabla p$$  \hspace{1cm} (3.46)

with variable viscosity and density, calculated by weighting each phase’s volume fraction.

$$\rho = \sum_{\alpha=1}^{N} r^\alpha \rho^\alpha$$  \hspace{1cm} (3.47)

$$\mu = \sum_{\alpha=1}^{N} r^\alpha \mu^\alpha$$  \hspace{1cm} (3.48)

However, heat transfer remains as a non-homogeneous process.

c) Free surface flow

Free surface flow is a special case of multiphase flow, where the fluids are separated by a well defined interphase. For a proper definition of the interphase, the effects of surface tension must be taken into account. Surface tension represents the force that keeps the surface together. The approach is done through the surface tension coefficient:

$$\sigma = \frac{F}{L}$$  \hspace{1cm} (3.49)
Surface tension determines wall adhesion tending to produce a curvature on the interface.

On the following, the model used to take account of phase change is described.

d) Heat transfer between phases

In order to complete the physics of the two-phase flow inside the heated tube, the interaction between phases has to be modeled. In the energy conservation equation for a phase $\alpha$, the term $Q_{\alpha\beta}$, which accounts for this mechanism depends on the model chosen, suitable for the events occurring inside the tube.

In general, the phases aren’t in thermal equilibrium; there’s a difference of temperature on the interphase surface. In order to determine the amount of heat crossing that surface a heat transfer coefficient has to be defined, along with the interphase area density.

On the equation for heat transfer between phases:

$$Q_{\alpha\beta} = h_{\alpha\beta}A_{\alpha\beta}(T_{\beta} - T_{\alpha})$$  \hspace{1cm} (3.50)

$h_{\alpha\beta}$, the general coefficient for heat transfer between phases can be correlated with Nusselt’s number. However, given that intentions of simulating mass transfer induced by heat transfer, the process must be analyzed separately at each side of the interphase. Thus, $h_{\alpha\beta}$ is calculated by means of two coefficients, individuals to each phase involved. This approach is known as the two resistance model for heat transfer between phases and is the base of the simulation for phase change in conditions out of thermal equilibrium.
Defining the flow of sensible heat towards phase $\alpha$ from the interphase as:

$$q_\alpha = h_\alpha (T_s - T_\alpha)$$  \hspace{1cm} (3.51)

and the flow of sensible heat to phase $\beta$ from the interphase as:

$$q_\beta = h_\beta (T_s - T_\beta)$$  \hspace{1cm} (3.52)

Where $h_\alpha$ and $h_\beta$ are the heat transfer coefficients for phases $\alpha$ and $\beta$, respectively, $T_s$ is the temperature at the interphase and it’s assumed as equal for both phases.

$h_\alpha$ and $h_\beta$ can be defined directly through as:

$$Nu_\alpha = \frac{h_\alpha d_p}{\lambda_\alpha}$$  \hspace{1cm} (3.53)

$$Nu_\beta = \frac{h_\beta d_p}{\lambda_\beta}$$  \hspace{1cm} (3.54)

with $d_p$ a length scale for the interphase (for the case of the Mixture Model, defined as the mixture scale; for the particle dispersion model, defined as the particles’ mean diameter).

Because the processes of heat transfer are analyzed by separated, a resistance equal to zero for one of the phases implies fixing the interphase’s temperature at a value equal to that phase’s. This strategy can be used to model a phase change in saturation conditions.
for one of the two phases. Numerically, it means modeling one phase as an energy source or sink.

In absence of mass transfer associated, the temperature on the interphase is calculated through thermal equilibrium \((q_a + q_\beta = 0)\).

For heat transfer between phases, it is determined by means of a total heat balance:

\[
Q_a + Q_\beta = 0 \quad (3.55)
\]

where:

\[
Q_a = q_a + \dot{m}_{\alpha\beta} H_{\alpha s} \quad (3.56)
\]

\[
Q_\beta = q_\beta - \dot{m}_{\alpha\beta} H_{\beta s} \quad (3.57)
\]

\(H_{\alpha s}, H_{\beta s}\) represent values for enthalpy carried to, and from phases due to phase change. In this case, a total thermal energy balance yields mass flow to phase \(\alpha\) from phase \(\beta\).

\[
\dot{m}_{\alpha\beta} = \frac{q_{a\beta} + q_{\beta\alpha}}{H_{\beta s} - H_{\alpha s}} \quad (3.58)
\]

It’s very important to define correctly the saturation temperature for the material undergoing phase change. To account for the discontinuity in enthalpy due to latent heat between phases, a slight modification in the calculation of secondary heat flow present on the denominator of \(\dot{m}_{\alpha\beta}\) is performed (Prakash, as cited in ANSYS, Inc., 2006). The adjustment consists into taking the bulk enthalpy from the outgoing phase, whereas the saturation enthalpy is taken from the incoming phase. This formulation allows the saturated phase to remain saturated, meanwhile the sub cooled phase approaches saturation. This means the following:

\[
\frac{\partial \dot{m}_{\alpha\beta}}{\partial t} > 0 \Rightarrow H_{\alpha s} = H_\alpha \text{ sat} \quad , \quad H_{\beta s} = H_\beta \quad (3.59)
\]

\[
\frac{\partial \dot{m}_{\alpha\beta}}{\partial t} < 0 \Rightarrow H_{\alpha s} = H_\alpha \quad , \quad H_{\beta s} = H_\beta \text{ sat} \quad (3.60)
\]

62
Therefore, \( H_{\beta s} - H_{as} \) results equal or greater than the latent heat:

\[
L = H_{\beta sat} - H_{as sat}
\]  

(3.61)

There are certain aspects of the simulation that must be set according to the conditions of the problem. The model for phase change doesn’t include explicitly the nucleation mechanism. Instead, both phases must be present (even in a small fraction) to trigger phase change. One way to simulate this is to set a ‘seeding’ of the gas phase inside the domain. That implies including certain volume fraction for steam on the inlet flow or setting a minimum value for that phase on the domain. This technique results especially important for phase change near the walls.

The setting of the thermal phase change model completes the equations for solving the case of phase change inside a horizontal tube.

Once described all the relevant aspects of the physics for both single and multiphase flow, on the following sections, the particular setup chosen for each case is detailed and justified.

3.3 Single phase flow simulations setup.

3.3.1 Subcooled liquid section.

![Figure 3-4: Single phase problem’s domain](image-url)
The domain for this section consists on the pipe structure (as a solid steel/copper domain), with liquid water flowing inside of it.

The chosen dimensions are based on the actual design of the absorber tube of a parabolic trough model ET-100 (Zarza et al, 2006), used on the INDITEP plant. The length considered for the domain was set in function of the discretization.

Table 3-1: Absorber tube dimensions

<table>
<thead>
<tr>
<th>External diameter (m)</th>
<th>0,07</th>
</tr>
</thead>
<tbody>
<tr>
<td>Internal diameter (m)</td>
<td>0,05</td>
</tr>
<tr>
<td>Domain length (m)</td>
<td>3,0</td>
</tr>
</tbody>
</table>

a) Pipe materials and properties

For the definition of the materials corresponding to the liquid phase inside the pipe, the equation of state IAPWS-IF97 (ANSYS Inc., 2006), an industrial standard is selected. It provides data for both sub cooled liquid, mixture and superheated steam.

For steel, the relevant properties are heat capacity (434 J/(kg K)) and thermal conductivity (60,5 W/(m K)); for copper, the values are 385 J/(kg K) y 401 W/(m K).

b) Domain’s discretization

Both solid and fluid domains are divided in small hexahedral control volumes. As commented above, the mesh has smaller elements close to the tube’s internal wall, for a better capture of gradients next to the fluid’s boundary layer.

The total number of elements is determined taking into consideration calculation time and scales of length and velocity for these cases (ANSYS Inc., 2006). Because there is also a computational resource limit (in terms of memory), the maximum domain’s length capable to be modeled is limited. Although, the geometry’s orientation and boundary
conditions make feasible to apply conditions of symmetry, simulating only one half of the domain. Thus, it is possible to save time and computational resources or to increase the total number of elements of the mesh.

Figure 3-5: Single phase case three dimensional mesh (front and isometric view)

Table 3-2: Mesh details for each case, single phase modeling

<table>
<thead>
<tr>
<th>Domain</th>
<th>Number of Elements</th>
<th>Number of Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>52.624</td>
<td>67.500</td>
</tr>
<tr>
<td>Fluid</td>
<td>322.322</td>
<td>339.300</td>
</tr>
<tr>
<td>Steel (70%)</td>
<td>26.312</td>
<td>40.500</td>
</tr>
<tr>
<td>Copper (30%)</td>
<td>26.312</td>
<td>40.500</td>
</tr>
<tr>
<td>Fluid</td>
<td>388.102</td>
<td>406.800</td>
</tr>
<tr>
<td>Steel (50%)</td>
<td>26.312</td>
<td>40.500</td>
</tr>
<tr>
<td>Copper (50%)</td>
<td>26.312</td>
<td>40.500</td>
</tr>
<tr>
<td>Fluid</td>
<td>388.102</td>
<td>406.800</td>
</tr>
<tr>
<td>Steel (30%)</td>
<td>26.312</td>
<td>40.500</td>
</tr>
<tr>
<td>Copper (70%)</td>
<td>26.312</td>
<td>40.500</td>
</tr>
<tr>
<td>Fluid</td>
<td>388.102</td>
<td>406.800</td>
</tr>
</tbody>
</table>
c) Modeling the energy flow on the pipe’s surface

While the heat flux is product of the concentrated radiation, for simulation purposes, it has been prescribed as a function of the position on the absorber’s external surface, by means of the direct normal irradiation and the parabola’s design.

![Reference angle for irradiation profile](image)

Figure 3-6: Reference angle for irradiation profile (Eck et al, 2004)

Depicted in Figure 3-6, the reference angle \( \varphi \) can be observed on a sketch for a cross section; meanwhile an approximated profile is shown in Figure 3-7.
Note that for simulating purposes, the profile was approximated by means of a normal distribution ($N(180,63)$).

d) Boundary conditions

Figure 3-7: Irradiation profile (Eck et al, 2004)

Figure 3-8: Boundary conditions

i) Inlet flow

Inlet flow was determined according to Figure 1-16, where a relation between pressure and mass flow with the flow pattern is presented. According to this
range of values, a flow of 0.4 kg/s is set on the domain’s inlet. Similarly, the reference pressure was set according to the levels tested in Eck’s research. Moreover, two levels were chosen, performing simulations at 64 and 98 bar.

Reference pressure determines saturation temperature, a critical parameter because of the single phase nature of these simulations. Thus, it was set a sufficient sub cooling which guaranteed the fluid’s temperature field to remain below saturation, thus avoiding phase change situations. For the case of 64 bar, the inlet temperature was set as 150°C, for a saturation temperature of 280°C; whereas for the 98 bar case the temperature was set as 190°C, for a saturation temperature of 310 °C.

ii) Outlet

Relative averaged pressure at the outlet was set as zero.

For the definition of the solid domain, four types of structure were simulated for each pressure: one simulation with the absorber made of steel only and three bimetallic domains of internal copper and external steel, in ratios of: 70%-30%; 50%-50% y 30%-70%, in order to be able to obtain information on the influence of copper’s addition to the structure.

iii) Heat flux

Heat flux is defined on the external surface by the adjusted function as commented above.

iv) Absorber ends:

The absorber’s ends present a zero gradient condition.

v) Symmetry condition
Finally, given that only half of the domain is modeled, symmetry conditions are set on the surfaces forming the symmetry plane.

vi) Initial conditions

Initial temperature for all domains is set equal to inlet temperature. An initial velocity field for the fluid is provided by previously solving the flow without heat flux on the external wall.

Regarding the simulation numerics, transport equations are solved with a second order precision scheme, using a timestep automatically calculated (for both solid and fluid domains) for the advance of the simulation, delivering a steady state solution.

3.3.2 Superheated steam section.

Simulations performed under this assumption present an inlet temperature greater than saturation’s for either reference pressure. Hence, the working fluid is dry water steam at all times. Modeled geometry and dimensions are identical to those of the sub cooled liquid case, as well as the bimetallic pairs simulated. External heat flux remains the same, only varying the inlet temperature for the same two reference pressures. For the 64 bar case, the inlet temperature was set as 281°C, meanwhile, for the 98 bar case, temperature was 311°C. The rest of the boundary conditions are modeled in the same fashion as in the sub cooled case, with the exception of the initial conditions, obtained by solving the dry steam flow at isothermal conditions for the same mass flow used in the sub cooled fluid case.

3.3.3 Convergence criteria for single phase simulations.

The convergence rules to end single phase simulations are based on the RMS value for the residual vector, obtained through iteration of the linear system, product of the domain and equations discretization. A value of 0.0001 is recommended (ANSYS Inc., 2006). Additionally the condition of a maximum unbalance for mass and energy
quantities of 0.1% is also set. The solution is considered as converged when the cross-area averaged outlet temperature has reached a steady state.

3.4 Two phase stratified flow simulation setup.

First, the chosen dimensions, meaning diameter and wall thickness, as domain extension are discussed.

The starting point for the selection of a domain suitable for the stratified flow simulations was the research carried out by Flores (2007). There, Flores work with tubes of reduced length (3 m) at low mass flows (up to 120 kg/h). With this kind of setup it is possible to observe steam generation in short segments of pipe. A similar criterion applies when conducting a numerical simulation based research.

Attempting to simulate the real length of the evaporator is unfeasible, due to the enormous difference in scales (in both time and space) existing on the domain. In previous chapters it has been seen that the collector field has an extension on the order of thousand meters; while phenomena occurring inside the absorber causing the fluid to change phase are on the order of one thousandth of a meter and even less (Collier & Thome, 2001). Those mechanisms, working at such relatively small scales are of interest for assessing the heat transfer in presence of phase change. Thus, in order to capture such effects and at the same time model a realistic length of tube would imply a number of elements such that the solution of the discretized problem would not be possible with today’s computational resources. In order to overcome this difficulty, a reduced length is modeled, keeping the problem’s size manageable.

So, there is a first factor to take into account: the domain’s dimensions, and along with it a second one: the level of discretization. Promptly, a third factor arises: the heat flux used in previous cases causes the liquid to boil over the length of the real absorber. Thus, a reduced domain in which such heat flux is applied, won’t achieve a significant
level of generated steam, if any. Then, a new heat flux must be set accordingly to the other factors.

A fifth factor: the temperature boundary condition at the inlet must also be discussed because of the effect on the thermal equilibrium of the phases.

While studying the problem of phase change, it would have been much simpler to deal with a saturated flow at the domain’s inlet (either $x = 0$ or a prescribed level of stratification at saturation temperature for both fluids). However the real problem isn’t in thermal equilibrium, meaning that inside the domain, sub cooled liquid and super heated steam coexists. Thus, to take into account this factor, a prescribed mass flow of sub cooled liquid (having a prescribed degree of sub cooling) must be set at the inlet.

In order to achieve this thesis’ objectives, meaning a comparative study based on a developed methodology to assess the performance of absorber tubes, the following criteria have been defined, thus characterizing the results to be obtained, namely:

- Steam must be generated, developing a stratified flow pattern inside the pipe.
- Saturation temperature must be respected near the interphase.
- Steep thermal gradients near the fluids interphase are expected on the metallic structure.
- Results are expected to be sensitive to the pipe’s structure.

In summary, the problem’s domains and boundary conditions were set from the literature such that it could be possible to study conjugated heat transfer between solid and fluid domains, with a fluid domain comprised by two phases, interacting through a thermal phase change mechanism that develops a growing interphase between them and the pipe’s wall.

The mixtures of materials for the bimetallic absorbers are the same as the used on the single phase cases: 70%-30%; 50%-50% y 30%-70%, while the tube’s dimensions are listed below.
Table 3-3: Absorber tube dimensions (two phase case)

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>External diameter (m)</td>
<td>0.0268</td>
</tr>
<tr>
<td>Internal diameter (m)</td>
<td>0.0216</td>
</tr>
<tr>
<td>Domain length (m)</td>
<td>1.0</td>
</tr>
</tbody>
</table>

a) Domain’s discretization

The process of discretization for the multiphase simulation is similar to the setup for the single phase case. One of the differences is the increase of the amount of elements used to discretize the model because of this case’s physics involving more phenomena. Finally, the number of elements is limited by computational resources.

b) Fluid selection and properties

The Redlich Kwong equation of state was used to model boiling in both equilibrium and non-equilibrium conditions. For materials that use the Redlich Kwong equation of state ANSYS CFX-11 approximates the vapor pressure by means of an equation including critical pressure and temperature.
Table 3-4: Mesh details for each case two phase modeling

<table>
<thead>
<tr>
<th>Domain</th>
<th>Number of Elements</th>
<th>Number of Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>111.720</td>
<td>136.800</td>
</tr>
<tr>
<td>Fluid</td>
<td>560.994</td>
<td>585.200</td>
</tr>
<tr>
<td>Steel (70%)</td>
<td>55.888</td>
<td>85.500</td>
</tr>
<tr>
<td>Copper (30%)</td>
<td>55.888</td>
<td>85.500</td>
</tr>
<tr>
<td>Fluid</td>
<td>757.482</td>
<td>788.500</td>
</tr>
<tr>
<td>Steel (50%)</td>
<td>44.688</td>
<td>68.400</td>
</tr>
<tr>
<td>Copper (50%)</td>
<td>44.688</td>
<td>68.400</td>
</tr>
<tr>
<td>Fluid</td>
<td>560.994</td>
<td>585.200</td>
</tr>
<tr>
<td>Steel (30%)</td>
<td>44.688</td>
<td>68.400</td>
</tr>
<tr>
<td>Copper (70%)</td>
<td>44.688</td>
<td>68.400</td>
</tr>
<tr>
<td>Fluid</td>
<td>560.994</td>
<td>585.200</td>
</tr>
</tbody>
</table>

3.4.1 Dynamics setup for stratified flow.

According to the tests carried out by Almanza, reference pressure is set as 15 bar.

First, the approach on the modeling of both phases is discussed. Because of the differences in velocity and the process of heat transfer itself, the multiphase problem is treated as separated flow, solving one set of governing equations for each phase (Chull Jo & Kim, 2004).

For the morphology of the phases inside the pipe, in a stratified pattern, the adequate approach is to consider each phase as continuous (a continuous phase is considered a permanently connected region). Additionally, as it was treated when describing the phase change model, a minimum steam volume fraction is set inside the domain, aiding the gaseous phase generation; the value is set as 0.001.
Due to high Reynolds number present on the fluid domain, both phases are modeled as turbulent, the use of a model similar to $k-\varepsilon$ approach is recommended: the $k-\omega$ model, suggested in cases where near-wall effects are important, translated into greater numerical stability when solving the linear systems (Ribeiro, 2008). However, testing this model resulted in instability of the solution. The $k-\varepsilon$ model remained as the selected choice for modeling turbulence.

To model the interaction between phases, a surface tension coefficient is provided $\sigma = 0.039$ N/m, a pressure dependant value.

For interphase transfer, the mixture model is chosen, which treats both phases symmetrically. The model requires both phases to be treated as continuous and to provide an interphase length scale, in order to calculate the surface area per unit volume:

$$A_{\alpha\beta} = \frac{r_{\alpha\beta}}{d_{\alpha\beta}}$$  \hspace{1cm} (3.62)

where $d_{\alpha\beta}$ is the interphase length scale. According to Frank (2003), the suggested value for the length scale is 0.001.

Momentum transfer is modeled through drag force,

$$F_{D,\alpha} = C_D \rho_{\alpha\beta} A_{\alpha\beta} (U_\beta - U_\alpha)$$  \hspace{1cm} (3.63)

where the default value is maintained (ANSYS Inc., 2006).

$$C_D = 0.44$$  \hspace{1cm} (3.64)

in addition:

$$\rho_{\alpha\beta} = r_\alpha \rho_\beta + r_\beta \rho_\alpha$$  \hspace{1cm} (3.65)
Heat transfer simulation is modeled by means of the thermal energy equation for each phase, which models enthalpy transport through the domain. Kinetic effects are neglected in the energy equation.

For heat transfer between phases, the two resistance model is used. The chosen method needs the definition of a Nusselt number for the liquid phase, based on the Dittus-Boelter correlation, assuming the liquid fraction fills the tube (Odeh et al, 1998):

\[
Nu_L = 0.023 \left( \frac{G(1-x)D_i}{\mu_L} \right)^{0.8} Pr_L^{0.4}
\]

(3.66)

With:

\[
G = \frac{\dot{m}}{A}
\]

(3.67)

And \(D_i\) represents the pipe’s internal diameter.

Where the data are taken from values of liquid near saturation at reference pressure, namely:

\[
\mu_L = 1.34 \cdot 10^{-4} Pa \cdot s
\]

(3.68)

\[
Pr_L = 0.906
\]

(3.69)

\[
\rho_L = 865 \text{ kg/m}^3
\]

(3.70)

Varying the steam fraction \(x\), the following graphic is generated:
The red shaded region represents the expected range of steam fraction to be obtained inside the domain. Because a single value for the Nusselt number must be provided on the problem’s setup, an averaged value of 260 is set based on previous numerical experiments of steam generation, where only the fluid domain was solved for several combinations of inlet mass flow and external heat flux.

Meanwhile, a zero-resistance approach is selected for the gaseous phase.

3.4.2 Setup of boundary conditions for the fluid domain.

a) Fluid Inlet

Inlet flow is set as 0.40 kg/s, a value which according to Eck (2004) should guarantee a stratified flow pattern. The fluid’s temperature is set as 170°C, which implies a subcooling degree respect to the saturation temperature (198.3°C) at reference pressure.

Turbulence is assumed as low intensity at the inlet (1%).

The composition of the inlet flow, expressed in terms of each phase’s volume fraction is completely liquid.

b) Fluid Outlet
The area-averaged static pressure at the outlet cross section equals zero.

c) Symmetry

Symmetry is represented by the XY plane (with \( z = 0 \)), cutting the pipe in half.

d) Solid-fluid interphase

An interphase boundary condition is defined in both domains, solid and fluid. Thanks to the mesh definition, the location of the nodes on the fluid side of the interphase, matches the location of those belonging to the solid side.

### 3.4.3 Boundary conditions setup for the solid domain.

The setting for solid domain is simpler, being only necessary to define conditions of symmetry at XY plane (as for the liquid domain, at \( z = 0 \)). For regions at each end of the tube, a zero gradient condition is established.

As in the single-phase case, the external wall has a prescribed heat flow. As commented on previous section, the flux is different from that used in the single phase case. To define it, the reference was taken from research based on modeling of vertical pipes present in some nuclear reactor designs, as presented in the works of Chull (2004) and in other CFD modeling approaches to this geometries (Egorov & Menter, 2004). Finally, a uniform value of 260,000 W/m\(^2\) was applied to the pipe’s external wall.

### 3.4.4 Initial conditions setup.

The problem’s set of initial values are obtained through solving the single phase, isothermal problem on steady state. The domain’s temperatures are set equal to the value for inlet flow: 170°C.
Hence, a developed velocity field and a homogeneous temperature field are obtained. Also, turbulent initial quantities are also obtained for the turbulence model (turbulence kinetic energy, turbulent eddy frequency).

### 3.4.5 Calculation procedure.

The numeric scheme used for solving the advection related terms is the upwind scheme, first order of precision. It has quite a robust performance; however, the results are conditioned by numerical diffusion (false diffusion).

Once obtained the results with the upwind scheme, these are used as the initial set for a new simulation using a high resolution scheme, which varies dynamically. Thus, in lower gradients zones, residual oscillation is prevented, while the precision is increased at lower gradients regions.

Finally, the simulation is carried out as a transient simulation, to explicitly use the time step variation as a “speeding up” parameter, to guide a steady-state solution. To achieve that, the transient terms of the discretized equation are treated by the first order Euler scheme.

### 3.4.6 Convergence criteria for the multiphase simulations.

For multiphase flows, is much harder to attain lower RMS levels of residuals, comparing with a single phase case. While the minimization of the value of the residuals is always pursued for every equation solved (mass, momentum, thermal energy and volume fraction), is not always possible to reduce as much as in the single phase case (ANSYS Inc., 2006). Because of this, other additional criteria are established: the outflows of steam and water are monitored, until achieving stabilization, along with the fluid domain’s imbalance of mass and energy at a maximum allowed of 1%.

Nevertheless, for all the systems solved, a maximum level for the RMS values is set at 0,0001.
3.5 Additional requirements of the simulations.

On what follows, the memory and processing requirements for the simulation are discussed.

a) Mesh size

The first relationship between use of resources (CPU time consumption and memory) is given by the level of the discretization of the problem, meaning the number of balances to be performed. For the studied case, where hexahedral elements were used for both domains, the number of nodes (i.e., the “place” where the material properties and information of the solution are stored) is in close proportion (in practice, a 1:1 ratio) with the number of elements. In particular, the domains of the multiphase case totalize over 650,000 elements.

The incremental factors mentioned below, apply to a base case of a single phase fluid in laminar regime, without heat transfer.

b) Turbulence

Both k-epsilon and k-omega models add two equations to the Navier-Stokes system. An increment of 50% in CPU time is expected, while the use of memory is modest.

c) Energy

Solving total energy equation or thermal energy equation require solving for an additional scalar: temperature. The increment of CPU time is close to 30%.

Conjugate heat transfer with solid domains: in solid regions, only energy equation is solved. Therefore, the increase in processing time and memory expense are little.

d) Multiphase flow:

Non-homogeneous: for a two-phase flow solved under the assumptions of the mixture model, memory requirements increment by a factor of 2.15 while CPU time also rises by a factor of 2.25.
4. SIMULATION RESULTS AND DISCUSSION

In this chapter the results obtained from numerical simulation of selected regions of the absorber are examined. The first part comprises the study of the solution of the single phase problem, namely the segments of sub cooled liquid and superheated steam. According to the setup described on the methodology, the solutions for each case are analyzed. The features of the solutions are also compared to the data available on the literature in order to validate the behavior of the solution. The rest of the chapter deals with the obtained solution of the two-phase simulation. Similar as the first part, the analysis focuses on the comparison of each case simulated (i.e. the different pipe wall composition). Also, the results are confronted with experimental data, when available. It must be taken into account that such analysis hasn’t been carried analytically at this scale so present data is scarce.

The structure of the analysis starts with a base case, represented by the results of the single material tube, made completely out of steel. The features of that case’s flow are first presented; after this discussion, the results for the solid domain are commented also. Afterwards, a comparative analysis between the base case and the bimetallic configurations is carried out.

It must be pointed out that the results displayed on this chapter center on the analysis of the methodology as a comparative tool for this kind of problems, aiming on the design of improved absorbers.

4.1 Single phase simulation results – sub cooled liquid.

On what follows, the results for the simulation of sub cooled flow are displayed, according to the structure commented above. It should also be noted that two reference pressure levels were solved (98 and 64 bar), thus generating two sets of results.
4.1.1 Base case analysis.

The result, as it was expected is highly dependent on the heat flux profile on the external surface, depicted on Figure 4-1.

![Heat flux profile](image)

Figure 4-1: Heat flux profile

Also, due to density variations and buoyancy forces causing the vertical translation of warmer fluid to upper regions of the pipe, the flow presents a minor rotating component, as depicted on Figure 4-2 for the velocity vector.
For a more quantitative analysis, data were sampled at several regions over the domain, represented as curves defined at the intersection of a cross section plane created every 0.6 m and the domain’s boundaries, as depicted on Figure 4-3.
By plotting the values for temperature and velocity for each centerline (as showed on the detail of Figure 4-3), both the effects of the heat flux profile and buoyancy can be observed. The legend on each curve tells the distance (measured on the z axis) from the domain’s inlet.

![Velocity plots for subcooled liquid (64 bar)](image)

Figure 4-4: Velocity plots for subcooled liquid (64 bar)

The first thing to mention is that, as for velocity, the results are practically independent of the reference pressure considered. Regarding the shape of the curves, as showed on Figure 4-4, it can be seen how the initially fully developed parabolic profile (from the solution the isothermal problem) changes into an eccentric set of curves, as the vector plot and the streamlines suggested. The development of this new profile is fairly quick, with the curve at a distance of 1.2 m already taken a characteristic shape. The maximum velocity is found below the tube’s axis.

In a similar fashion, Temperature profiles’ shape is not altered by pressure, observing only a difference on the magnitudes, not in the shape of each curve, with an identical behavior. Temperature profiles begin with the curve at $z = 0$, depicting a constant value at the inlet. As the distance increases, the profile develops asymmetrically, due to the rising of warmer liquid, a phenomenon in agreement with the observations of Sommer (1996) on horizontal flow inside pipes influenced by buoyancy forces. As it
should be expected, the maximum values for temperature are found on the lower and upper region, given the heat flux distribution.

Figure 4-5: Temperature Plots for subcooled liquid (64 bar)

Temperature contour plots are used to describe the distribution of the temperature on the tube structure, on the cross sections depicted on Figure 4-6. Based primarily on this plots is that the effectiveness of bimetallic pipes is proved.
4.1.2 Comparison of the bimetallic results against the base case

Flow analysis of bimetallic tubes is based both on temperature profiles plots and on the contour plots as depicted for the base case. As in the single material tube, the pressure
levels didn’t show difference in the shape of the curves plotted, or in the images obtained. Certainly, the value of the temperature for each pressure level differs. Thus, only the results for the 64 bar pressure are studied.

The nomenclature used on these plots is as follows:

The legend $z = 0.6 \, m \, 70/30$ represents the information on the center line at 0.6 meters from the inlet, for the case where the pipe is composed by two concentric tubes, giving a wall thickness made of 70% steel (external tube) and a 30% copper (internal wall, in contact with the flow).

It is possible to observe a difference to the base case; while toward the center the curves are similar to each other, the values change when approaching the wall. Increasing the ratio of copper for on pipe’s thickness causes the temperatures on the lower region to decrease, while the values near the top increases accordingly. This effect is more easily observed in plots nearer to the outlet. Moreover, the upper end of the curve for the steel tube is distanced from the ‘bimetallic profiles’, showing a redistribution of the temperature of the flow.

Intending to clarify the effect of the pipe’s structure on the flow’s temperature, a plot showing the maximum temperature difference on the flow per case and pressure is presented on Figure 4-7.
The behavior of the maximum temperature difference (taken on the symmetry plane, as depicted in Figure 4-7) shows a constant drop with every increase on the copper fraction given the set of constraints and simplifications of the model analyzed. Once established the temperature redistribution on the pipe due to copper inclusion, the cause of this effect is discussed: the variation on the temperature fields as a result of bimetallic tubes.

For the thermal analysis of the pipe wall the temperature contour plots on every cross section and on the entire external pipe surface for all cases are compared to those corresponding to the base case. In order to complement the analysis and to compare with data from literature, at every cross section the external temperature is plotted in function of the angle $\varphi$, as defined in previous chapters. This will illustrate the effect on the angular temperature gradient.

Figure 4-7: Maximum temperature difference, sub cooled liquid
Figure 4-8: Cross section temperature contour plots for the 70/30 case

Figure 4-9: Cross section temperature contour plots for the 50/50 case
Observing the contour plots case by case, it is immediate the change on the temperature distribution, especially for the lower zone of the absorber, which presents the highest heat flux entering the domain.

It should be noted that the scale defines twenty temperature contours. With regard to the base case (Figure 4-6), each bimetallic result shows a smaller number of contours, thus representing a tendency to reduce temperature gradients on the pipe.

On the figures plotter above, it is evident the temperature reduction, towards the lower part of the tube, in contrast with the steel absorber’s steep thermal gradients even at small distance from the inlet.

The following set of plots show the temperature on the external surface, related to φ. Each plot represents one of the cross sections and each curve, a type of absorber, including the base case (noted as ‘100/0’).
As verified on the analysis of the flow temperature, the temperature distribution case by case happens to be very similar. In this case, $\phi$ as increases (up to a maximum of 180°, the lower point of the cross section) the absorber’s temperature raises, according with the heat flux distribution. However, with 30 per cent of the wall thickness made out of copper, certain flatness occurs with increasing additions of copper, even at the first cross section (0.6m from the inlet). On further cross sections this trend is accentuated, with the bimetallic cases curves’ ends more and more distanced from the base case. At smaller $\phi$, the differences aren’t so accentuated, although the distance between the whole set of bimetallic cases (presenting very similar temperatures between them) and the base case...
is sustained. On the other hand, when $\phi$ approaches 180°, the commented tendency heightens. Nevertheless, the temperature difference between the each bimetallic curve is always smaller than the difference between the base case and the first bimetallic case (namely, the 70/30 case). This suggests that the effectiveness of the copper does not remain constant.

All in all, for every sub cooled liquid case simulated, the maximum temperature difference on the absorber is reduced, as suggested by the chart displayed on Figure 4-12, showing the maximum temperature difference measured on the flow.

The chart showed below delivers a summary of the simulations for sub cooled liquid (at 64 bar), for the maximum temperature difference, this time on the solid domain, per each case.

![Figure 4-12: Maximum temperature difference in solid domain (subcooled liquid)](image-url)
From the chart, it can be observed that for the single material case, the maximum temperature difference reaches 20°C, section by section. Bimetallic pipes present a distinct tendency to the reduction of this value, showing a sudden drop even for the case with less copper and on the section closer to the inlet, reaching 14°C. Greater additions of copper produce a larger effect, but some decreased effectiveness can be observed through the analysis of this set of results. Regardless, the effect of the copper in the homogenization of the temperature field of the absorber can be asserted.

It must be pointed out that the observed behavior of the bimetallic structures simulated is based only on the results of this set of models. This includes the assumptions made for the materials (both fluid and solids), boundary conditions and the dynamics of the flow field.

4.2 Single phase simulation results – superheated steam

The analysis for the superheated steam simulations follows the same sequence discussed in the previous set of simulations, making use of the same criteria to plot relevant variables.

4.2.1 Base case analysis

For the base case, like in the liquid flow analysis, contour plots and charts relative to the domains temperature and velocity fields are presented. As commented in literature, steam has a convective heat transfer coefficient lower than the liquid phase (Eck et al, 2004). Therefore, the temperature differences are expected to be higher than in the previous section given the reduced capability of the steam to remove heat form the pipe wall.

Unlike the fluid simulations, the steam flow only develops an axial velocity component. Hence, heated steam remains on the lower part of the tube.
When the results are taken on the middle line at the cross sections, the velocity profile appears much more symmetric, as depicted in Figure 4-13.

![Figure 4-13: Velocity profile per cross section (64 bar)](image)

Note that higher velocities can still be observed below the pipe’s axis. It should also be noted that the mass flow between liquid and steam simulations is maintained. Because of this, the steam flow develops higher velocities than in the liquid case.

As for the temperature profiles, they show a distinctive shape. From Figure 4-14, maximum temperatures are to be found on the lower region. Hence, due to the heat flux profile, the minimum values can be observed well above the pipe’s axis.

![Figure 4-14: Temperature profile per cross section (64 bar)](image)
As in the liquid case, the results for superheated steam also show a steep temperature gradient on the tube material, showing differences of over 30°C, as observed on Figure 4-15.
4.2.2 Comparison of the bimetallic results against the base case

Figure 4-14 showed once more that while the values on the temperature field are differentiated by each reference pressure, the distribution of temperatures is basically the same. Bimetallic results also present almost no difference between reference pressures. Hence, and in the same fashion as in the liquid simulations, the results showed below correspond to the 64 bar reference pressure.

As commented above, temperature profiles at the cross section for the base case present certain asymmetry, according with the shape of the heat flux distribution. The simulation of a bimetallic domain shows how, section by section the profile tends to adopt a symmetric shape and causing a redistribution of the temperatures by lowering the high temperatures at the bottom and increasing the temperatures at the top. This behavior becomes more evident in the last cross sections, as observed in the upper part of the bimetallic curves regarding the base case. On the other end, it is possible to see the bimetallic curves ends further and further from the end of the curve representing the base case.

![Graph showing temperature difference](image)

Figure 4-16: Maximum temperature difference, superheated steam
Probably the only aspect in which the reference pressure presents a difference in the trend of the results for superheated steam is the maximum temperature difference observed on the flow.

Although for both pressure levels the results obtained show a decrease in the temperature difference, at higher pressure, the values are lower than for 64 bar. The more homogeneous distribution of temperatures for higher pressures can be explained through the decreasing ability of the steam flow to remove heat from the walls; hence the flow temperature would not vary significantly.

When analyzing the pipe’s temperature field for the superheated steam case, the effectiveness of the bimetallic structures in reducing the thermal gradients is proved once more. Depicted in Figure 4-17 though Figure 4-19 is possible to see a decreasing number of temperature contours with increasing copper addition indicating a reduced temperature gradient, when compared to the base case.

Figure 4-17: Cross section temperature contour plots for the 70/30 case
Figure 4-18: Cross section temperature contour plots for the 50/50 case

Figure 4-19: Cross section temperature contour plots for the 30/70 case.
As with liquid simulations, maximum temperatures are reduced and the distribution tends to be more uniform, reducing the areas with high temperature gradients, especially at the lower part of the absorber. This effect is regardless of the amount of copper on the wall. The temperature charts comparing every bimetallic case with the base case (related to the $\phi$ angle) show quantitatively this effect in superheated flow.

Figure 4-20: Set of temperature profile plots (external surface), superheated steam
The characteristic ‘S’ shape appears once again on the temperature charts. Compared to the liquid case, the steam flow shows a larger temperature difference in each section of the tube. This observed effect is in agreement with the literature, about the occurrence of high temperatures when the absorber is dried. However the positive effect repeats in this case. A bimetallic wall reduces the temperature difference in the cross section. Although the temperature drop with increased copper thickness decreases quickly, it is possible to achieve a similar difference as those observed in the sub cooled liquid cases.

Figure 4-21: Maximum temperature difference at external pipe surface (superheated steam)

The chart in Figure 4-21 summarizes the results (for the 64 bar case) in terms of the maximum temperature difference found on the external pipe surface.
As commented above, it should be noted the great reduction of temperature between the base case and the 70/30 case. While in the liquid simulations the reduction was of about 7°C, steam flow simulations show reductions of twice this magnitude.

4.3 Two-phase simulation results – stratified flow

This section begins presenting the results of the simulation defined in Chapter 3 for the steam generating section. The data is analyzed taken into account the ‘objectives’ or ‘requirements’ defined for two-phase simulation, being:

- Steam generation in a stratified flow pattern.
- Saturation temperature must be respected near the interphase.
- Steep thermal gradients near the fluids interphase are expected on the metallic structure.
- Results are expected to be sensitive to the pipe’s structure.

4.3.1 Base case analysis

a) Fluid domain results

The simulation of steam generation was achieved, with a well defined interphase developing on the pipe’s superior region. In order to count with a larger amount of steam, the simulated total length was comprised of two segments of 1 m each.

The velocity fields will be described prior to the thermal analysis of the results.
Figure 4-22: Liquid superficial velocity vector plot (at symmetry plane)

Figure 4-23: Liquid superficial velocity vector plot (at plane perpendicular to symmetry plane)
The velocity fields are quite regular, as it can be seen on Figure 4-22 and Figure 4-23, with the velocity increasing when approaching the center of the domain.

![Superficial velocity vector plot](image)

Figure 4-24: Superficial velocity vector plot (at cross plane 0.8 m from the inlet, tangential projection)

When the liquid’s superficial velocity is plotted on a cross section, a distinctive pattern can be observed, as Figure 4-24 shows. This rotational component of the velocity implies the rise of warmer liquid to zones of higher temperature, as the pipe’s internal surface temperature field suggests. This velocity field inside pipes can also be found in previous works on CFD simulations applied to phase change inside pipes (Chull & Kim, 2004).
It should be pointed out that for two phase simulations, the temperature field is separated on two variables, one for each phase. Due to this, temperature fields are defined all over the fluid domain for both materials (liquid and gas), even in regions where they aren’t present.

Figure 4-26 shows the values for the liquid’s temperature, being possible to see the progressive heating of the fluid. This result shows a certain level of sub cooling until the pipe’s outlet, coexisting areas of liquid below saturation temperature (specifically near the pipe’s axis) at locations where steam above saturation condition accumulates on top of the pipe. Note that when the steam generation starts, a sudden rise appears on the liquid temperature field.

As for the ‘objectives’ mentioned before, a relationship is observed between the interphase level and the saturation temperature (198,3°C). Although it is not always the exact value as given by tables, it represents (as seen on the figures) a division of the temperature field, being very similar to the division of the mass fraction field.
Figure 4-26a-b: Temperature contour plot at cross sections and liquid-steam interphase every 0.2 m

As for steam temperature on the symmetry plane, a strange behavior was observed as steam is generated into the domain, with a sudden temperature rising up to 200°C
approximately. After this jump, it stabilizes showing a distinct difference between the areas where the gas phase is actually generated and accumulates.

There is in fact a certain overheating of the generated steam far above saturation temperature; meanwhile the liquid shows strong subcooling in the core region, as depicted in detail on the Appendix A. Previous works on theoretical analysis of two-phase flow also show an overestimation of temperatures (Martínez & Almanza, 2007). In the cases simulated, given the models chosen and the simplifications applied to the models’ definition, the main cause proposed for the overestimation of the temperatures is the model used to simulate the phase change. As mentioned in the methodology chapter, the lack of generalized models for nucleated and convective boiling forces the use of simplified models. In this case, the model used doesn’t model the nucleation effect. This restriction causes the calculated temperatures to be higher.

b) Solid domain results

It is possible to observe in Figure 4-27 how even with a uniform heat flux (such as the one chosen) for the two-phase simulations, the effects of stratification shape the temperature field on the internal and external surface of the pipe.
Steam appears after a certain distance, due to the initial sub cooling of the liquid; however, its effects are clear on the absorber, generating an area of high temperature gradients, represented by the increasing number of temperature contours near the surface that separates both phases inside the pipe.
It should be pointed out that whether the magnitudes of the maximum temperature difference in the structures simulated is much larger than the one observed on experimental devices such as the one presented on Almanza’s research (2007), for different boundary conditions, the focus of the study was to present a comparative study on the behavior of bimetallic tubes by the application of the methodology devised for the simulation of phase change in a stratified pattern.

On the following section, the different bimetallic structures results are compared, showing the effects of adding a second material on the absorber’s structure, especially into the reduction of maximum temperatures on the pipe material.

### 4.3.2 Comparison between base case and bimetallic absorbers

The analysis of the results will be based on the main purposes of this work, applying a methodology for the simulation of stratified flows generated due to thermal phase change, and to present a selection criterion for tube materials on the absorber element. Thus, these results, along with the simulations for the single phase section (liquid and steam) give valuable information about the use of new designs for the absorber.

For comparison purposes in the figures presented in this section, all four simulated domains (base case plus three bimetallic structures) are displayed at the same time.

a) Fluid domain results

The amount of steam generated at the outlet is similar in every case, except for the single metal case where the simulations gave a slightly higher value. In regards of the dynamics of the two phase flow (velocity fields for liquid and steam), they didn’t show any significant variation when changing the absorber structure from the base case results.
The temperature contour plots obtained for liquid and steam at every cross section can be found on Appendix A, comparing the four absorbers simulated. The following analysis is based on the figures presented there.

Table 4-1: Steam volume fraction at half distance from the inlet.

<table>
<thead>
<tr>
<th>Case</th>
<th>Average Steam Volume Fraction at half domain</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Steel-Copper)</td>
<td></td>
</tr>
<tr>
<td>100%-0%</td>
<td>0,0324</td>
</tr>
<tr>
<td>70%-30%</td>
<td>0,0277</td>
</tr>
<tr>
<td>50%-50%</td>
<td>0,0268</td>
</tr>
<tr>
<td>30%-70%</td>
<td>0,0265</td>
</tr>
</tbody>
</table>

The liquid temperature contours show a similar evolution between the different cases, with higher values in the 100/0 case, particularly in the zone near the interphase. The steam temperature field at the cross sections depicts the most notorious difference between the solid structures, with a higher superheating of the steam over the interphase. Between the assumptions made in the design of the methodology, the choice of the model for simulating phase change is the strongest cause for the phases to reach such high temperatures, especially for the steam, due to its inability to model nucleated boiling at the walls. This is one of the main features that must be developed on the next steps on this line of research.

For the cases here simulated, adding copper to the pipe’s wall doesn’t show a dramatic effect on the amount of steam generated; instead, it is the steam in a stratified pattern that appears to be affecting the absorber temperature, being the drying phenomenon at the top the main cause of the temperature gradients. Once again, it is very important to
note that this analysis is based on the results of the current simulation, and are subject to
its assumptions and simplifications.

b) Solid domain results

Depicted in Figure 4-28, the temperature field on the external surface is greatly
influenced by the stratified pattern. The mentioned figure shows clearly the effect of
adding copper on the absorber. In fact, every bimetallic absorber shows a smaller
area of maximum temperature according to the scale on the pictures.
Figure 4-28a-b: Temperature contour plot, absorber external surface (all domains)
Comparatively, the temperatures near the dried area (part of the inner surface covered by steam) are much lower even in the case with less copper (i.e. 70/30), as Figure 4-29 shows.

Figure 4-29: Temperature contour plot, absorber internal surface (all domains)
Furthermore, the set of temperature contour plots on the cross sections (again, taken every 0.4m) show the effect of copper addition. Steep temperature gradients, represented on the images by the distance between successive temperature contours appear near the zone near the interphase, and are attenuated in the bimetallic cases.

In order to illustrate better the thermal response of the simulated absorbers, a set of charts plotting the temperature on the pipe’s exterior in function of the $\phi$ angle (defined the same as in the single phase simulations) are presented. In these, it is possible to clarify the influence of the copper on the temperature distribution on the cross section of the absorber, which reflects on the advantage of building the pipe with a larger copper/steel ratio.
c) 

![Diagram c](image)

**Temperature [°C]**

![Graph c](image)

1 - φ / P

---

d) 

![Diagram d](image)

**Temperature [°C]**

![Graph d](image)

1 - φ / P

e) 

![Diagram e](image)

**Temperature [°C]**

![Graph e](image)

1 - φ / P

---

113
Note that in the charts showed in Figure 4-30 the horizontal axis is defined as \((1 - \varphi / P_i)\). The temperature in the external surface (given the boundary conditions) presents a sudden rise near the area corresponding to the interphase of the fluids (near 0.7 on the horizontal axis), and rising towards the top of the absorber, where the generated steam flows. This temperature distribution should appear as a logical consequence of a constant heat flux on the external surface of the absorber as a boundary condition and a flow of fluid with poor capabilities of removing heat, when compared to the liquid phase. However, this results show that unlike in the single phase case, the effect of a higher ratio of copper on the absorber’s wall weakens; the 30/70 absorber presents almost no difference between the 70/30 case regarding temperature reductions. This result should be considered of interest for the analysis, but also be considered given the simplifications of the model, taking into account the several assumptions made to define this methodology.

![Figure 4-30 a-f: Set of temperature profile plots (external surface)](image-url)
Finally, in Figure 4-31, the maximum temperature per section is displayed. It is possible to see that meanwhile the temperature difference increases along with the axial distance, the bimetallic tubes show a much lower value, slightly decreasing with the amount of copper that makes up the wall increases.

**4.4 Summary of the results obtained**

The results for the three sections simulated (only liquid, liquid-steam and only steam) delivered information about the advantages of using bimetallic absorbers which reduce the maximum temperature difference on the external surface, a critical parameter for the performance of these structures, as it was discussed already on Chapter 1. While the results are not perfectly adjusted to the experimental results, the behavior of the solutions can be addressed as accurate, when comparing to the results of Flores (2007). Figure 4-32 shows analytic results for the simulation of a section of tube with a level of stratification of 50% and a heat flux only focused on the bottom part of the tube.
When comparing this research results to the set of charts on Figure 4-30, several differences exist, such as the heat flux on the exterior and the level of stratification set. This causes the difference on the curves orientation: in Flores’, the temperature increases when approaching the bottom, whereas our simulations show growing temperatures near the dried region of the absorber. Nonetheless, Flores’ results point out a key fact, both on analytic and experimental levels: bimetallic absorbers actually help to narrow temperature differences on the absorber’s cross section, and that magnitude of this temperature drop doesn’t appear constant with every addition of copper to the pipe’s structure. The results obtained thanks to the methodology here presented show temperature reductions for every bimetallic absorber, as well as the commented trend on relation between the amount of copper in the wall and the reduction of the maximum temperature difference.

The reasons for having a validation orientated to verify the response of the designed absorbers to stratification conditions in qualitative terms rather than a direct comparison between experimental data and performed simulations lie on the fact that this investigation centered on the design of a methodology for assessment of bimetallic absorbers. Indeed, this research generated such methodology; the immediate conclusion that arose was that it is possible to continue the line of research in order to improve the quality of the results. The design of this new methodology implied a number of simplifications and assumptions such as boundary conditions different to those appearing on experimental facilities.

When comparing with the curves obtained for single phase and two-phase simulations, it can be seen that while the range of temperatures is different, the behavior of the solution is very similar. It should be regarded though, that the results of the simulations give guidelines and criteria for the final design process.
Focusing on the results of the two-phase section, the steam generation has been accomplished jointly with heat transfer on a solid domain, which represents a step forward in this line of research, taking as a reference the works of Olivesky (Olivesky et al, 2008) for stratified boiling inside horizontal tubes. It was possible to simulate a gaseous phase inside a liquid domain as a product of thermal induced mass transfer. This, caused by a heat flux prescribed on a solid domain, where the effects of the presence and generation of a two phase flow influenced the temperature of the pipe where the flow developed. By changing the tube’s materials it was possible to observe a decrease on the thermal gradients on the absorber.

Under the comparative approach, the presented results have proven to be useful, accounting for the difference on the thermal response of the different bimetallic structures designed for simulation.
5. CONCLUSIONS

The results of the numerical study referred to the performance of absorber tubes of solar parabolic trough collectors have been presented. Thanks to the use of computational fluid dynamics tools based on the finite volume method it was possible to model, simulate and obtain information on the characteristics of the temperature fields of the different bimetallic tubes designed and the fluid flow among other variables describing the dynamics of the flow.

The main objective of this research was to study the temperature gradients on the absorber tube caused by the interaction between the external heat flux (caused by concentrated solar radiation) and the flow inside the absorber and the capability of novel designs of the absorber to improve the process of energy conversion. In particular, steep temperature gradients must be avoided on the tube because they can cause deflections of the structure, stopping the functioning of the collector. It is proposed that the mixture of materials of high thermal conductivity and mechanical resistance can solve this issue.

Three representative segments of the events occurring on the solar field were chosen for simulation: a sub cooled liquid section (present on the first part of the collectors line) a super heated steam section (at the end of the field) and a intermediate section where both phases are present inside the absorber due to phase change caused by the heat flux on the tube’s external surface. The effort of the research was focused on this last segment, because the different properties of the phases promote high temperature difference on the cross section of the tube. Because of gravity and density difference between phases, they tend to separate with the steam travelling toward the highest part of the absorber.

The use of CFD on the processes described above has a well defined structure which can be summarized as: domain definition and discretization; definition relevant physics to be accounted for and the definition of the domain’s boundary conditions. These three stages are interrelated; therefore, it is important to consider that numerical simulation should be taken into account as a solution which deepens the understanding of a problem. Because
of this, it is necessary to have a solid base of knowledge about the key variables and physics that describe the problem. This leads, as with any engineering problem to simplifications and assumptions that apply in each particular case.

Computer simulations depend strongly on the assumptions made for a given problem, conditioning the processing cost which translates on the time spent on obtaining a solution.

This issue is a very important trade off in every CFD problem: how to define a problem in such a way that its solution delivers the most precise answer in the shortest time possible. Achieving the balance between processing time and the solution’s quality is based on the previous knowledge of the physics that affects the three stages of the process above commented.

For the first and last segments that were simulated (meaning, single phase problems), the use of CFD is straightforward as it is a problem of convective heat transfer between fluid flow and a solid domain with a constant heat flux on the exterior. Thus, the physics were defined based on the mass, momentum and energy conservation equations. In addition, turbulent effects were modeled through a vortex dissipation model, adding transport equations for two more variables.

The definition of the two-phase case resulted much more complex. In order to obtain a solution, transport equations for both fluids must be solved. Along with conservation equations for each phase, the closure relations that account for interaction between them, represented as transport of mass, energy and momentum must also be modeled. As of today, a generalized CFD model representing the complete physics of thermal phase change (as occurs inside the absorber) is not available. Thermal phase change process happens inside the tube along hundreds of meters, starting with sub cooled liquid and ending with a certain vapor mass fraction. But, the mechanisms that promote and sustain the process have length scales of millimeters. This limits the size of the domain to be modeled, as the discretization must be relative to the scale of the physics but must
respect the computational resource limits as a large domain with a small discretization generates a problem that could not be solved because of size considerations.

Regarding the phase change process, it was said before that there isn’t a comprehensive and general mechanistic model of boiling to simulate all the patterns generated between phases and the transitions between them as the amount of steam increases. Moreover: although the simulation of a ‘bubbly flow’ morphology is feasible to be modeled, there is not a general implementation of nucleated boiling near the walls, as in fact happens inside the absorber tube. Some mechanistic models for boiling flow inside vertical pipes are available, though their application to horizontal orientations was discarded in early stages of the research due to their inability for taking into account the effects of gravity.

With all, it was decided to take a rather indirect approach on the thermal phase change phenomenon, generating a new methodology that focus on a flow pattern that gathered the most relevant features to be studied when applied to the case of the absorber tubes.

When modeling thermal phase change through a mechanism where phases are generated or destroyed related to the saturation temperature at reference pressure on the fluid domain, the conditions were chosen for the formation of the stratified pattern. In this way, it was possible to examine temperature differences caused by the presence of two fluids inside the absorber, while the interphase between them is born and grows along the three-dimensional domain.

It was concluded then, that studies of this kind must be focused on particular segments. Doing so limits the boundary conditions that can be applied to the domain, given that they must assure the formation of the stratified pattern. In addition, it was shown that the studied case (flow in horizontal orientation) results especially complex to model due to the difficulties for the correct modeling of the gas phase as it accumulates on the top of the pipe (because of gravity).
The achieved advances on the detailed simulation of this kind of flows (through future works and research) should present a natural progression according to the evolution of the flow patterns that are formed inside the heated pipe causing thermal phase change. First it is necessary a general model able to solve correctly the mechanism of wall nucleation. Afterwards, the implementation of equations to model breakage and coalescence of larger steam bubbles inside the pipe would allow the transition to flow patterns with a higher steam quality.

Once defined, the methodology devised to study the cases above mentioned (single phase: sub cooled liquid and superheated steam, and two phases: stratified flow) was applied to a base case consisting in flow inside a steel absorber and three bimetallic structures. The bimetallic absorbers were designed based on two concentric tubes made of materials of different properties. The design included three cases of an internal copper pipe with the external pipe made of steel. The wall thickness is maintained in all cases, varying the share of each material. The shares simulated were: 70% (steel) - 30% (copper), 50%-50% and 30%-70%.

It is worth mentioning that carrying out a comparative study for all the absorbers designed appears to be the most adequate choice, given the strong assumptions applied when defining the methodology for the assessment of two-phase flow (due to the aforementioned limitations).

When analyzing the obtained results it was possible to observe that including a material of high thermal conductivity reduced the temperature difference measured on the absorber’s cross section. Specifically, in the single phase cases (for both fluids simulated) it was concluded that comparatively, for the pipes with a higher share of copper in the wall, the reduction of the temperature is also bigger, though the relative effectiveness falls. The stratified flow cases showed, as expected, the highest temperature difference due to the capability of each material to remove heat from the wall. The results showed that when comparing to the base case, all cases including copper shows a great reduction of the maximum difference of temperature at the cross
section. Comparatively, for single phase and stratified flows, higher additions of copper have little effectiveness respect the case with less copper (70%-30%), showing very similar thermal behavior.

The results obtained, comprising both the methodology developed for the simulation of stratified flow interacting with a solid domain and its application to the design of absorber tubes can be very useful in the design of heat exchangers where the working fluid undergoes phase change (as in traditional boiler design), a design process marked for the experience of the manufacturer rather than thorough calculations and modeling of the physical processes involved, basically due to their complexity.

Summarizing, the presented research’s objectives show different degrees of compliance. Based on the main contributions of this work, they are: the development of the methodology for the assessment of absorber tubes where the flow undergoes phase change and the application to the bimetallic designs. Regarding the first topic, compliance is not comprehensive, as it was explained that the understanding of the phenomenon of phase change and the level of development of models to represent it is neither complete nor optimal. Existing methods are not applicable to all cases and can present large margins of error (as occurs with some correlations). The methodology here presented share some of those flaws because it was designed based on existing models and is applicable (so far) only for cases of stratified flow. Nevertheless, it represents an advance respect the current state of the research of CFD applied to thermal phase change. It represents a step forward to the development of more robust models.

Regarding the results of the simulation of bimetallic tubes, the research was successful to establish a relation between bimetallic configurations and the reduction of temperature gradients. CFD tools were effective in showing the usefulness of such designs in zones with one or two phase flow.

Finally, as future work recommendations, some can be mentioned:
a) Improvement of the presented methodology for phase change modeling: it is of interest the implementation of a local Nusselt number for the phase change model, rather than the averaged value used at this stage.

b) Study of discretization-related sensitivity: at this stage of the research line, the meshes generated responded to very general criteria regarding element side, aspect ratio and boundary layer refinement. Following a conservative criterion, it is possible that the number of elements generated is overestimated. In any case, future advances and improvements of the models used on this methodology will make it more expensive in terms of computational resources. Because of this it will be necessary to assess the adequate level of discretisation of the domains where the models apply.

c) Robust implementation of a wall boiling model: since current models fail to model horizontal direction, a possible line of work could be the extension of the model to such positions.

d) Integrate the results of CFD simulations under current and upcoming methodologies to FEA analysis tools, in order to account for the effects of temperature gradients on the structure through stress and deformation analysis.
REFERENCES


Comisión de Minería y Energía, Cámara del Senado. (2006). Informe de la comisión de minería y energía sobre fomento de las fuentes de energía renovables y no contaminantes.


E. Krepper, Y. Egorov, CFD-modelling of subcooled boiling and application to simulate a hot channel of a fuel assembly, Proc. 13th Int. Conf. Nuclear Engineering (ICONE-13), Beijing, China, May 16-20, 2005


APPENDIXES
APPENDIX A: STRATIFIED FLOW DETAILED IMAGES

Figure A-1: Liquid temperature contour plot (0.4 m from the inlet).

Figure A-2: Liquid temperature contour plot (0.8 m from the inlet)
Figure A-3: Liquid temperature contour plot (1,2 m from the inlet)

Figure A-4: Liquid temperature contour plot (1,6 m from the inlet)
Figure A-5: Liquid temperature contour plot (2.0 m from the inlet)

Figure A-6: Steam temperature contour plot (0.4 m from the inlet)
Figure A-7: Steam temperature contour plot (0.8 m from the inlet)

Figure A-8: Steam temperature contour plot (1.2 m from the inlet)
Figure A-9: Steam temperature contour plot (1.6 m from the inlet)

Figure A-10: Steam temperature contour plot (2.0 m from the inlet)
Figure A-11: Solid domains temperature contour plot (0.4 m from the inlet).

Figure A-12: Solid domains temperature contour plot (0.8 m from the inlet)
Figure A-13: Solid domains temperature contour plot (1,2 m from the inlet)

Figure A-14: Solid domains temperature contour plot (1,6 m from the inlet)
Figure A-15: Solid domains temperature contour plot (2.0 m from the inlet).
APPENDIX B: STARTING POINT CONCEPTUALIZATION

In order to illustrate the capabilities of ANSYS CFX-11 for predicting the thermal response of the proposed configurations, the results of analytic calculations applied to a simplified case is confronted with the results of a simulated domain.

The problem is one of a single phase steady state flow inside a pipe with a constant heat flux on the external surface. The pipe comprises a bimetallic wall, with the following dimensions:

- Length: 5 m
- Internal diameter: 0.03 m
- External diameter: 0.05 m
- Copper wall thickness: 0.01 m

Figure B-1: projected view of the domain for the heat transfer problem
Water flows through the pipe at 10 litres per minute, entering the domain at 15 °C. By means of an electric heater a heat flux of 34.444 W is supplied. It is assumed that the outside of the pipe is perfectly insulated.

Given the boundary conditions, the bulk temperature of the fluid, the temperature of the internal wall and the external temperature of the pipe are to be obtained through both solving methods mentioned above.

Analytic Calculation:

Given a completely developed flow at a steady state, the energy balance in a control volume comprising only the fluid inside the pipe yields (for an infinitesimal control volume of width $dx$, as depicted on Figure B-2):

$$\dot{m} c_p T_b + q'' dx \rho P = \dot{m} c_p T_b + \dot{m} c_p dT \quad (B.1)$$

thus:

$$\frac{dT}{dx} = \frac{q'' \rho P}{\dot{m} c_p} \quad (B.2)$$

$$T_b(x) = \frac{q'' \rho}{\dot{m} c_p} \cdot x + T_i \quad (B.3)$$
Where:

\[ q'' = \frac{Q}{A_{\text{int}}} \]  \hspace{1cm} (B.4)

\[ P = \pi \cdot D \]  \hspace{1cm} (B.5)

\[ A_{\text{int}} = \pi \cdot D \cdot L \]  \hspace{1cm} (B.6)

\[ \dot{m} = \rho \cdot \dot{V} \]  \hspace{1cm} (B.7)

with:

\[ D = 0.03 \, \text{m} \]

\[ L = 5 \, \text{m} \]

\[ T_l = 15 \, ^\circ \text{C} \]

\[ Q = 34.444 \, \text{W} \]

\[ q'' = 73.092 \, \frac{\text{W}}{\text{m}^2} \]

\[ \rho = 997 \, \frac{\text{kg}}{\text{m}^3} \]

\[ \dot{m} = 997 \, \frac{\text{kg}}{\text{m}^3} \cdot 0.01 \, \frac{\text{m}^3}{\text{min}} \cdot \frac{1}{60 \, \text{s}} = 0.165 \, \frac{\text{kg}}{\text{s}} \]

Equation D.3 yields:

\[ T_b(x) = 10 \cdot x + 15 \]  \hspace{1cm} (B.8)

For wall temperature, Reynolds number must be known first hand:

\[ Re = \frac{V \cdot D}{u} \]  \hspace{1cm} (B.9)

where

\[ V = \frac{0.01 \, \frac{\text{m}^3}{\text{min}} \cdot \frac{1}{60 \, \text{s}}}{\pi \cdot 0.03^2/4} = 0.236 \, \frac{\text{m}}{\text{s}} \]
\[ v = 0,633 \cdot 10^{-6} \, \text{m}^2 \, \text{s}^{-1} \]

Yields:

\[ Re = 10.760 \]

Therefore, the flow is turbulent and a convective heat transfer coefficient must be modeled after a suitable correlation. The convective heat transfer coefficient \( h \) is determined as:

\[ h = \frac{k}{D} Nu \quad (B.10) \]

Where Nusselt number is determined after the Dittus-Boelter correlation for turbulent heat transfer (Thome, 2007):

\[ Nu = 0,023 \cdot Re^{0,8} Pr^{0,4} \quad (B.11) \]

Where

\[ Pr = 4,33 \]
\[ k = 0,633 \, \frac{W}{m \, K} \]

Thus,

\[ h = 1462 \, \frac{W}{m^2 \, K} \]

Now, for a completely developed flow, the following relation is valid:

\[ h = \frac{q'''}{(T_w - T_b)} \quad (B.12) \]

And the value for \( h \) remains constant.

Then,

\[ T_w(x) = 50,33 + T_b \quad (B.13) \]
\[ T_w(x) = 10 \, x + 65,33 \quad (B.14) \]
Finally, in order to calculate the external temperature of the pipe, the resistances’ method is employed, as showed in Figure B-3:

![Figure B-3: Thermal resistances scheme for the bimetallic structure](image)

For copper:

\[
R_c = \frac{\ln \left( \frac{0.02}{0.015} \right)}{(2 \cdot \pi \cdot k_c \cdot L)}
\]  

(B.15)

With \( k_c \), the thermal conductivity of copper:

\[ k_c = 401 \frac{W}{m \cdot K} \]

For steel:

\[
R_s = \frac{\ln \left( \frac{0.025}{0.02} \right)}{(2 \cdot \pi \cdot k_s \cdot L)}
\]  

(B.16)

With \( k_s \), the thermal conductivity of steel:
\[ k_s = 60.5 \, \frac{W}{m \cdot K} \]

Finally,

\[ R_T = R_s + R_C \quad \text{(B.17)} \]

\[ R_T = 1.4 \cdot 10^{-4} \]

\[ Q = \frac{T_e - T_w}{R_T} \quad \text{(B.18)} \]

Thus:

\[ T_w(x) = 10 \, x + 70.1 \quad \text{(B.19)} \]

Now, let’s compare this results with those obtained through the CFD computations

**Simulation setup and results**

A single phase, steady state simulation is performed on ANSYS CFX-11 using a model of the exact same dimensions as in the analytic calculation.

A three dimensional domain is created based on those dimensions, generating a rather coarse mesh, sufficient for the purposes of verifying the accuracy of the simulations when compared to analytic calculations.

Figures B-4 show the aspect of the generated mesh, composed of 109,809 hexahedric elements, comprising 133,000 nodes for the three domains where the problem’s physics are solved. As usual in numerical simulations, a mesh sensibility study was carried out, in order to obtain results independent from the mesh’s dimensions.
The design of the mesh for this simple case was based on the guidelines given on the CFD literature (ANSYS Inc., 2006). Several other meshes were used to solve this problem (varying the number of elements in the axial direction), showing that for the range of nodes created, the results obtained did not vary in a significantly manner. Table B-1 shows the sensitivity of the average temperature on the outlet respect to the mesh size:

Table B-1: Results sensibility to mesh size.

<table>
<thead>
<tr>
<th>Number of Nodes</th>
<th>Temperature average at domain’s outlet [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>67.600</td>
<td>338,202</td>
</tr>
<tr>
<td>101.400</td>
<td>338,276</td>
</tr>
<tr>
<td>133.000</td>
<td>338,242</td>
</tr>
<tr>
<td>142.400</td>
<td>338,037</td>
</tr>
</tbody>
</table>
It stands clear that the dimensions of the mesh used in these calculations yields results that not vary when changing the discretization level.

The boundary conditions defined for the problem are exactly the same as in the analytic procedure: prescribed heat flux on the external wall, a defined inlet mass flow with static temperature. Besides that, the averaged static pressure is defined as zero on the fluid’s outlet region, and both ends of the pipe have a zero gradient conditions for all variables. In addition to that, the inlet boundary condition that was assumed with a developed velocity profile, is applied here as well: previously, the isothermal case is solved with the mass flow already defined, obtaining at the outlet a developed velocity profile, used as at the inlet in the case including heat transfer.

In regards of the solver setup, the simulation was defined as steady state and a second order numerical scheme was used to treat the advection terms. The stopping criterion established termination when the maximum value for the residuals is under 0.0001.

To reproduce the results obtained on the analytic calculations, a sampling based on three lines, located on the center of the pipe, internal and external surfaces (in axial orientation) is plotted.

The plot showed on Figure B-5 shows the temperature obtained in the lines previously defined. They define the bulk temperature (line on the pipe’s axis) as the internal wall temperature and external wall temperature.
Figure B-5: Simulation results for the same temperatures calculated in analytic procedure

When comparing the plot on Figure B-5 to the functions obtained on the first calculations (equations B.8, B.14 and B.19), the difference is practically negligible:

When calculating the temperatures through the analytic method, it yields:

\[ T_b(x = 5) = 65 \, ^\circ\text{C} \]
\[ T_w(x = 5) = 115,3 \, ^\circ\text{C} \]
\[ T_{ext}(x = 5) = 120,1 \, ^\circ\text{C} \]

Even the linear approximation of the curves yields functions very similar to those obtained ‘by hand’, even when the initial segment is included, as depicted on Figure B-6:
By means of the modeling of this relatively simple heat transfer problem through an analytic procedure and CFD tools it is possible to verify the capabilities of the computational code used in this research to represent accurately the actual heat transfer phenomenon.

Figure B-6: Linear approximation to the calculated temperatures on the simulated results