

MINISTÉRIO DA EDUCAÇÃO
UNIVERSIDADE FEDERAL DO RIO GRANDE
PROGRAMA DE PÓS-GRADUAÇÃO EM MODELAGEM COMPUTACIONAL

CONSTRUCTAL DESIGN APPLIED TO A CHANNEL WITH TWO ALTERNATED HEATED
BLOCKS SUBJECTED TO FORCED CONVECTIVE TURBULENT FLOWS WITH BOILING

by

Bruno Costa Feijó

Thesis for obtaining the title of
Doctor in Computational Modeling

Rio Grande, April, 2023

Bruno Costa Feijó

**“CONSTRUCTAL DESIGN APPLIED TO A CHANNEL WITH TWO ALTERNATED
HEATED BLOCKS SUBJECTED TO FORCED CONVECTIVE TURBULENT FLOWS
WITH BOILING”**

Tese apresentada ao Programa de Pós-Graduação
em Modelagem Computacional da Universidade
Federal do Rio Grande - FURG, como requisito
parcial para obtenção do Grau de Doutor. Área
de concentração: Modelagem Computacional.

Aprovado em:

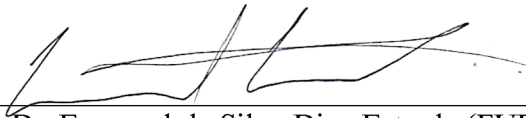
BANCA EXAMINADORA



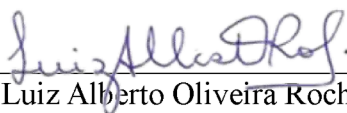
Prof. Dr. Elizaldo Domingues dos Santos (Orientador – FURG)



Prof. PhD. Sylvie Lorente (Coorientadora – Villanova University)



Prof. Dr. Emanuel da Silva Diaz Estrada (FURG)



Prof. Dr. Luiz Alberto Oliveira Rocha (FURG)



Prof. Dr. Cicero Coelho de Escobar (UFPel)

Ficha Catalográfica

F297c Feijó, Bruno Costa.

Constructal design applied to a channel with two alternated heated blocks subjected to forced convective turbulent flows with boiling / Bruno Costa Feijó. – 2023.

163 f.

Tese (doutorado) – Universidade Federal do Rio Grande – FURG, Programa de Pós-Graduação em Modelagem Computacional, Rio Grande/RS, 2023.

Orientador: Dr. Elizaldo Domingues dos Santos.

Coorientadora: Dra. Sylvie Lorente.

1. Transferência de calor 2. Dinâmica dos fluidos 3. Avaliação geométrica 4. Teoria construtal 5. Ebulição I. Santos, Elizaldo Domingues dos II. Lorente, Sylvie III. Título.

CDU 004:536.2

Catálogo na Fonte: Bibliotecário José Paulo dos Santos CRB 10/2344

This work is dedicated to my mother, father, sister, the pug Duke and all my family

ACKNOWLEDGEMENTS

First, I would like to thank my parents for providing all the conditions during my life for me to be able to complete a doctoral thesis. I would also like to thank my entire family for their support during this time. I also thank the pug Duke for his companionship during the preparation of this thesis.

I would like to thank Professor Doctor Elizaldo Domingues dos Santos for all these years of great guidance and friendship. I also thank Professor Doctor Sylvie Lorente for contributing with her enormous talent and spirit to the elaboration of this thesis. It was an immensely good experience to work with both of them.

I would like to thank the Graduate Program in Computational Modeling at FURG and also the Federal University of Rio Grande (FURG) for making this doctoral thesis possible.

ABSTRACT

The present thesis is a study on the geometric evaluation of channels with rectangular blocks subjected to a boiling flow. The proposed problem was numerically simulated using computational fluid dynamics tools and consisted of two-dimensional channels with two mounted rectangular blocks. One block is placed on the upper wall and another on the lower one, where they and their adjacent region have a temperature higher than the saturation temperature of the fluid, initially water in its liquid state, which undergoes the boiling process when passing through this heated region. Cases in the turbulent flow regime were analyzed. The geometric evaluation was performed using the Constructal Design method, associated with the Exhaustive Search method and the TOPSIS multi-criteria method. Within the scope of Constructal Design, two degrees of freedom were defined: the ratio between the height and width of each block, H_1/L_1 and H_2/L_2 . The effect of the ratio between the distance between the center of the blocks and the length of occupation, L_3/L_{ocu} , was also investigated. Three constraints were also defined: the channel area (A), the occupancy area of each block (A_1 , the same for both blocks) and the area of the blocks (A_2 , the same for both blocks). The geometric evaluation process has two objectives: the maximization of the heat exchange between the blocks and the flow and the minimization of the pressure difference between the inlet and outlet of the channel, i.e., a thermal and a fluid dynamic objective. All cases evaluated were also simulated with the boiling process disabled, to assess the influence of boiling on the geometric evaluation process. The multi-objective analysis was performed based on the application of the TOPSIS multi-criteria method, where weights were considered for the thermal (w_T) and fluid dynamic (w_F) objectives. Regarding the cases without boiling, the effect of the geometry of the two blocks over the relative proximity of the ideal solution, C_b , was similar to that observed in the cases with boiling. However, a considerable difference was noticed in the optimal geometry of the second block when $w_T = 0.7$ and $w_F = 0.3$. It was possible to observe that the cases with boiling presented not only a better multi-objective performance for all combinations of weights but also had a higher heat transfer rate and a lower pressure drop along the channel. An analysis of the geometric evaluation considering the Coefficient of Performance (COP) as an objective was also carried out.

Keywords: heat transfer; fluid dynamic; geometric evaluation; constructal theory; boiling.

RESUMO

Título em português: Design Construtal aplicado a um canal com dois blocos alternados aquecidos sujeito à convecção forçada no regime turbulento com ebulição

A presente tese é um estudo sobre a avaliação geométrica em canais com blocos retangulares sujeitos a um escoamento com ebulição. O problema proposto foi simulado numericamente através de ferramentas da dinâmica dos fluidos computacional e consiste em canais bidimensionais com dois blocos retangulares, um na parede superior e outro na inferior, onde os mesmos e a sua região adjacente se encontram em uma temperatura superior a temperatura de saturação do fluido, inicialmente a água em seu estado líquido, que sofre o processo de ebulição ao passar por esta região aquecida. Foram analisados casos no regime de escoamento turbulento. A avaliação geométrica foi realizada com a utilização do método Design Construtal, associado ao método de Busca Exaustiva e do método multi-critério TOPSIS. No âmbito do Design Construtal, foram definidos dois graus de liberdade: a razão entre a altura e a largura de cada bloco, H_1/L_1 e H_2/L_2 . O efeito da razão entre a distância entre o centro dos blocos e o comprimento de ocupação, L_3/L_{ocu} , também foi investigado. Também foram definidas três restrições: a área do canal (A), a área de ocupação de cada bloco (A_1 , igual para os dois blocos) e a área dos blocos (A_2 , igual para os dois blocos). O processo de avaliação geométrica teve dois objetivos: maximizar a troca de calor entre os blocos e o escoamento e minimizar a diferença de pressões entre a entrada e a saída dos canais, ou seja, um objetivo térmico e um fluidodinâmico. Todos os casos avaliados também foram simulados com o processo de ebulição desabilitado, para avaliar a influência da ebulição no processo de avaliação geométrica. A análise multi-objetivo foi realizada a partir da aplicação do método multi-critério TOPSIS, onde foram considerados pesos para os objetivos térmico (w_T) e fluidodinâmico (w_F). Em relação aos casos sem ebulição, o efeito da geometria dos dois blocos sobre a proximidade relativa da solução ideal, C_b , foi parecido ao observado nos casos com ebulição. No entanto, uma grande diferença foi notada na geometria ótima do segundo bloco quando $w_T = 0.7$ e $w_F = 0.3$. Foi possível observar que os casos com ebulição apresentaram não somente um melhor desempenho multi-objetivo para todas as combinações de pesos, mas também tiveram uma maior taxa de transferência de calor e uma menor queda de pressão ao longo do canal. Uma análise da avaliação geométrica considerando o Coeficiente de Desempenho (COP) como objetivo também foi realizada.

Palavras-chaves: transferência de calor; dinâmica dos fluidos; avaliação geométrica; teoria construtal; ebulição.

INDEX

1. INTRODUCTION	22
1.1. Motivation	22
1.2. Bibliographic Review	24
1.2.1. Boiling Flows in Conventional Channels	24
1.2.2. Boiling Flows in Minichannels and Microchannels	26
1.2.3. Numerical Modeling of Internal Boiling Flows	30
1.2.4. Geometric Evaluation of Channels with Boiling Flow	35
1.2.5. Constructal Design Applied to Heat Transfer and Fluid Dynamics	39
1.2.6. Constructal Design Applied to Channels	43
1.3. Objectives	45
1.3.1. Specific Objectives	45
2. THEORETICAL FOUNDATION	47
2.1. Fundamentals of Boiling	47
2.1.1. Pool Boiling	48
2.1.2. Forced Convective Boiling	50
2.1.3. Forced Convective Boiling in Internal Flows	51
2.2. Coefficient of Performance	59
2.3. Constructal Theory	60
2.4. TOPSIS	64
3. MATHEMATICAL MODELING	67
3.1. Problem Description	67
3.2. Constructal Design	69
3.3. Geometrical Evaluation	70
3.4. The Eulerian Multiphase Model	73
3.5. The RPI Boiling Model	79
3.6. Turbulence Modeling – The κ - ω SST model	82
4. NUMERICAL MODELING	84
4.1. Numerical Procedures	84
4.1.1. Finite Volume Method	85
4.1.2. Numerical Parameters	89
4.2. Mesh Independence Study	89

4.2.1. Mesh Independence Test - Cases without Boiling	90
4.2.2. Mesh Independence Test – Cases with Boiling	91
5. RESULTADOS	93
5.1. Validação do Modelo Numérico Aplicado	93
5.2. Avaliação Geométrica de um Canal com Dois Blocos Alternados Aquecidos Sujeito a Escoamentos Turbulentos Convectivos com e sem Ebulição	95
5.2.1. Análise Térmica	96
5.2.2. Análise Fluidodinâmica	102
5.2.3. Análise Multi-Objetivo	107
5.2.4. Análise do Coeficiente de Desempenho (<i>COP</i>)	121
6. CONCLUSÕES	126
6.1. Proposta de Continuidade	129
7. REFERENCES	131
APPENDIX 1 - Work published in Reports in Mechanical Engineering, Vol. 3, nº 1, 2022	146

LIST OF FIGURES

Figure 1.1 – Typical consecutive images of bubble growth, departure and lift-off (Guan et al., 2015).	26
Figure 1.2 – Influence of vapor quality (x) over the heat transfer coefficient, obtained in the work by Huo et al. (2004).	28
Figure 1.3 – One of the simulations performed in the work by Ling et al. (2015).	34
Figure 1.4 – Geometries proposed by Gao and Bhavnani (2019).	38
Figure 1.5 – Temperature fields of some of the geometries studied by Lorenzini et al. (2015).	41
Figure 1.6 – Temporal mean of velocity fields for some of the cases studied by Teixeira et al. (2018).	42
Figure 1.7 – Temperature (a), pressure (b) and velocity (c) fields of the optimal geometry found in the work by Feijó et al. (2018).	44
Figure 2.1- Stages of pool boiling (Kandlikar et al., 1999).	49
Figure 2.2 - Patterns of flow and heat transfer in vertically and upward convective boiling flows (Collier and Thome, 1994).	53
Figure 2.3 – Vapor volume fraction in subcooled boiling (Hewitt, 1978).	55
Figure 2.4 – Satellite view of the city of Rome, Italy (Google Earth, October 2021).	63
Figure 2.5 – Example of a multi-criteria problem (Hwang and Yoon, 1981).	65
Figure 3.1 – Computational domain of the channel/microchannel with two heated rectangular blocks.	67
Figure 3.2 – Schematic representation of the methodology applied to the geometric evaluation and optimization of the present study.	71
Figure 3.3 – Geometrical evaluation process applied to the channel.	72
Figure 3.4 – Comparison of phase volume fraction fields between the VOF and Eulerian models for the same multiphase flow. Adapted from Guerrero et al. (2017).	74
Figure 4.1 – Examples of meshes of the finite volume method (Zikanov, 2010).	86
Figure 4.2 - Mesh with triangular and rectangular finite volumes employed in the cases without boiling.	91
Figure 4.3 - Mesh with triangular and rectangular finite volumes employed in the cases with boiling.	92
Figure 5.1 – Domínio computacional utilizado para validar o modelo numérico aplicado no presente estudo.	93

Figure 5.2 – Comparação da fração de volume de vapor ao longo do canal obtida no presente estudo em relação a outros trabalhos.	94
Figure 5.3 – Comparação da temperatura média do líquido ao longo do canal.	94
Figure 5.4 – Comparação da temperatura da parede ao longo do canal.	95
Figura 5.5 – Efeito de H_1/L_1 sobre a taxa de transferência de calor (q) para diferentes valores de H_2/L_2 .	97
Figura 5.6 – Efeito de H_2/L_2 sobre a taxa de transferência de calor uma vez maximizada (q) _{1,max} .	97
Figure 5.7 – Efeito de H_2/L_2 sobre a razão H_1/L_1 uma vez termicamente otimizada, $(H_1/L_1)_{o,T}$.	98
Figura 5.8 – Efeito de L_3/L_{ocu} sobre a taxa de transferência de calor duas vezes maximizada (q) _{2,max} .	99
Figura 5.9 – Efeito de L_3/L_{ocu} sobre a razão H_1/L_1 duas vezes otimizada termicamente, $(H_1/L_1)_{oo,T}$.	100
Figura 5.10 – Efeito de L_3/L_{ocu} sobre a razão H_2/L_2 uma vez otimizada termicamente, $(H_2/L_2)_{o,T}$.	100
Figura 5.11 – Campos de temperatura (K) (a), fração de volume (b), pressão (Pa) (c), velocidade (m/s) (d) e linhas de fluxo (e) para as geometrias ótimas térmicas com ebulição, $(H_1/L_1)_{ooo,T} = 1.55$, $(H_2/L_2)_{oo,T} = 1.55$ e $(L_3/L_{ocu})_{o,T} = 0.375$.	101
Figura 5.12 – Campos de temperatura (K) (a), pressão (Pa) (b), velocidade (m/s) (c) e linhas de fluxo (d) para as geometrias ótimas térmicas sem ebulição, $(H_1/L_1)_{ooo,T} = 1.55$, $(H_2/L_2)_{oo,T} = 1.55$ e $(L_3/L_{ocu})_{o,T} = 0.375$.	101
Figura 5.13 – Efeito de H_1/L_1 sobre a queda de pressão (ΔP) para diferentes valores de H_2/L_2 .	102
Figura 5.14 – Efeito de H_2/L_2 sobre a queda de pressão uma vez minimizada (ΔP) _{1,min} .	103
Figura 5.15 – Efeito de H_2/L_2 sobre a razão H_1/L_1 uma vez otimizada fluidodinamicamente, $(H_1/L_1)_{o,F}$.	103
Figura 5.16 – Efeito de L_3/L_{ocu} sobre a queda de pressão duas vezes minimizada (ΔP) _{2,min} .	104
Figura 5.17 – Efeito de L_3/L_{ocu} sobre a razão H_1/L_1 duas vezes otimizada fluidodinamicamente, $(H_1/L_1)_{oo,F}$.	105
Figura 5.18 – Efeito de L_3/L_{ocu} sobre a razão H_2/L_2 uma vez otimizada fluidodinamicamente, $(H_2/L_2)_{o,F}$.	105
Figura 5.19 - Campos de temperatura (K) (a), fração de volume (b), pressão (Pa) (c), velocidade (m/s) (d) e linhas de fluxo (e) para as geometrias ótimas fluidodinâmicas com ebulição, $(H_1/L_1)_{ooo,F} = 0.0625$, $(H_2/L_2)_{oo,F} = 0.0625$ e $(L_3/L_{ocu})_{o,F} = 0.625$.	106
Figura 5.20 - Campos de temperatura (K) (a), pressão (Pa) (b), velocidade (m/s) (c) e linhas de	

fluxo (d) para as geometrias ótimas fluidodinâmicas sem ebulição, $(H_1/L_1)_{000,F} = 0.0625$, $(H_2/L_2)_{00,F} = 0.0625$ e $(L_3/L_{ocu})_{0,F} = 0.625$. 107

Figura 5.21 – Efeito de H_1/L_1 sobre a proximidade relativa da solução ideal (C_b) para diferentes H_2/L_2 , $L_3/L_{ocu} = 0.5$ e para: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$. 109

Figura 5.22 – Efeito de H_2/L_2 sobre a proximidade relativa da solução ideal uma vez maximizada (C_b)_{1,max}, para $L_3/L_{ocu} = 0.5$ e para: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$. 110

Figura 5.23 – Efeito de H_2/L_2 sobre a razão H_1/L_1 uma vez otimizada, $(H_1/L_1)_0$, para $L_3/L_{ocu} = 0.5$ e para: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$. 111

Figura 5.24 – Efeito de L_3/L_{ocu} sobre a proximidade relativa da solução ideal duas vezes maximizada (C_b)_{2,max}, $L_3/L_{ocu} = 0.5$ e para: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$. 113

Figura 5.25 – Efeito de L_3/L_{ocu} sobre a razão H_1/L_1 duas vezes otimizada, $(H_1/L_1)_{00}$, $L_3/L_{ocu} = 0.5$ e para: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$. 114

Figura 5.26 – Efeito de L_3/L_{ocu} sobre a razão H_2/L_2 duas vezes otimizada, $(H_2/L_2)_0$, L_3/L_{ocu} e: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$. 115

Figura 5.27 – Campos de temperatura (K) para as geometrias ótimas multi-objetivo com ebulição para diferentes combinações de pesos dos objetivos. 117

Figura 5.28 – Campos de fração de volume para as geometrias ótimas multi-objetivo com ebulição para diferentes combinações de pesos dos objetivos. 117

Figura 5.29 – Campos de pressão (Pa) para as geometrias ótimas multi-objetivo com ebulição para diferentes combinações de pesos dos objetivos. 118

Figura 5.30 – Campos de velocidade (m/s) para as geometrias ótimas multi-objetivo com ebulição para diferentes combinações de pesos dos objetivos. 118

Figura 5.31 – Campos de linhas de fluxo para as geometrias ótimas multi-objetivo com ebulição para diferentes combinações de pesos dos objetivos. 119

Figura 5.32 – Campos de temperatura (K) para as geometrias ótimas multi-objetivo sem ebulição para diferentes combinações de pesos dos objetivos. 119

Figura 5.33 – Campos de pressão (Pa) para as geometrias ótimas multi-objetivo sem ebulição para

diferentes combinações de pesos dos objetivos.	120
Figura 5.34 – Campos de velocidade (m/s) para as geometrias ótimas multi-objetivo sem ebulição para diferentes combinações de pesos dos objetivos.	120
Figura 5.35 – Campos de linhas de fluxo para as geometrias ótimas multi-objetivo sem ebulição para diferentes combinações de pesos dos objetivos.	121
Figura 5.36 – Efeito de H_1/L_1 sobre o coeficiente de desempenho (COP) para diferentes valores de H_2/L_2 .	122
Figura 5.37 – Efeito de H_2/L_2 sobre o coeficiente de desempenho uma vez maximizado (COP) _{1,max} .	122
Figure 5.38 – Efeito de H_2/L_2 sobre a razão H_1/L_1 uma vez otimizada, $(H_1/L_1)_o$.	123
Figura 5.39 – Efeito de L_3/L_{ocu} sobre o coeficiente de desempenho duas vezes maximizado (COP) _{2,max} .	123
Figura 5.40 – Campos de temperatura (K) (a), fração de volume (b), pressão (Pa) (c), velocidade (m/s) (d) e linhas de fluxo (e) para as geometrias ótimas com ebulição, $(H_1/L_1)_{ooo} = 0.9$, $(H_2/L_2)_{oo} = 0.0625$ e $(L_3/L_{ocu})_o = 0.625$.	124
Figura 5.41 – Campos de temperatura (K) (a), pressão (Pa) (b), velocidade (m/s) (c) e linhas de fluxo (d) para as geometrias ótimas sem ebulição, $(H_1/L_1)_{ooo} = 1.3$, $(H_2/L_2)_{oo} = 0.0625$ e $(L_3/L_{ocu})_o = 0.625$.	125

LIST OF TABLES

Table 2.1. Determination of the parameters F_1 , F_2 , F_3 and F_4 for the calculation of the critical heat flux	59
Table 3.1 – Dimensions of the geometry of the channel	69
Table 3.2 – Conditions applied to the studied cases	69
Table 3.3 - Thermofluidic properties of the phases at the saturation temperature	70
Table 3.4 – Comparison between the Volume of Fluid and Eulerian models	74
Table 4.1 – Numerical parameters applied in the present study.	89
Table 4.2 - Mesh independence test - cases without boiling	91
Table 4.3 - Mesh independence test - cases with boiling	92
Tabela 5.1 – Geometrias ótimas multi-objetivo	116
Tabela 5.2 – Resultados obtidos das geometrias ótimas na análise multi-objetivo	116

LIST OF SYMBOLS

A	Area [m ²]
A^*	Positive ideal solution in the TOPSIS method [-]
A^-	Negative ideal solution in the TOPSIS method [-]
A_1	Occupation area of the first block [m ²]
A_2	Occupation area of the second block [m ²]
A_b	Area of influence [m ²]
A_i	Concentration of the interfacial area [m ² /m ³]
a	Ad hoc constant of the $\kappa - \omega$ SST model [-]
Alt_i	i -th alternative to be considered in the TOPSIS method [-]
Bo	Boiling number, $\frac{q''}{G i_{lv}}$ [-]
b	TOPSIS method weight combination set [-]
C	Nucleate site density empirical parameter [-]
C_D	Drag coefficient [-]
C_i	Relative proximity between each alternative and the ideal solution, $\frac{D_i^-}{D_i^- + D_i^*}$ [-]
C_l	Lift coefficient [-]
C_o	Constant relative to the volume fraction distribution in the flow [-]
c_p	Specific heat at constant pressure [kJ/(kg.K)]
$C_{s,f}$	Constant of interaction between liquid and surface finish of the solid [-]
C_{vm}	Virtual mass coefficient [-]
C_{wt}	Bubble waiting time coefficient [-]
CC	Convergence criterion [-]
$CD_{k\omega}$	Function 5 of constants and variables of the $\kappa - \omega$ SST model [-]
COP	Coefficient of performance [-]

D	Diameter [m]
D_w	Bubble departure diameter [m]
D_i^*	Distance from an alternative i to the positive ideal solution in the TOPSIS method, $\sqrt{\sum_{j=1}^m (v_{ij} - v_j^*)^2} \text{ [-]}$
D_i^-	Distance from an alternative i to the negative ideal solution in the TOPSIS method, $\sqrt{\sum_{j=1}^m (v_{ij} - v_j^-)^2} \text{ [-]}$
d	Vapor bubble diameter [m]
$\vec{F}$	Force vector [N]
F_1	Function 1 of constants and variables of the $\kappa - \omega$ SST model [-]
F_2	Function 2 of constants and variables of the $\kappa - \omega$ SST model [-]
$\vec{F}_{lift}$	Lift force vector [N]
$\vec{F}_{td}$	Turbulent dispersion force vector [N]
$\vec{F}_{vm}$	Virtual mass force vector [N]
f	Frequency of separation of the vapor bubbles [1/s]
f_{drag}	Default drag function, $\frac{C_D Re_{rel}}{24} \text{ [-]}$
G	Mass flux [kg/m ² .s]
g	Acceleration of gravity [m/s ²]
h	Convective heat transfer coefficient [W/m ² .K]
H_1	Height of the first block [m]
H_2	Height of the second block [m]
I	Identity tensor [-]
I_{sr}	Invariant measure of the strain rate [1/s]
i	Enthalpy [kJ/kg]
i_{lv}	Latent heat of the fluid [kJ/kg]

Ja_{sub}	Subcooled Jacob number, $\frac{\rho_l c_{p_l} \Delta T_{sub}}{\rho_v i_{lv}}$ [-]
K	Dimensionless constant of the channel's geometry from Zuber correlation of the Critical Heat Flux, [-]
K_i	RPI Boiling empirical coefficient [-]
k	Thermal conductivity of the liquid [W/(m.K)]
k_t	Turbulent kinetic energy [m ² /s ²]
L	Length [m]
L_1	Length of the first block [m]
L_2	Length of the second block [m]
L_3	Distance of center-to-center of the blocks [m]
L_{ocu}	Length of occupation of the blocks [m]
$\dot{m}$	Mass flow rate [kg/s]
N_{faces}	Number of faces around a control volume [-]
N_w	Density in the nucleation area [-]
n	Fluid constant [-]
$\vec{n}$	Unit vector normal to the surface of a control volume [-]
n_{nsd}	Nucleate site density empirical parameter [-]
Nu	Nusselt number, $Nu = 2.0 + 0.6 Re_{rel}^{1/2} Pr^{1/3}$ [-]
P	Pressure [Pa]
P_k	Function 3 of constants and variables of the $\kappa - \omega$ SST model [-]
$\tilde{P}_k$	Function 4 of constants and variables of the $\kappa - \omega$ SST model [-]
Pr	Prandtl number, $c_p \mu / k$ [-]
Q	Convective heat transfer rate per unit of volume [W/m ³]
q	Heat transfer rate [W]
q''	Heat flux [W/m ²]

$\vec{r}$	Displacement vector from the centroid of the face of the control volume to the centroid of the upstream control volume [-]
r_{ij}	Normalized x_{ij} in the TOPSIS method [-]
Re_D	Reynolds number, $\rho V D / \mu$ [-]
S	Perimeter [m]
S_ϕ	Energy source or sink term per unit volume [W/m ³]
T	Temperature [K]
t	Time [s]
V	Volume [m ³]
$\vec{V}$	Relative velocity vector [m/s]
v	Velocity in the y axis direction [m/s]
$\vec{v}$	Velocity vector [m/s]
v_{ij}	Weighted normalized x_{ij} in the TOPSIS method [-]
u	Velocity in the x axis direction [m/s]
u_∞	Terminal velocity [m/s]
w_j	Weight corresponding to a criterion j in the TOPSIS method [-]
x	Vapor quality [-]
x_{ij}	Numerical value of the i -th alternative with respect to the j -th criterion in the TOPSIS method [-]
y	Distance to the nearest wall [m]
z	Longitudinal distance inside a channel [m]

Greek symbols

α	Volume fraction [-]
β	Ad hoc constant of the $\kappa - \omega$ SST model [-]
β^*	Ad hoc constant of the $\kappa - \omega$ SST model [-]

Γ	Diffusion coefficient [-]
Δi_{sub}	Difference between the enthalpies of the saturated liquid and the liquid at the inlet of the channel [kJ/kg]
ΔT_e	Excess of temperature [K]
ΔT_{sub}	Degree of subcooling [K]
ζ	Bulk viscosity [N.s/m ²]
λ	Diffusivity [m ² /s]
μ	Dynamic viscosity [N.s/m ²]
μ_t	Turbulent dynamic viscosity [N.s /m ²]
ν	Specific volume [m ³ /kg]
ρ	Density [kg/m ³]
σ	Surface tension [N/m]
σ_k	Ad hoc constant of the $\kappa - \omega$ SST model [-]
σ_ω	Ad hoc constant of the $\kappa - \omega$ SST model [-]
$\bar{\tau}$	Stress-strain tensor [N/m ³]
τ_p	Particle relaxation time [s]
φ	Product of the relative Reynolds number and the vorticity Reynolds number [-]
Φ	Representation of constants of $\kappa - \omega$ SST model [-]
ϕ	Scalar quantity being transported [-]
ϕ_1	Ratio between the area of the first block over its area of occupation, A_2/A_1 [-]
ϕ_2	Ratio between the area of the second block over its area of occupation, A_2/A_1 [-]
ψ	Interphase momentum exchange coefficient [-]
ω	Specific turbulence dissipation rate [1/s]

Upper and lower indexes

1,max Once maximized

1,min	Once minimized
2,max	Twice maximized
2,min	Twice minimized
3,max	Three times maximized
3,min	Three times minimized
a	Actual
C	Convective
D	Diameter
dis	Distorted
E	Evaporative
F	Fluid dynamic
f	Sub-index referring to a given face of the control volume
in	Inlet
l	Liquid phase
NB	Nucleate boiling
nb	Neighboring control volumes
o	Once optimized
oo	Twice optimized
ooo	Three times optimized
ocu	Occupation
op	Operating
out	Outlet
p	Phase p
Q	Quenching
q	Phase q
rel	Relative

s	Surface
sat	Saturation
T	Thermal
v	Vapor phase
vis	Viscous
w	Wall

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamics
CHF	Critical Heat Flux
COP	Coefficient of Performance
CPU	Central Processing Unit
DFM4	Four-Equation Drift Flux Model
EAHE	Earth Air Heat Exchanger
FVM	Finite Volume Method
MIT	Massachusetts Institute of Technology
PISO	Pressure-Implicit with Splitting of Operators
RPI	Rensselaer Polytechnic Institute
SIMPLE	Semi-Implicit Method for Pressure-Linked Equations
SST	Shear Stress Transport
TOPSIS	Technique For Order of Preference by Similarity to Ideal Solution
UDF	User Defined Functions
VOF	Volume of Fluid

1. INTRODUCTION

This thesis proposes the study of the geometric evaluation of two-dimensional channels containing two heated rectangular blocks, mounted in an alternating way on their wall surfaces, subjected to forced convective boiling flows of water in the turbulent flow regime. This study was carried out through the use of numerical simulation tools to solve the proposed problems. The geometric evaluation aims to analyze the behavior of the performance indicators, fluid dynamics and thermal, with the variation of the established geometric degrees of freedom. For this, the principles of the Constructal Theory (Bejan and Lorente, 2008; Bejan and Zane, 2012; Bejan and Lorente, 2013; Bejan, 2020) were applied through the Constructal Design method associated with the exhaustive search, where several geometric possibilities within a search space were evaluated. In addition to these, the TOPSIS multi-criteria method (Hwang and Yoon, 1981; Hwang et al., 1993) was used in the multi-objective analysis of the geometric evaluation process.

1.1. Motivation

The constant technological evolution that occurs in engineering provides many advances and benefits for society, but also generates new problems to be solved by engineers and designers. One of these problems is the need to dissipate the heat generated by many systems (computers, machines, equipment, among others), a fact inherent to their functioning (Kondepudi and Prigogine, 2015). In recent decades, this problem has evolved due to advances in manufacturing technologies and industry sophistication, which have allowed the construction of equipment with extremely small dimensions and high performance, but which produce a high rate of unwanted heat generation (Ahmed et al., 2018).

Searching for ways to alleviate the problem of heat dissipation, the study of flow in dissipative channels and microchannels has been the subject of much research. These consist of one or several small channels that together form a single structure through which a confined fluid flows and dissipates the heat that is generated (Ohadi et al., 2012). They are present in heat exchangers in refrigeration systems, electronics and microelectronics, process industries, power generation systems, among many other applications (Ohadi et al., 2012; Siddiqui and Zubair, 2017; Rai et al., 2018).

Within the field of dissipative channels and microchannels, the study of the phase change of the flowing fluid stands out, especially that of boiling. Fluid boiling is associated with the formation of bubbles on the inner surface of the channels when it reaches a temperature above the boiling point

of the fluid and can result in an increase in heat dissipation rates (Kandlikar et al., 1999). This characteristic can make the boiling of flows in channels and microchannels attractive for systems that need high heat dissipation rates.

The application of the boiling process in flows inside channels and microchannels is a possible strategy that can provide an increase in the performance of the system (Karayiannis and Mahmoud, 2017; Çebi et al., 2018). Boiling causes a considerable increase in the heat transfer coefficient for a small difference of temperature between the fluid and the channel. This is a great advantage over single-phase flows, where a much greater temperature variation is required for a similar increase in the heat transfer coefficient to be observed. Also, boiling can bring benefits such as a lower flow rate required for the fluid when compared to single-phase flows with the same heat dissipation rate, which reduces the need for pumping the fluid (Markal et al., 2016). Another important benefit is the small temperature variations present in the channels. As a consequence, the durability of the equipment that they are part can increase due to the reduction of thermomechanical stresses (Cheng and Xia, 2017).

Regarding the study of channel and microchannel design (with or without phase change), the evaluation and the geometry optimization process can produce designs with better performances through a better understanding of the phenomena involved in the process. This thesis will apply the concepts of the Constructal Theory, through the Constructal Design method, to evaluate how the design of the channel can affect the system performance. The Constructal Theory determines that the evolution of the design of flow systems follows an universal law of physics, the Constructal Law, where its configuration freely evolves in such a way that it facilitates access to the currents that flow in this finite size flow system so that these systems can continue to exist over time (Bejan and Zane, 2012).

Despite the possible advantages of boiling, its numerical modeling and geometric evaluation processes still present many challenges and potential for improvement, mainly due to the high complexity of the physical phenomena involved (Kandlikar et al., 1999; Khan and Fartaj, 2011). There is a significant number of studies where boiling flows in channels are computationally reproduced. However, due to the complexity of the problems, most of these studies are limited to two-dimensional axisymmetric problems in circular channels or three-dimensional channels with a constant cross section and few studies are dedicated to complex geometries (Ferrari et al., 2018). In this way, the process of geometric optimization of channels with complex geometries with boiling flows and a better understanding of its computational modeling are currently highly necessary.

The development of computational technologies that took place in the last decades allowed several complex engineering problems to be addressed through computational modeling in a

completely new and revolutionary way. Due to the evolution of the processing capacity of computers, it is currently possible to solve problems of computational fluid dynamics (CFD) through commercial software or by creating specific algorithms, with an unimaginable precision and speed when compared to what was practiced in the 1980s and 1990s. Furthermore, the costs involved in numerical simulations today make them potentially advantageous when compared to the costs involved with experimental equipment and facilities (Versteeg and Malalasekera, 2007; Moukalled et al., 2016; Ferziger et al. al., 2020). The computational modeling of the boiling phenomenon is one of the great challenges of computational fluid dynamics and thanks to advances in programming and processor capacity in recent years, commercial software has begun to implement this phenomenon, as is the case with Ansys Fluent 18.1 (ANSYS, 2017a; ANSYS, 2017b), used in this thesis.

Therefore, the combination of the possibility of computational modeling of boiling flows along with the necessity to improve the research of geometric evaluation of dissipative channels and microchannels, present in several systems and equipment fundamental to human development, creates an opportunity for study with high relevance in a topic that still represents an open problem.

1.2. Bibliographic Review

The topic of internal boiling flows (flows inside channels) has received a high level of attention from researchers in recent decades due to the complexity of the problem and also due to the importance of its practical application. In this section, the main works carried out on the topics covered in this thesis will be presented. Firstly, it is presented a general context of the boiling flow theme, where it will be presented a review of the main works on boiling flows in conventional channels (diameters greater than 3.00 mm), in minichannels (diameters between 0.20 and 3.00 mm) and microchannels (diameters between 0.01 and 0.20 mm). Later, a review of the numerical modeling of this problem is depicted. In sequence, it will be presented the main works where the topic of geometric evaluation of channels with boiling flow was addressed, followed by a review of some important works that studied the Constructal Theory and the application of the Constructal Design method.

1.2.1. Boiling Flows in Conventional Channels

The analysis of convective flows without phase change in channels can be done from analytical equations and empirical correlations solidly established, where it is possible to verify the influence exerted by the different variables of the problem on the behavior of the flow and heat transfer. The results obtained from these equations and correlations serve for the development and optimization of channel geometries and are useful for the design of thermal equipment. The

phenomenon of heat transfer in boiling flows, however, is more complex than that observed in single-phase flows. There is greater difficulty in the analysis and mathematical modeling of problems of this type due to the complex interaction between the two phases with each other and with the geometry of the channels, in addition to the simultaneous presence of the different modes of heat transfer by convection and boiling (Rohsenow et al., 1998).

As a result of the complex interaction between the two phases in boiling flows inside channels, several thermal regions can be identified throughout the length of the flow, each with the predominance of a different heat transfer mechanism. In this way, the development of maps for boiling flows is an important topic, extensively addressed by several works. Collier (1981) presented a map for boiling flows in channels demonstrating the basic relationship between the heat transfer coefficient and the vapor quality (x), with the heat flux as a parameter. This map applies for water at low pressures and is an approximate representation of the behavior of the heat transfer coefficient for different regions. Later, Kandlikar (1988) investigated the validity of the map proposed by Collier and noted that the heat transfer behavior on the map is valid only for water at atmospheric pressure. He concluded that the use of this map (and others proposed by other authors at the time) is fluid specific and generally not applicable beyond the range of conditions applied to each experiment. After the initial study, Kandlikar (1991) proposed new maps, applicable to different types of fluids.

Many of the initial works on boiling flows, published from the 1960s onwards, focused on seeking a better understanding of the physical phenomena involved in the process, as in the work of Larsen and Tong (1969), who from experimental results proposed a semi-analytical model for the calculation of the volume fraction of the vapor generated in the boiling process. The model was based on the parameters of formation and growth of vapor bubbles. Even in the present, several authors extensively address this approach. Al-Hayes and Winterton (1981) investigated the diameter of bubbles formed in boiling flows. They proposed equations for the drag forces, buoyancy, and surface tension acting on the bubbles based on experimental results. Later, Winterton (1984) proposed a theory to estimate the detachment of bubbles from the channel walls. More recently, Guan et al. (2015) developed an analytical model to estimate the departure diameter of bubbles from experiments using a high-frequency digital camera (Figure 1.1).

An important issue in internal boiling flows is the critical heat flux (CHF), which is the point where a further increase in channel surface temperature causes a sudden drop in heat transfer due to the accumulation of vapor in the walls. This accumulation of vapor also causes a significant drop in the heat transfer coefficient. The achievement of the CHF can cause a catastrophic failure on machines or devices and, for this reason, the need to predict it is vital for the functioning of systems that depend

on boiling or evaporation. The CHF has been widely studied in recent decades due to its importance (Hewitt and Hall-Taylor, 1970; Tong et al., 1997) and several theories have been proposed to explain its mechanism, the main ones being those proposed by Kutateladze and Leont (1966), Tong (1968), Weisman and Pei (1983), Lee and Mudawar (1988) and Vijayarangan et al., (2006).

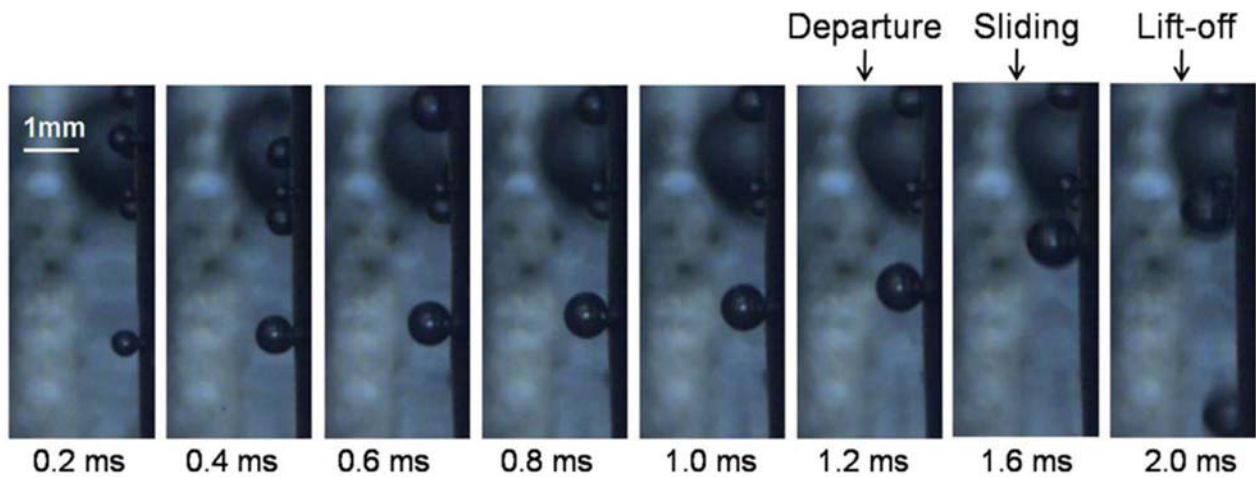


Figure 1.1 – Typical consecutive images of bubble growth, departure and lift-off (Guan et al., 2015).

One of the main challenges of the study of the boiling process is the correct determination of the heat transfer coefficient. In internal flows, this becomes considerably complicated as the two phases present generally have very different thermophysical properties. Chen (1966) defined one of the most used correlations to define the heat transfer coefficient. It was obtained through the sum of the coefficient for forced convection without phase change with the coefficient for a case of pool boiling, both obtained through empirical correlations. Wadekar (1996) proposed an improvement of the Chen correlation, where it was considered that the coefficient relative to boiling should be kept constant while the coefficient for forced convection should be gradually reduced according to the reduction of the area available for forced convection, due to the increased presence of vapor bubbles on the channel walls.

1.2.2. Boiling Flows in Minichannels and Microchannels

There is no consensus in the literature over the exact classification of channels according to their diameter, but some authors on the subject suggest some form of classification. One of the most accepted is that of Kandlikar and Grande (2003), which classified conventional channels as those with a diameter greater than 3.00 mm; minichannels as those with a diameter between 0.20 and 3.00

mm, and microchannels as those with a diameter between 0.01 and 0.20 mm. Channels with diameters smaller than 0.01 mm are called transition channels and nanochannels.

The number of works carried out on heat and mass transfer in small diameter channels has increased considerably in recent years. The motivation for these studies derives from the high demand for technological applications of microchannels combined with the fact that there are differences between microchannels (and minichannels) and conventional channels regarding the heat and mass transfer phenomena (Karayiannis and Mahmoud, 2017; Dadhich et al., 2021; Deng et al., 2021). Minichannels and microchannels find application in systems such as microevaporators, microcondensers, microdissipators, micropumps, microvalves, among others, and these systems are of vital importance for several industries. Microevaporators in particular, which consist of one or more microchannels where the fluid absorbs heat and gradually evaporates, are widely used for cooling various thermal systems, such as the central processing unit (CPU) of computers and also in various types of microelectronic components (Thome and Cioncolini, 2017).

One of the pioneers to investigate the phenomenon of boiling in small diameter channels were Lazarek and Black (1982). They experimentally tested a vertical channel with a diameter of 3.1 mm where there was a saturated flow of R-113 with the intention of measuring the local heat transfer coefficient. They found that with the conditions applied in the test, the heat transfer mechanism is dominated by the boiling process rather than convection. They proposed a correlation for the local heat transfer coefficient using the Nusselt number as a function of the Reynolds number. Nevertheless, it is necessary to note that this correlation cannot be applied to conditions different from those applied in this experiment.

In a similar experiment to that of Lazarek and Black, Bao et al. (2000) conducted experimental tests using fluids R-11 and HCFC123 in minichannels with diameter of 1.95 mm. Mass fluxes between 50 and 1,800 kg/m².s, heat fluxes between 5 and 200 kW/m² and saturation pressures between 200 and 500 kPa were tested. It was concluded that, in boiling flows, the heat transfer coefficient is independent of mass flux and vapor quality, but it is strongly influenced by heat flow and saturation pressure. They found some trends similar to those found by Lazarek and Black (1982). Huo et al. (2004), however, found a more complex behavior for vapor quality, especially for values greater than 20%, as shown in Figure 1.2, where from this value a large drop in the heat transfer coefficient is observed. The values of vapor quality below zero means that the flow is subcooled, where the channel's walls have a temperature higher than the saturation temperature but the bulk temperature of the flow is below that point. They studied channels with diameters between 2.01 and 4.26 mm using R134a and a mass flux of 300 kg/m².s.

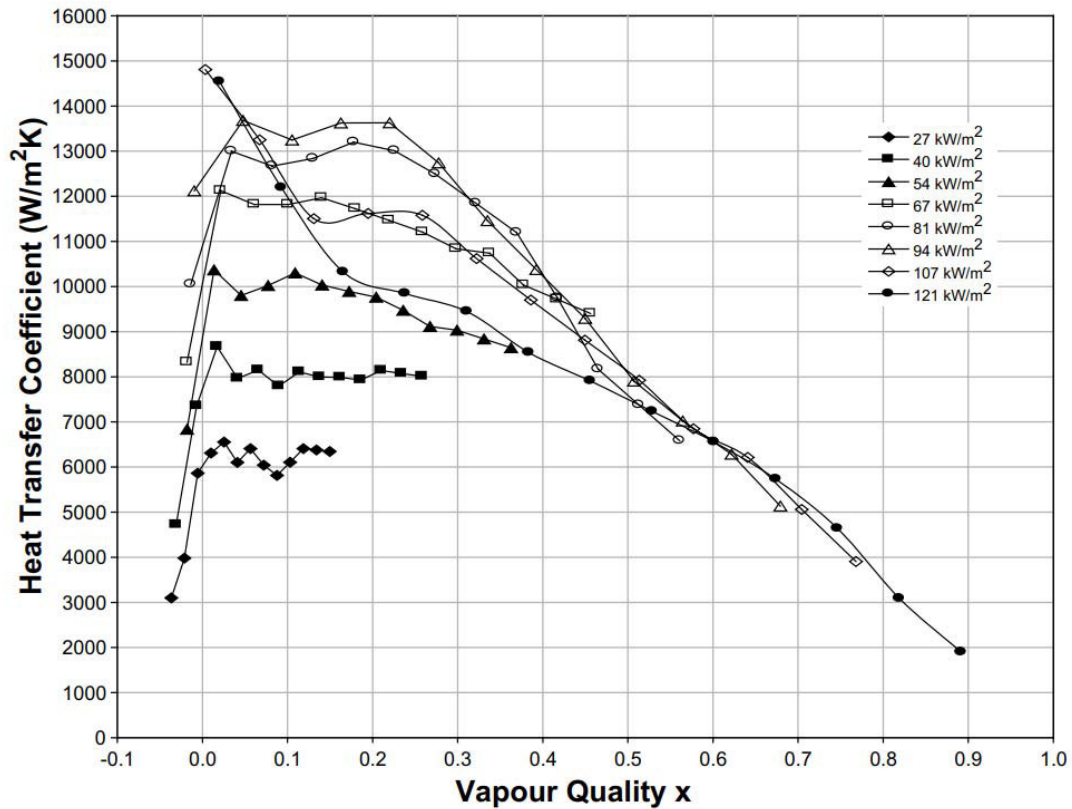


Figure 1.2 – Influence of vapor quality (x) over the heat transfer coefficient, obtained in the work by Huo et al. (2004).

Warrier et al. (2002) studied boiling in 0.75 mm diameter minichannels, using FC-84. They applied mass fluxes from 557 to 1,600 $\text{kg/m}^2\text{s}$, heat fluxes between 0 and 59.9 kW/m^2 and temperatures of the liquid at the inlet of the channel of 26, 40 and 60 $^{\circ}\text{C}$. From the results obtained, they proposed a correlation for the heat transfer coefficient using only the boiling number (Bo) and the vapor quality (x). According to the authors, this correlation is valid for $0.00027 \leq Bo \leq 0.00089$ and for $0.03 \leq x \leq 0.55$.

Zhang et al. (2004) performed a broad comparison of existing correlations for heat transfer in minichannels and microchannels with boiling flow and found that these are not suitable for the microscale. They concluded that, due to many of the correlations used so far considered the two phases, liquid and gas, both in the turbulent flow regime, they would not be appropriate for calculating the heat transfer coefficient in microchannel boiling flows. For the latter case, generally, only the gas phase is in the turbulent regime, while the liquid phase is in the laminar regime. Also, a new correlation was proposed, based on that of Chen (1966), capable of determining the heat transfer coefficient in minichannels for any of the four combinations of two-phase flow regime (laminar-laminar, laminar-turbulent, turbulent-laminar and turbulent-turbulent). A comparison with several

experimental results showed that the new correlation by Zhang et al. (2004) was able to obtain satisfactory results.

Lee and Mudawar (2005) experimentally studied the boiling flow of R134a in copper microchannels with a diameter of 0.23 mm. They observed that different levels of vapor quality are associated with different heat transfer mechanisms. It was also observed that at vapor qualities lower than 5%, the mechanisms of bubble formation and nucleated boiling occur. At intermediate values, between 5 and 55%, or values greater than 55%, but with high heat fluxes, boiling is dominated by the annular film evaporation mechanism. They proposed three correlations that are recommended for the boiling flow of R134a and water in microchannels, each according to the level of vapor quality.

Chen and Garimella (2006) studied boiling flows in dielectric fluids in rectangular minichannels, each with dimensions of $0.389 \text{ mm} \times 0.389 \text{ mm}$, integrated into silicon microchips, in order to observe and analyze the different flow regimes and the behavior of the heat transfer during the boiling process. They observed that with low heat fluxes, the main flow regime was that of bubbles. As the heat flux increased, the bubbles coalesced and pockets of vapor formed. Increasing the heat flux even more, there was the formation of annular flows, which, depending on the mass flow, varied between ordered and agitated. It was also observed that, in the cases studied for intermediate heat fluxes, the heat transfer coefficient is independent of the mass flow and that for heat fluxes greater than 542 kW/m^2 , the heat transfer coefficient decreases when the heat flux increases, due to the liquid being practically all evaporated at the end of the microchannel in these cases.

Based on the various correlations that existed so far, Sun and Mishima (2009) carried out an extensive evaluation of heat transfer models for boiling flows, both in minichannels and microchannels as well as in conventional sized channels. They confirmed that the correlation proposed by Chen (1966) and also those based on that are not suitable for boiling problems in microchannels, only in conventional channels and minichannels. Among the correlations analyzed by Sun and Mishima, that of Lazarek and Black (1982) and also that of Kew and Cornwell (1997) are the ones that best represent the phenomenon of boiling in microchannels among those analyzed in the study, indicating the predominance of the nucleated boiling in microchannels, which is considered in these two correlations. Based on the results obtained, a new correlation was proposed based on those of Lazarek and Black (1982) and Kew and Cornwell (1997).

In an experimental work with pore-lined parallel metallic microchannels, Bai et al. (2013) studied the boiling of anhydrous ethanol, comparing the results of these microchannels with those without the presence of pores. They investigated differences in pressures, heat transfer and also boiling instability. The results showed that an increase in pressure drop occurs in the microchannel

with pores and also a significant attenuation of its fluctuation. An improvement in boiling stability was observed, which can be explained by changes in bubble dynamics. A large increase in heat transfer was also observed in the microchannel with pores, especially at low values of vapor quality. The authors concluded that this was due to more favorable conditions for bubble nucleation in the porous microchannels, which decreased as the vapor quality increased.

Charnay et al. (2014) studied the boiling flow of R-245fa in horizontal minichannels with 3 mm of diameter in order to analyze the effect of heat flux over the heat transfer coefficient, for mass flows of $700 \text{ kg/m}^2\cdot\text{s}$ at two different saturation temperatures: 60 and 80 °C. At 60 °C, the heat transfer coefficient was strongly influenced by heat flux only for low vapor qualities, while at 80 °C it was strongly influenced by heat flux for all levels of vapor quality. They concluded that the higher the saturation temperature, the greater the heat transfer coefficient, as well as the greater the influence of the nucleate boiling mechanism over the coefficient.

More recently, designers and engineers have been proposing some strategies to mitigate the heat dissipation problem in minichannels and microchannels. Some examples are seeing in the use of nanofluids (Fang et al., 2015; Wang et al., 2020; Mukherjee et al., 2022) the use of porous surfaces (Chen et al., 2017; Zhang et al., 2018; Otomo et al., 2021), the addition of micro and nano particles of cooper and other metallic materials to modify the channel surface (Morshed et al., 2013; Sujith Kumar et al., 2016; Borba Marchetto and Ribatski, 2020) and the insertion of nano structures like nano-wires and carbon nanotubes (Kousalya et al., 2015; Kumar et al., 2017; Alam et al., 2018; Li et al., 2020).

1.2.3. Numerical Modeling of Internal Boiling Flows

Numerical modeling of internal boiling flows presents great challenges and it has been the subject of several recent works and studies. The constant advances in computer technologies and numerical simulation of physical phenomena have allowed this field of study to have an increasing number of possibilities and different problems to be solved. Among the main topics studied are the creation of numerical models capable of reproducing the boiling phenomenon with precision, the refinement and evolution of these models, the comparison with experimental and analytical results and also their practical application in real situations and conditions.

One of the first and best-known computational models was the one proposed by Kurul and Podowski (1991), known as the RPI Boiling model because they developed it at the Rensselaer Polytechnic Institute. Since then, this model has been recognized as a reference for modeling the phenomenon of boiling flows. Although it was proposed based on pool boiling (boiling of a

stationary, confined fluid) experiments, this model was adapted to be able to simulate boiling flows. According to the model, the total heat flux from the channel walls to the liquid is given by the sum of three components, which are: the convective heat flux of the liquid phase, the quenching heat flux and the evaporative heat flux. To calculate these heat fluxes, an area of internal surface of the channel, A_b , which represents the area where the formation of bubbles occurs, is considered, as well as the expression $(1 - A_b)$ which represents the rest of the internal area of the channel, occupied by the liquid phase.

Juric and Tryggvason (1998) proposed a model to simulate boiling flows where a system of conservation equations for mass, momentum and energy are used to describe both phases. They added source terms to describe surface tensions, mass transfer between phases and also latent heat. The equations were discretized by a finite difference method and the boundary between the phases is explicitly calculated. The model is applicable for low vapor volume fraction values (below 30 %). The results for the heat transfer rates and the temperature of the channel walls showed a good agreement compared to experimental and analytical results, however there were considerable differences with other works in the representation of the interface of the two phases.

Most numerical models of boiling flows are suitable for flows with high pressures and are not applicable for flows with low pressures (close to atmospheric pressure) due to the complexity of the interaction between the two phases. Tu and Yeoh (2002) studied this problem and proposed a numerical model for pressure-independent boiling flows. They developed a functional model for low pressures from a high-pressure model, using CFX-4.2 code. They demonstrated that three parameters are essential for the correct determination of the volume fraction along the flow: the heat flux in the channel walls, the average diameter of the vapor bubbles and the bubble departure diameter. The results obtained with this model had a very good agreement with experimental results. The model was also implemented in the RELAP5/MOD2 code and significantly increased the accuracy of the results for the volume fraction in boiling flows with low pressures. Subsequently, Koncar et al. (2004) also investigated the problem of boiling at low pressures and proposed a model considering lift, wall lubrication and turbulence forces, in addition to considering the variations in bubble diameters according to local flow conditions. They obtained good results compared to experimental results in the literature.

Steiner et al. (2005) presented a numerical model for the heat transfer from channel walls in boiling flows. They proposed a model similar to that of Chen (1966), in which there is a superposition of the heat transfer rate of the walls, where the total heat flux is considered as the sum of the convective and nucleated boiling heat transfer components. Two suppression factors were added in

order to better represent the contribution of nucleated boiling, these being related to the effects of flow forces and the thermal boundary layer. The comparison with experimental results, using water as a fluid, showed good results for low values of vapor quality, however some discrepancies were observed as this value increases.

Nayak et al. (2006) applied an Eulerian numerical model for boiling flows to the study of boiling instabilities in a thermosyphon reactor system. Instabilities are extremely undesirable in these systems as they can induce oscillations in the reaction rate or in the heat generation rate in the catalysts. In this model, the formulation of the two phases is considered separately. The MONA computer code was used (the authors of the model do not explain the meaning of the acronym MONA). It contains seven conservation equations based on three flow fields: liquid, vapor and liquid particles in vapor regions. The conservation equations are three for the conservation of mass, two for the conservation of momentum and two for the conservation of energy. To facilitate the solution, the mass conservation equation was decoupled from the momentum, energy and volume fraction equations. The time step was limited by the Courant criterion. They concluded that conventional homogeneous multiphase models tend to overestimate the volume fraction of vapor generated in the reactors and that the Eulerian model can simulate flow with good accuracy compared to measurements in experimental reactors. With this simulation they were able to determine the exact amount of heat that the reactors can withstand without creating harmful instabilities in the system.

Using the Volume of Fluid (VOF) multiphase model, Yang et al. (2008) studied the boiling flow of R141B in a horizontal serpentine channel. In the VOF model, the volume fraction occupied by each phase, in a given computational cell, is calculated and the sum of the value of all phases in this cell must always be equal to 1. The software used in the simulations was Ansys Fluent 6.0. As this version of the software did not yet contain the phase change process implemented in its source code, the authors created their own code, to be inserted into the program (known as User Defined Functions or UDF). The geometric reconstruction scheme was used to represent the interface between the phases and the $k-\varepsilon$ model was used as closure model of the time-averaged equations of turbulence. For the pressure-velocity coupling, the PISO scheme was used. The authors concluded that the simulations presented consistent results compared to the experiments, and they were able to successfully show the different flow and heat transfer regimes observed during the boiling process.

Aiming to study the behavior of vapor bubbles in flows with boiling in microchannels in relation to their hydrodynamics and heat transfer, Magnini et al. (2013) numerically studied the problem, using the Ansys Fluent 12 software. The vapor bubbles studied were more precisely those of the vapor pockets type, which form and occupy practically the entire diameter of the channel, with

only a thin layer of liquid around the channel walls. The VOF model was used to reproduce the interface between the phases. This model was improved with a height function method created by the authors and implemented via UDF, to better estimate the local capillarity effects and also better calculate the local rates of mass and energy exchange at the interface of the two phases. With this model, the authors were able to accurately detail the dynamics of the vapor bubbles. The results showed that, under typical operating conditions of microchannel boiling flow experiments, the vapor pockets accelerate towards the microchannel exit following an exponential function with respect to time, in agreement with existing theoretical models.

The growth and coalescence of vapor bubbles is also a topic recently addressed by numerical simulation studies. Ling et al. (2015) used a code based on the finite volume method in this problem, which was represented three-dimensionally (Figure 1.3). The Projection method was used to solve the velocity and pressure fields. The interface between the phases was calculated using the VOSET model (an acronym for Volume of Fluid and Level Set), which is a combination of the VOF and Level Set models. The results were compared with experiments performed previously by other authors and showed good agreement. The authors verified that the appearance of vapor bubbles significantly increases the heat transfer in the microchannel. As observed by Magnini et al. (2013), the growth of bubbles accelerates the flow of liquid around them to the outlet of the microchannel. They also analyzed the impact of the fusion of vapor bubbles over the heat transfer and found that it can produce an increase in heat flux over time. However, this is not as significant as that caused by the thin layer of liquid between the bubbles and the microchannel walls, which is mainly responsible for the high heat flux existing in flows with boiling in microchannels.

Regarding flow regimes present in boiling flows, Liu et al. (2017) applied numerical modeling to study the transition in a microchannel from the slug regime to the annular regime, where a thin layer of liquid forms around the walls. As well as Ling et al. (2015), the authors also used the VOSET model. They proposed a new initialization model, where the initial velocity and temperature profiles are generated from a simulation without a phase change. Regarding the transition between flow regimes, the results obtained a good comparison with a visual evaluation of experimental data. The authors demonstrated that the vapor bubbles grow faster when the flow approaches the annular regime. This occurs because the coalescence of the bubbles with the annular region causes a disturbance of the thermal boundary layer and, therefore, there is an increase in evaporation at the front of the vapor bubbles.

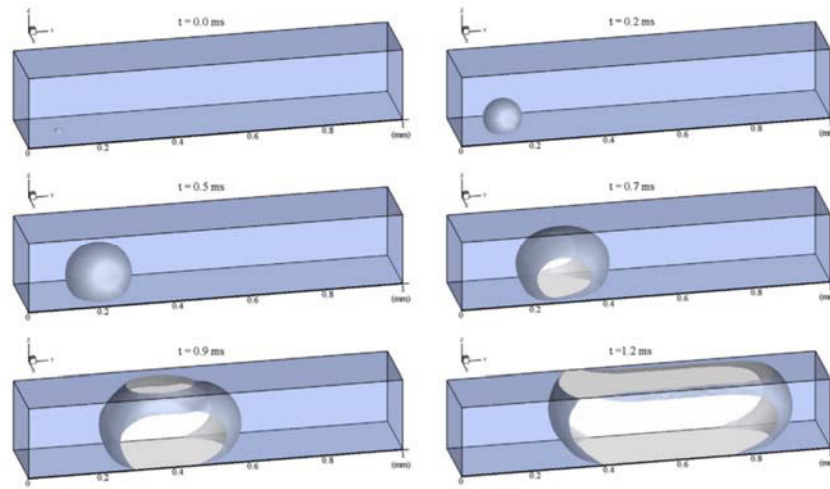


Figure 1.3 – One of the simulations performed in the work by Ling et al. (2015).

Free open-source softwares are also used in the modeling of boiling flows in channels, as in the study by Ferrari et al. (2018), in which OpenFOAM 2.3.1 software was employed. This software uses the finite volume method for the numerical solution and has the VOF multiphase model. The authors implemented modifications in the software for modeling the phase change, since OpenFOAM does not have this functionality. They analyzed the effects of cross-sectional geometry (square and circular) and also of liquid flow in microchannels. It was possible to observe that in relation to the two analyzed geometries, there is a significant difference in the dynamics of the vapor bubbles and also in the heat transfer. In square geometry the bubbles travel at a higher speed than in circular geometry while the thin layer of liquid that forms on the walls has values an order of magnitude smaller. These effects have a great influence over the heat transfer in square geometries, since there is a very large increase in the heat flux on the walls and a decrease in the corners.

OpenFOAM software was also used by Luo et al. (2020) in the study of boiling in microchannels with square cross section. Using the VOF multiphase model present in the software, the authors added source terms to simulate the interactions of mass, momentum and energy between the phases. To model the phase change, it was employed the model proposed by Rattner et al. (2014), known as the Temperature Recovery Model, in which a constant saturation temperature is applied to the cells at the interface between the two phases at each time step. The model was validated with experimental results and they were able to observe that with an increase in the heat flux or in the quality of the vapor at the entrance of the channel, there is a decrease in the thickness of the liquid layer that forms on the channel walls, which causes an increase in heat transfer. Also, keeping the heat flux fixed at a certain value, the heat transfer coefficient decreases as the mass flux increases.

The ease of creating new computer codes certainly had a positive impact on the study of CFD,

providing the opportunity for researchers to create and modify existing codes, as performed in the study by Bhati and Paruya (2021). They numerically studied the dynamics of vapor bubbles in a boiling flow in a vertical channel. The four-equation drift flux model (DFM4) was used, which has already been successfully used in previous works and is similar to the Eulerian model used by Nayak et al. (2006), where the formulation of the two phases is considered separately. They performed experimental tests to validate the model in addition to comparing the results obtained with analytical solutions. The results of the numerical simulations compared with the analytical solutions were considered extremely good for the Unal correlation, however they registered low values for the exit diameter of the vapor bubbles compared to other correlations. They also noted that it is necessary to consider the effect of the evaporation microlayer on the bubble exit diameter, which is accentuated in flows with low mass flow rates, close to the boiling situation in a pool.

1.2.4. Geometric Evaluation of Channels with Boiling Flow

The geometric evaluation of channels is a topic widely addressed both experimentally and numerically. The design of new, more efficient geometries contributes enormously to the performance of different types of machines, equipment and systems. The challenge is to understand the effects of geometry on the overall performance of these systems. The correct identification of the objectives of a geometric optimization in a system is fundamental because many times conflicting objectives are present simultaneously, since the search for the improvement of a parameter defined as the objective of the geometric optimization can cause a negative effect in another important parameter of the system. In the case of dissipative channels, the objective of increasing the heat transfer can cause an undesirable increase in the flow resistance in the channel if this factor is not considered in the geometric optimization. This fact applies to flows with or without phase change. Many of the studies of geometric evaluation in channels were carried out in flows without phase change due to its simplicity, with a smaller number of works having been carried out considering the boiling process.

Regarding flows in channels without phase change, several types of geometries have already been studied, such as circular, rectangular, square, trapezoidal, triangular, hexagonal, in addition to more complex geometries (Perret et al., 1998). Cavities, obstacles and fins are commonly used in channels with the intention of increasing the interaction between the fluid and the channel and thus increasing heat dissipation (Xia et al., 2011). Channels with different shapes such as zig-zag, curved and wave shape have also been studied (Mohammed et al., 2011a; Mohammed et al., 2011b). Several studies on channel geometry have shown that it is possible to significantly increase thermal

performance with geometric optimization (Ghani et al., 2017; Qasem and Zubair, 2018; Naqiuddin et al., 2018; Iwaniszyn et al., 2021).

The topic of geometric evaluation of channels with boiling flow has gained the attention of researchers in the last decade. Few works were found on this topic prior to 2010. Seeking to explore the problem of instability in boiling flows, Lu and Pan (2008) experimentally studied parallel microchannels with a divergent cross section, with an average diameter of 120 μm and an angle divergence of 0.5° , where water was used as the fluid. In addition to verifying that heat flux and mass flux significantly affect flow stability, they observed that microchannels with divergent cross-section increase flow stability in relation to channels with constant cross-section. Subsequently, Miner et al. (2013) advanced research on diverging channels with boiling flow. They also observed that diverging channels increase the stability of these flows and concluded that there is an optimal rate of expansion of the cross-sections of the channels, and proposed a mathematical model to calculate it. Also, it was observed that boiling flows in diverging channels provide a higher heat transfer rate before reaching the critical heat flux.

One possibility to be investigated is the influence of fins and microfins in channels. In this topic, He et al. (2012) conducted experiments to evaluate the influence of microfins in channels where the R410A fluid boiled. The channels were tubes made of copper with an internal diameter of 11.4 mm and a length of 2 m. The height of the fins was changed between 0.152 and 0.272 mm, as well as the number of microfins was changed between 60 and 75 and the spiral angle formed by the microfins, between 18° and 23° . The study was limited by the very small number of experiments carried out, however it was possible to observe that the geometry has a considerable influence on the heat transfer and the boiling process. It was noted that the greater the number of fins, the greater the heat transfer. Regarding the height and spiral angle of the microfins, it was concluded that these two parameters had an optimal intermediate value in relation to heat transfer, and more studies are needed to better determine these values.

The effects of diameter and length in minichannels with boiling were studied by Sadaghiani and Kosar (2016). They experimentally and numerically analyzed stainless steel channels with internal diameters between 0.6 and 0.9 mm and lengths between 60 and 120 mm. The software used for the numerical simulations was Ansys Fluent 14.0, using the Eulerian multiphase model, $k-\epsilon$ turbulence model and RPI boiling model. The main observation of the work was that the diameter has an inversely proportional relationship with the heat transfer coefficient, while the length has a proportional relationship. It was also observed that the boiling heat transfer is a function of the diameter, length and conditions of the flow. As the mass flux increases, greater heat fluxes are

achieved. Another observation, already made by several authors, is that in larger mass fluxes, the convective heat transfer mechanism is dominant over nucleated boiling. However, the effect of mass flux over the heat transfer is stronger compared to the effect of heat flux as the diameter is reduced.

The relationship between the width and height of the cross section in rectangular copper microchannels with boiling flow was studied by Özdemir et al. (2019). They experimentally investigated the influence of the width to height ratio (W/L) for three different values: 0.5, 2.56 and 4.94, for a mass flux between 200 and 800 kg/m².s and a heat flux between 4 and 1350 kW/m². They observed that the pressure difference between the inlet and outlet of the microchannels decreased with increasing width-to-height ratio. For the case with heat flux equal to 600 kW/m² for example, there was an 11% decrease in the pressure difference from W/L equal to 0.5 to W/L equal to 2.56, and a further 35% decrease in W/L equal 2.56 to 4.94.

Differences between circular, square and triangular cross-sections in boiling microchannels were investigated by Sempértegui-Tapia and Ribatski (2017). They conducted 1,943 experimental tests, using the fluid R134a, heat flux between 15 and 85 kW/m² and mass flux between 100 and 800 kg/m².s. It was observed that the heat transfer coefficient increases proportionally to the heat flux for the three types of geometry. For circular and square geometries, the highest values of the heat transfer coefficient were obtained with the lowest values of heat flux. In the triangular geometry, this trend did not occur, with the highest heat transfer coefficients being found with the highest heat fluxes and for values between low and intermediate vapor quality. From the results obtained in the tests, it was also observed that some methods developed in the past, such as the one by Chen (1966), served well for the prediction of these flows, for the three types of geometry. Despite this, none of these methods was able to adequately calculate the heat transfer coefficient for the triangular geometry.

Gao and Bhavnani (2019) experimentally studied different types of geometries in microchannels with boiling in order to reduce flow instabilities. They proposed geometries in which the cross section varies periodically (Fig. 1.4), called toothed geometries, so that the fabrication process of such channels would have low complexity. The proposed geometries induce flow recirculation, which increases the mixing of the two phases, reduces the size and increases the amount of vapor bubbles, increasing heat transfer and reducing flow instability. A high-speed camera was used to visualize the flow. The results were compared with channels with constant cross section to verify the effects of the new proposed geometries. The results showed that the heat transfer was increased in all cases of the toothed geometries compared to the geometry with constant cross section. It was observed that for the toothed geometries, keeping the heat flux up to a certain maximum value, there is a mixture of the two phases sufficient to avoid the occurrence of reverse flow and thus

significantly reduce the instabilities in the flow.

As demonstrated, most works on the geometric evaluation of channels with boiling flow seek a better understanding of the influence exerted by geometry on the behavior of the boiling process and on the flow performance. Aiming at the geometric optimization of rectangular aluminum microchannels with boiling, Ariyo and Bello-Ochende (2019) numerically studied this problem, using deionized water as a fluid. They evaluated mass fluxes between 99.82 and 4,491.9 kg/m².s and heat fluxes between 1×10^6 and 6×10^6 W/m². To perform the numerical simulations, the software Ansys Fluent 18.1 was used and a three-dimensional model was considered, with the Eulerian multiphase model and the RPI boiling model, and the flow was considered laminar. The geometric parameters of height, width and length, as well as the thickness of the microchannel, were varied in order to obtain the geometry that presented the lowest possible thermal resistance and to compare the results with flows without phase change. The results showed that the lower the Reynolds number, the greater the difference between the thermal resistance of flows with and without boiling, being smaller in flows with boiling. Although they do not present the results for the pressure drop along the microchannel, the authors believe that this, although greater in boiling flows, is not sufficient to cause a significant drop in the overall performance of the microchannel. Therefore, the thermal advantage of boiling flows makes them preferable to flows without phase change, within the parameters used in this work.

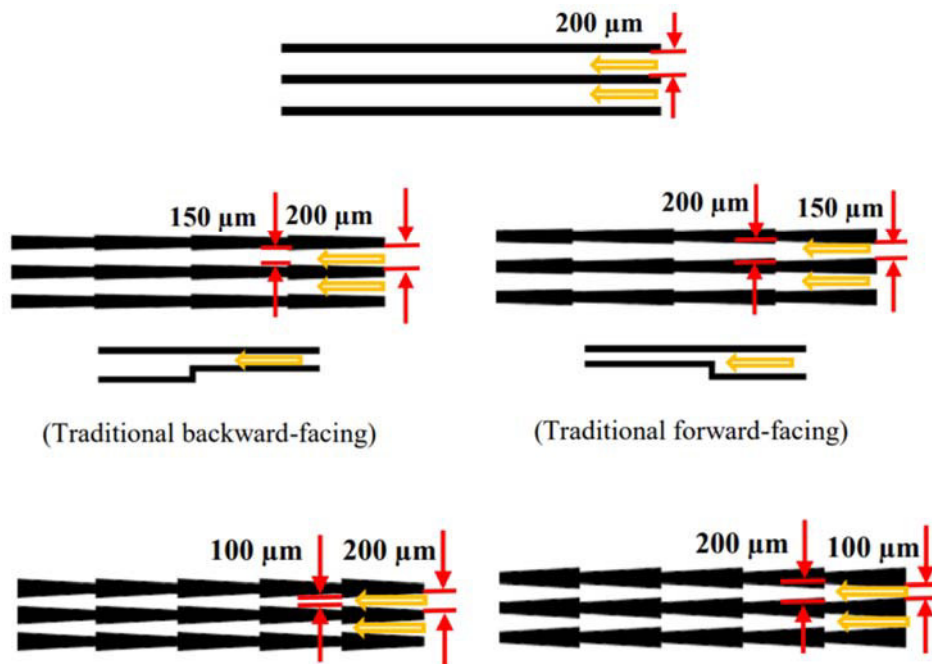


Figure 1.4 – Geometries proposed by Gao and Bhavnani (2019).

1.2.5. Constructal Design Applied to Heat Transfer and Fluid Dynamics

In the scope of the study of geometries of fluid systems, which is the case of flows inside channels, the Constructal Design method stands out, created from the Constructal Theory developed by the romanian mechanical engineer Adrian Bejan. Since the 1990s, several works have been developed applying the principles of the Constructal Theory, but it was only from the 2000s that the first works appeared where the method known as Constructal Design was applied, especially in problems of heat transfer and fluid dynamics.

Some of the first works with the application of Constructal Design were dedicated to problems of conductive paths. Rocha et al. (2002) studied conduction cooling of disc-shaped bodies in order to optimize the design of inserts made of high-conductivity material present in the disc. Concepts such as restrictions (total disk volume and insert volume) and objectives (minimization of global thermal resistance) were applied. From an analytical investigation, the development of branches in the geometry of the high conductivity inserts was observed, forming a tree-shaped structure, where, in order to maintain the objective of minimum thermal resistance, there was an increase in ramification with increasing radius of the disks.

The application of Constructal Design in numerical problems of heat conduction in solids with cavities was also extensively explored. Geometries with cavities in the shape of rectangular (Rocha et al., 2005), H (Biserni et al., 2007), T (Lorenzini et al., 2014), T-Y (Lorenzini and Rocha, 2009; Lorenzini et al., 2012) have already been addressed. In practically all these works, the geometries that best addressed the problem of heat conduction, that is, the geometries where the maximum heat dissipation occurred, were the geometries where the cavities had the greatest penetrations in the solids, since these configurations were the ones that generally provided better heat distribution throughout the solid. More recently, Biserni et al. (2017) applied the Constructal Design to the numerical analysis of a conduction problem in a trapezoidal solid that generated heat coupled to a T-shaped fin exposed to a low-temperature flow. For the geometric optimization process, exhaustive search and genetic algorithms were used. The results showed that the freedom given to the fluid system to transform itself, in the form of the possibility of adapting the geometry, provided a better thermal performance to the system. According to the authors, this result agrees with the principle of “optimal distribution of imperfections” proposed by the Constructal Theory.

More recent studies on the application of the Constructal Design in conduction problems in solids with cavities have added more complexity to the analyzed systems. In Lorenzini et al. (2015), a three-dimensional cylindrical solid with several cavities was studied in order to minimize the maximum temperature in excess of the solid from geometric optimization (Figure 1.5). The work

demonstrated that for a fixed value of the fraction of area occupied by the cavities, there is an optimal number of cavities that minimizes the maximum temperature in excess and this number increases proportionally to the increase of the fraction of area of the cavities. As the geometry evolves to more complex configurations, the system better redistributes heat and thus increases performance. Later, Estrada et al. (2020) studied a similar geometry, this time with extra branches in each cavity, in a sort of tree-like configuration. Better results were found than in Lorenzini et al. (2015) since the greater freedom for adaptation allowed the geometry to have a better heat distribution, thus confirming the principles of the Constructal Theory.

The geometric evaluation of cavities with fins subjected to convective flows has also been studied with Constructal Design, as in the work by Lorenzini et al. (2016), where there was mixed convection within a cavity with a rectangular fin. The problem was numerically simulated and was subject to two constraints: the cavity area and the fin area. The cavity geometry was optimized with the objective of maximizing the heat transfer by mixed convection between the fin and the flow and the problem was analyzed for different values of the Reynolds and Rayleigh numbers to investigate the effect of the fin geometry over the Nusselt number. The application of the Constructal Design was able to improve the thermal performance of the system and demonstrated that the mixed convection mechanism has a significant influence on the fin geometry. This work served as a basis for similar ones where different geometries were evaluated, as in Razera et al. (2018) with a semi-elliptical fin and in Rodrigues et al. (2020) in which two rectangular fins were used.

The Constructal Design is also applicable to the development of sustainable energies such as wave (Martins et al., 2018), ocean thermal (Wu et al, 2019), ground thermal (Nunes et al., 2021) and solar energies (Miguel, 2008; González, 2016; Ojeda et al., 2021). It has also attracted some interest from solid mechanics more recently (Lima et al., 2020; Mardanpour et al., 2021; da Silveira et al., 2021).

Whithin the field of ground thermal energy, Rodrigues et al. (2015) applied Constructal Design in the numerical study of the thermal potential of ground-to-air heat exchangers (EAHEs). EAHEs are systems that take advantage of the thermal energy present in the soil to cool or heat environments through ducts through which the air circulates, which cross the soil and enter the desired environments, providing an adequate climate for these environments. Five different configurations for the EAHE were tested, with the number of ducts varying between 1 and 4, and the degree of freedom was defined as the ratio between the vertical and horizontal spacing between the ducts. Two constraints were also defined for the problem: the volume fraction occupied by the air and the volume fraction occupied by the EAHE in the soil. The geometry that presented the highest value of the

thermal potential was considered the best, being the one with four ducts in a rectangular arrangement. The results showed that Constructal Design can be successfully used in the geometric optimization of EAHEs.

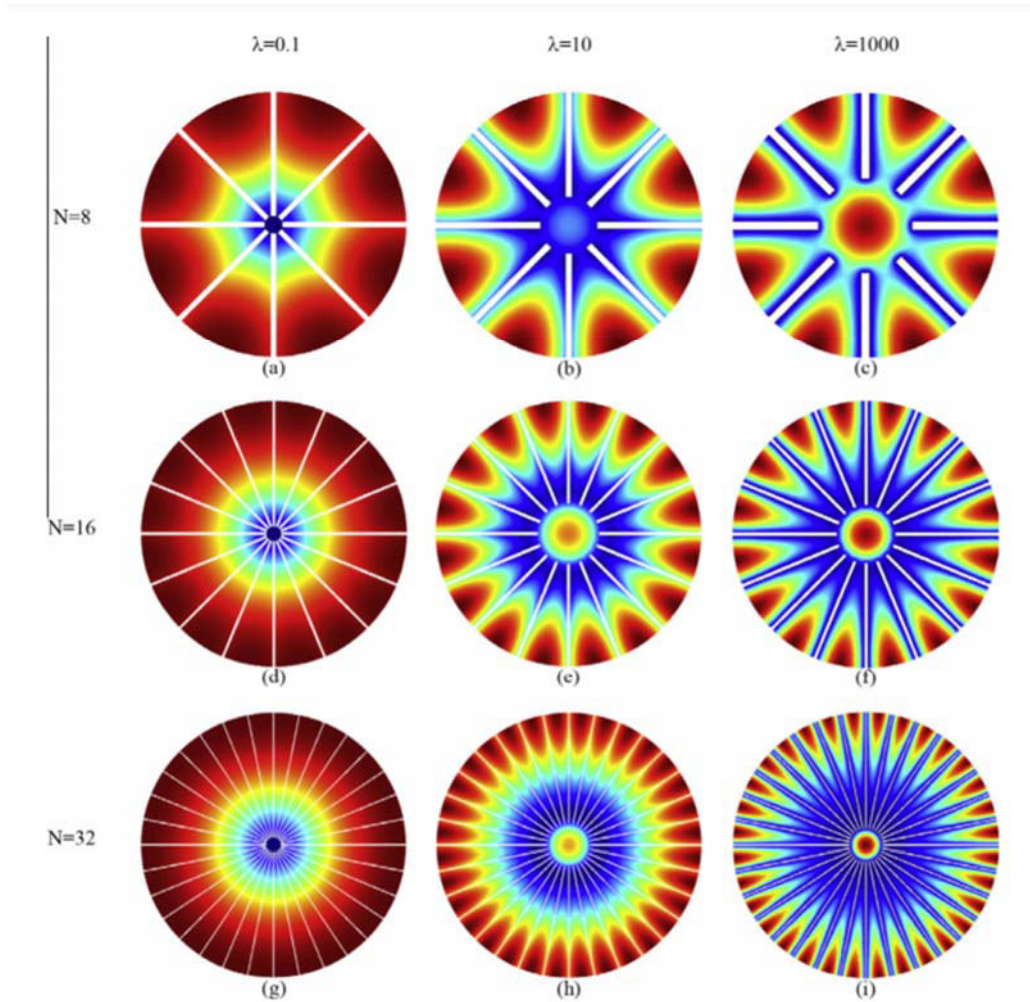


Figure 1.5 – Temperature fields of some of the geometries studied by Lorenzini et al. (2015).

Vieira et al. (2017) studied the Constructal Design applied to the geometric evaluation of solar chimneys through numerical simulations. Three degrees of freedom were defined referring to the geometry of the chimney and also three restrictions were defined: the areas of the chimney, the collector and the turbine. The objective of the work was to evaluate the effect of the geometry over the available power of the solar chimney. It was observed that the design of the chimney and the collector need to be considered in combination to optimize the performance of the system. The results also showed that the ratio between the radius and the height of the chimney has an optimal intermediate value and that, therefore, extreme chimney designs do not provide a good performance.

The authors considered that the application of Constructal Design was effective for the geometric evaluation of the solar chimney, justifying its use in future works.

Teixeira et al. (2018) numerically studied the turbulent convective flow around three blunt rectangular bodies, using Constructal Design for the geometric evaluation of the arrangement of bodies (Figure 1.6). The objective of this work was to evaluate the effect of the arrangement geometry on the drag coefficient and on the Nusselt number, thus being a multi-objective problem. The problem had two constraints, the cross-sectional area of the bodies and the occupancy area of the arrangement, and also two degrees of freedom, the ratio between the longitudinal step and the size of the bodies and the ratio between the cross step and the size of the bodies. The results showed that the geometry of the arrangement of bodies has a great influence on the behavior of the thermal and fluidic performance of the analyzed flow. The multi-objective evaluation found the same geometries found by the fluid dynamic analysis as optimal geometries, which demonstrated in this case study a strong predominance of the drag coefficient in relation to the Nusselt number. These results demonstrate that in a multi-objective analysis, the flow regime can affect the dominance of one of the objectives in a geometric evaluation of an obstacle arrangement.

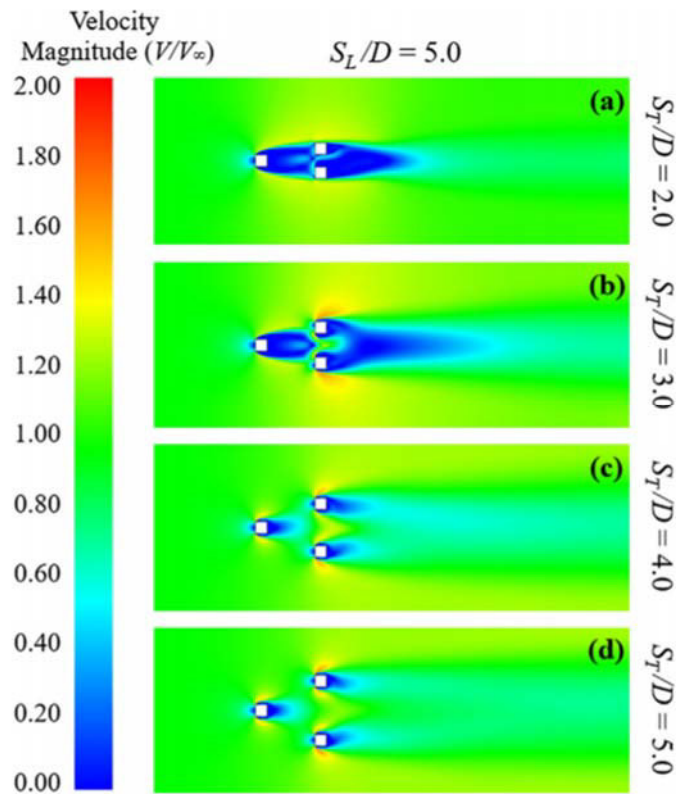


Figure 1.6 – Temporal mean of velocity fields for some of the cases studied by Teixeira et al. (2018).

Freire (2021) applied the Constructal Design to numerically investigate the geometric influence of the heating surface of a finned cavity over the heat convection with boiling. The work used the ANSYS Fluent software to perform the simulations and the VOF and Lee evaporation-condensation models were both employed. The numerical model was validated comparing the results obtained with a correlation given by Rohsenow et al. (1998) for the heat flux in nucleate pool boiling. The results showed that the geometry (ratio between the height and length of the fins) and the number of fins have a strong effect over the heat transfer rate and over the vapor generation. It was found that the geometry with the greatest number of fins and greatest insertion into the reservoir among those analyzed obtained the highest heat transfer rate and highest vapor generation.

1.2.6. Constructal Design Applied to Channels

Constructal Design has also been successfully used to study channels with flows. With the application of the method, it is possible to evaluate the influence of the channel geometry over the flow and heat transfer, and thus propose improvements to the channel design.

In one of the first works to apply Constructal Design in the geometric evaluation of channels, Bello-Ochende et al. (2007) numerically studied microchannels with rectangular cross-sections heated through which a low-temperature flow passed. The aim of the study was to find the geometry that resulted in the lowest temperature on the walls. The length of the channel, the relationship between the wall thicknesses and the ratio between the height and length of the microchannel were defined as degrees of freedom for the problem. This work demonstrated that the geometric variables defined as degrees of freedom had a strong influence on the microchannel heat transfer in the studied problem.

In a new study, Bello-Ochende et al. (2009) applied Constructal Design to the numerical study of the geometric evaluation of rectangular ducts where a part of the length at the entrance of the channel had wrinkles in its profile, in order to increase the interaction between the cooled flow and the heated duct and thus increase heat transfer. The flow was in the laminar regime and the Constructal Design was used in the geometric evaluation of the geometry of the wrinkled channels and the results showed that the use of wrinkles, compared to ducts without it, produces a 15% increase in heat transfer.

Song et al. (2015) used Constructal Design in the geometric evaluation of wavy channels with flow in the laminar regime and with heat transfer, with the specific objectives of finding the geometry that presented the highest heat transfer and the lowest pressure drop. The upper and lower walls of

the channel were undulating and the degrees of freedom were defined as the width of the channel entrance and the ratios between the wavelengths and the amplitudes of the two walls. The results showed that with the application of Constructal Design it was possible to find geometries that increased heat transfer by up to 26% and reduced pressure drop by 54%.

The use of fins (also called as blocks or bodies) in channels is one way to increase the interaction between the fluid and the channel and thus increase the heat transfer. Feijó et al. (2018) numerically studied a channel with two rectangular heated fins, one on the upper wall and another on the lower wall of the channel, where there was air flow in the laminar regime (Figure 1.7). The Constructal Design was applied for the geometric evaluation of the two fins. The problem had two objectives: to maximize the heat transfer rate between the fins and the flow and to minimize the pressure difference between the inlet and outlet of the channel, and was subjected to five constraints and two degrees of freedom, these being the ratios between the height and width of each fin. The influence of the fraction of the area occupied by the two fins in the channel was also evaluated. As expected, the geometry where the fins had the greatest intrusion into the channel was the best from the thermal objective point of view while the geometry where the fins had the least intrusion was the best from the fluid dynamics point of view. When considering the multi-objective, the best geometry was one in which the fins had intermediate values, but in an asymmetric way. This work demonstrated the usefulness of the Constructal Design methodology in understanding multi-objective problems in internal flows and also reinforced the importance of multi-objective analysis in convective flow problems in channels.

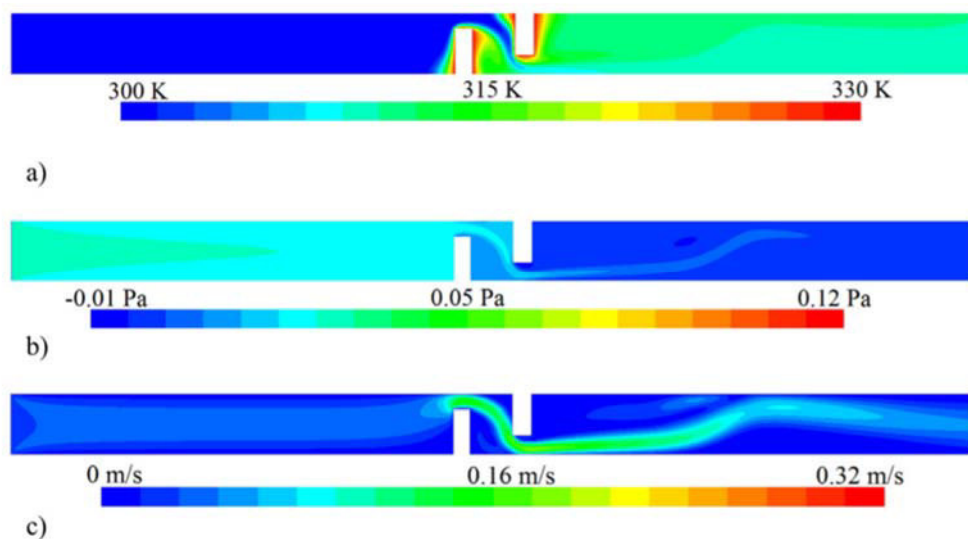


Figure 1.7 – Temperature (a), pressure (b) and velocity (c) fields of the optimal geometry found in the work by Feijó et al. (2018).

1.3. Objectives

The general objective of this thesis is to numerically study the influence of geometrical configurations by means of Constructal Design of two heated rectangular blocks mounted in channels surfaces subjected to turbulent convective flows over fluid dynamic and thermal performance (multiobjective problem) with and without phase change (boiling).

1.3.1. Specific Objectives

In the present work, the geometric optimization is performed through the Constructal Design method associated with the exhaustive search method. In this process, two objectives are established: maximizing the total heat transfer rate between the heated region of the channel and the surrounding flow (thermal objective) and minimizing the pressure difference between the inlet and outlet of the channel (fluid dynamic objective). To analyze the geometric evaluation process considering simultaneously the two proposed objectives (multi-objective analysis), the TOPSIS method is applied. The following specific objectives are established in this work:

- Evaluate the effect of the ratio between the width and height of the first block (H_1/L_1) over the heat transfer rate (q) and over the pressure difference between the inlet and outlet of the channel (ΔP) for the two cases under consideration: with and without boiling;
- Evaluate the effect of the ratio between the width and height of the second block (H_2/L_2) over the heat transfer rate once maximized $(q)_{1,\max}$, and over the pressure difference between the inlet and outlet of the channel once minimized $(\Delta P)_{1,\min}$, and over the ratio H_1/L_1 once optimized $(H_1/L_1)_o$, for the two cases under consideration;
- Evaluate the effect of the ratio between the distance of center of the two blocks and the length of occupation (L_3/L_{ocu}) over the twice maximized heat transfer rate $(q)_{2,\max}$, and over the pressure difference between the inlet and outlet of the channel twice minimized $(\Delta P)_{2,\min}$, over the twice-optimized ratio H_1/L_1 $(H_1/L_1)_{oo}$ and over the once-optimized ratio H_2/L_2 $(H_2/L_2)_o$, for the two cases under consideration;
- Evaluate the effect of the boiling process over the studied objective functions and over the optimal geometries;
- Investigate the influence of the geometrical configurations over the fluid dynamic and thermal behavior of the turbulent convective flows for the cases with and without phase of change;
- Evaluate the effect of the geometrical configurations over the relative proximity to the ideal solution C_b , for the two cases under consideration;

- Evaluate the effect of the geometrical configurations over the Coefficient of Performance (*COP*), for the two cases under consideration.

2. THEORETICAL FOUNDATION

This section aims to present the most relevant theoretical foundations regarding the topics covered in this thesis. It is divided into three parts: in the first, the fundamentals of forced convective boiling flows inside channels are presented. In the second part, the Constructal Theory, its principles and its application through the Constructal Design method are presented. In the third part, the multi-criteria method TOPSIS, applied in the multi-objective analysis in this study, is presented.

2.1. Fundamentals of Boiling

Boiling can be defined as the process of adding heat to a liquid in a way that leads to the formation of vapor bubbles due to heating it above its boiling temperature by contact with a solid surface (Collier and Thome, 1994). This process is present in several important applications for society, such as in machinery and equipment in the energy generation sector, in the chemical industry, refrigeration, air conditioning, among many others (Kandlikar et al., 1999).

According to the conditions of the fluid and the boiling process, different types of boiling can occur and be defined based on the fluid motion state and the physical mechanism involved in the process. The main way of classifying boiling is in relation to the fluid motion. In this sense, there are two different situations: the first, called *pool boiling*, where the fluid is initially at rest and in contact with a heated solid surface, and the second, called *forced convective boiling*, where the fluid initially has a relative velocity with respect to the heated surface (this situation can be further subdivided into internal and external boiling flows). Regarding the physical mechanism, the types of boiling are: *nucleate boiling* and *film boiling*. Nucleate boiling occurs at the liquid-solid interface with the formation of vapor bubbles through nuclei. As the temperature of the solid surface increases, bubbles will eventually form, which end up growing and detaching from the surface. Film boiling occurs when a layer of vapor forms around the heated surface due to surface temperatures considerably higher than saturation, thus causing heat to be transferred by conduction and radiation through this layer to the region where there is the liquid phase.

Another way of classifying boiling can be defined according to the temperature of the liquid. *Subcooled boiling* occurs when the average temperature of the flow is below the saturation temperature. Vapor bubbles form at the point of contact between the liquid and a heated solid surface and can condense as they move and approach the lower temperature region of the liquid. The other possible situation is called *saturated boiling*, which is when the temperature of the liquid phase is equal to its saturation temperature. It is important to mention that different boiling mechanisms and

classifications can occur simultaneously in different regions of the same flow and that is usually the case in internal boiling flows.

An important concept in boiling is *excess temperature*, also known by some authors as *wall superheat temperature*, which is determined by the following equation (Bejan, 2013):

$$\Delta T_e = T_w - T_{sat} \quad (2.1)$$

where T_w is the temperature at which the heated solid surface is (K) and T_{sat} is the saturation temperature of the fluid, referring to the pressure at which it is (K).

The progressive increase in excess temperature causes changes in the dynamics of the formation of vapor bubbles and causes a considerable increase in the heat flux up to a certain point. This point is known as *critical heat flux* (CHF) and is related to the transition from nucleate boiling to film boiling and when it is exceeded it causes a sudden drop in heat flux.

2.1.1. Pool Boiling

Although there are some significant differences in the process between pool boiling and forced convective boiling, it is important in the present thesis that pool boiling is also addressed because its concepts are important for a better understanding of boiling flows.

Pool boiling is defined as boiling that occurs in a static liquid, confined by solid surfaces, where a part (or all) of the solid surface is at a temperature above the saturation temperature of the liquid. As the name implies, a pool in which the walls are heated and the water is boiling is a typical example of pool boiling. The formation of vapor bubbles occurs at the interface of the heated solid with the liquid, and the bubbles tend to ascend due to the difference in densities between the liquid and vapor phases (Kandlikar et al., 1999).

Figure 2.1 qualitatively illustrates the typical boiling curve for a case of pool boiling of a saturated liquid. It demonstrates the relationship between the heat flux from the wall to the liquid and the excess temperature (ΔT_e) and also shows the different stages of the pool boiling process, according to the level of excess temperature. Initially, up to point A, the heat transfer occurs only by natural convection. Point A represents the beginning of nucleate boiling, when small vapor bubbles begin to form, grow and later detach from the solid-liquid interface. As ΔT_e increases, the density and frequency with which the vapor bubbles appear and the detachment increases. From point B to point C, the vapor bubbles that arise begin to merge into larger bubbles and may take the form of jets of bubbles from the solid-liquid interface. Point C marks the point where critical heat flux occurs and,

from this point forward, a layer of steam quickly covers most of the wall, causing a significant increase in the wall temperature. After a transition period where boiling happens extremely unstable, the heat flux increases again at point D, with the beginning of film boiling, but now with a considerably higher ΔT_e .

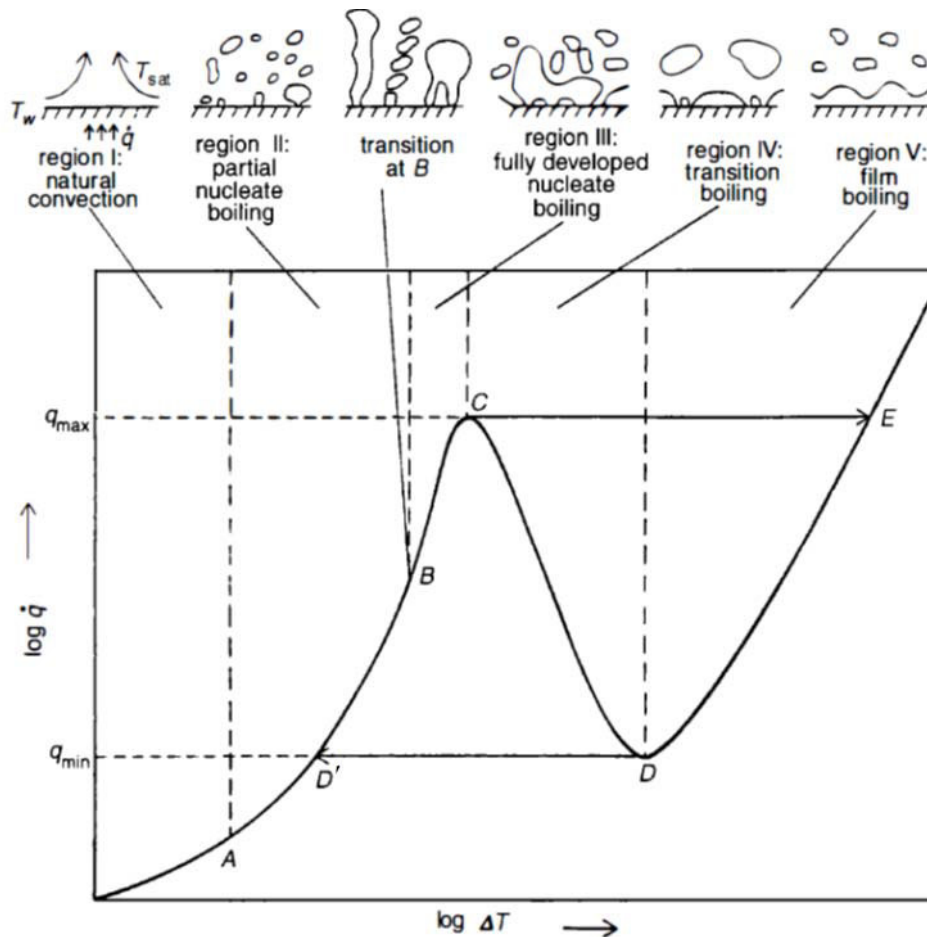


Figure 2.1- Stages of pool boiling (Kandlikar et al., 1999).

An important observation about the boiling curve is that it is only obtained correctly when heating methods that allow independent control of ΔT_e are used. When some means of power control is used to supply heat to the medium, where ΔT_e is a dependent variable, the curve “jumps” from point C to E (in heating) and from D to D' (in cooling). These observations were made by Nukiyama (1934) and later confirmed by Drew and Mueller (1937).

The main way to calculate heat transfer in pool boiling is through correlations. Several authors have already published numerous correlations that seek to describe the dynamics of heat transfer. One of the best known and most used is the Rohsenow (1952) correlation. It is valid for the nucleated

boiling region and it takes into account the effects of the solid-liquid interface, being given by:

$$\frac{c_{p,l}\Delta T_e}{i_{lv}} = C_{s,f} \left\{ \frac{q''}{\mu_l i_{lv}} \sqrt{\frac{\sigma}{g(\rho_l - \rho_v)}} \right\}^{0,33} \left[\frac{c_{p,l}\mu_l}{k_l} \right]^n \quad (2.2)$$

where $c_{p,l}$ is the specific heat of the liquid phase at constant pressure (kJ/(kg.K)), i_{lv} is the latent heat of the fluid (kJ/kg), $C_{s,f}$ is a constant that depends on the liquid and the surface finish of the solid, q'' is the heat flux through the solid towards the fluid (W/m²), μ_l is the dynamic viscosity of the liquid phase (N.s/m²), σ is the surface tension (N/m) of the fluid, g is the acceleration of gravity (m/s²), ρ_l and ρ_v are the densities of the liquid and vapor phases (kg/m³), k_l is the thermal conductivity of the liquid (W/(m.K)) and n is a constant equal to 1 for water and 1.7 for other fluids.

Taking the term $c_{p,l}\mu_l/k_l$ as the Prandtl number of the liquid (Pr_l) and rearranging Eq. (2.2), it is possible to obtain an equation for heat flux for nucleate boiling:

$$q'' = \mu_l i_{lv} \left[\frac{g(\rho_l - \rho_v)}{\sigma} \right]^{1/2} \left(\frac{c_{p,l}\Delta T_e}{C_{s,f} i_{lv} Pr_l^n} \right)^3 \quad (2.3)$$

This equation is valid to calculate the heat flux up to the CHF point. The CHF point can be determined through another correlation given by Zuber et al. (1963):

$$q''_{crit} = K \rho_v^{1/2} i_{lv} [\sigma g(\rho_l - \rho_v)]^{1/4} \quad (2.4)$$

where K is a dimensionless constant that depends on the geometry and is related to instabilities existing in boiling.

2.1.2. Forced Convective Boiling

The dynamics of the boiling process has significant differences between the cases of pool boiling and forced convective boiling. In this, the fluid has its movement due to some force exerted on it, added to the buoyancy effect on the vapor bubbles, while in pool boiling, the fluid movement occurs only due to the circulation of the bubbles by the buoyancy. This difference adds considerably more complexity to the study of the boiling phenomenon.

Forced convective boiling is divided into two forms: internal and external. On the external, the flow is directed around heated surfaces such as plates, cylinders, spheres, either individually or

over a set of these. Some authors call this type of boiling *cross flow boiling*. Its main applications in industry are in horizontal thermosyphons and reboilers, commonly used in distillation operations (Whalley, 1992; Kandlikar, 2002). On the other hand, boiling in internal flows takes place inside ducts and is the type to be studied in this thesis. The most important applications where boiling occurs are in internal flows and include vertical thermosyphons, compact heat exchangers, nuclear reactor components, among others (Karayiannis and Mahmoud, 2017).

2.1.3. Forced Convective Boiling in Internal Flows

Despite the bubble formation phenomenon of nucleated boiling being equal to that of pool boiling, the boiling process in internal flows presents significant differences in relation to this one. There is greater complexity in the dynamics of the process and several new characteristics and parameters must be taken into account. This greater complexity also means that boiling problems in internal flows are approached in a relatively different way. In these problems, there is a very large interaction between the flow and the boiling process, since as boiling occurs along the flow, different flow patterns develop according to the amount of steam that is generated and each of these patterns uniquely influence heat transfer and boiling (Rohsenow et al., 1998; Bejan and Kraus, 2003; Incropera et al., 2007).

Boiling Flow Patterns in Channels

In internal boiling flows, it is possible to observe that some patterns naturally form along the flow. These patterns emerge as vapor bubbles begin to form and interact with the liquid. This interaction significantly alters the heat transfer mechanisms that take place between the channel walls and the fluid, mainly due to the different thermophysical properties generally existing between liquids and gases. The main difference is between the densities of the two phases, with gases typically having a much lower density than liquids. Because of this, as the flow starts to contain more vapor than liquid, the tendency is for the overall velocity of the flow to increase considerably. This is one of the main factors that alter the flow dynamics. In addition to the lower density, gases generally have thermodynamic properties that lead to less heat transfer compared to liquids (Kandlikar et al., 1999).

A good way to demonstrate the different flow patterns that occur in internal boiling flows and their respective heat transfer regions is through Figure 2.2, by Collier and Thome (1994), which represents a case of a vertical channel with upward flow. This figure is a typical representation of boiling in vertical internal flows. It is important to say that there are some differences between the vertical and horizontal boiling flow patterns, but in general these two situations are relatively similar. The main flow patterns found in internal boiling flows (vertical upwards) are:

Bubbly flow: In this region, the beginning of bubble formation occurs along the heated walls of the channel. They vary in size but are still small relative to the diameter of the channel. Boiling starts as subcooled and then becomes saturated;

Slug flow: At this stage the liquid is already fully saturated and the vapor bubbles coalesce and fuse with each other, forming bubbles of significant size in relation to the diameter of the channel. Depending on the flow parameters, they can have a shape similar to firearm bullets, known as Taylor bubbles (Bejan and Kraus, 2003);

Annular flow: From a certain point in the flow, only a thin layer of liquid remains around the channel walls, with small liquid droplets in the vapor region at the center of the channel;

Mist flow: The liquid layer around the walls is completely evaporated and only small liquid droplets remain in the vapor medium.

The boiling process in internal flows begins when the liquid is still subcooled, that is, its average temperature in a given section is lower than its saturation temperature for the pressure that defines the thermodynamic state. At this point, the temperature of the liquid near the wall is high enough to cause boiling to begin, and we have the so-called subcooled boiling. In this region (bubbly flow), the vapor bubbles that form can condense when they come into contact with the most central region of the flow, which still has a lower temperature than the saturation temperature. Later, at a certain point, the temperature of the center of the flow reaches the saturation temperature, and it conducts to the beginning of the saturated boiling, where the vapor bubbles can spread from the walls to the center of the flow. The vapor bubbles that form get bigger and bigger and merge with each other (slug flow) until eventually only a thin layer of liquid remains around the walls of the channel, which tends to decrease in size as the flow and boiling advance (annular flow). After this layer of liquid is completely transformed into vapor, only small droplets of liquid remain in a region entirely of vapor (mist flow), which undergoes a gradual increase in its temperature. From this region, there is also a large increase in wall temperature and the heat transfer coefficient undergoes an equally large decrease. Finally, the liquid droplets are completely evaporated and only superheated steam remains in the flow, which returns to be a simple single-phase flow.

In the study of boiling in internal flows, two concepts are very important and commonly used in the literature as a way of measuring the boiling process. They are *quality* and *volume fraction*.

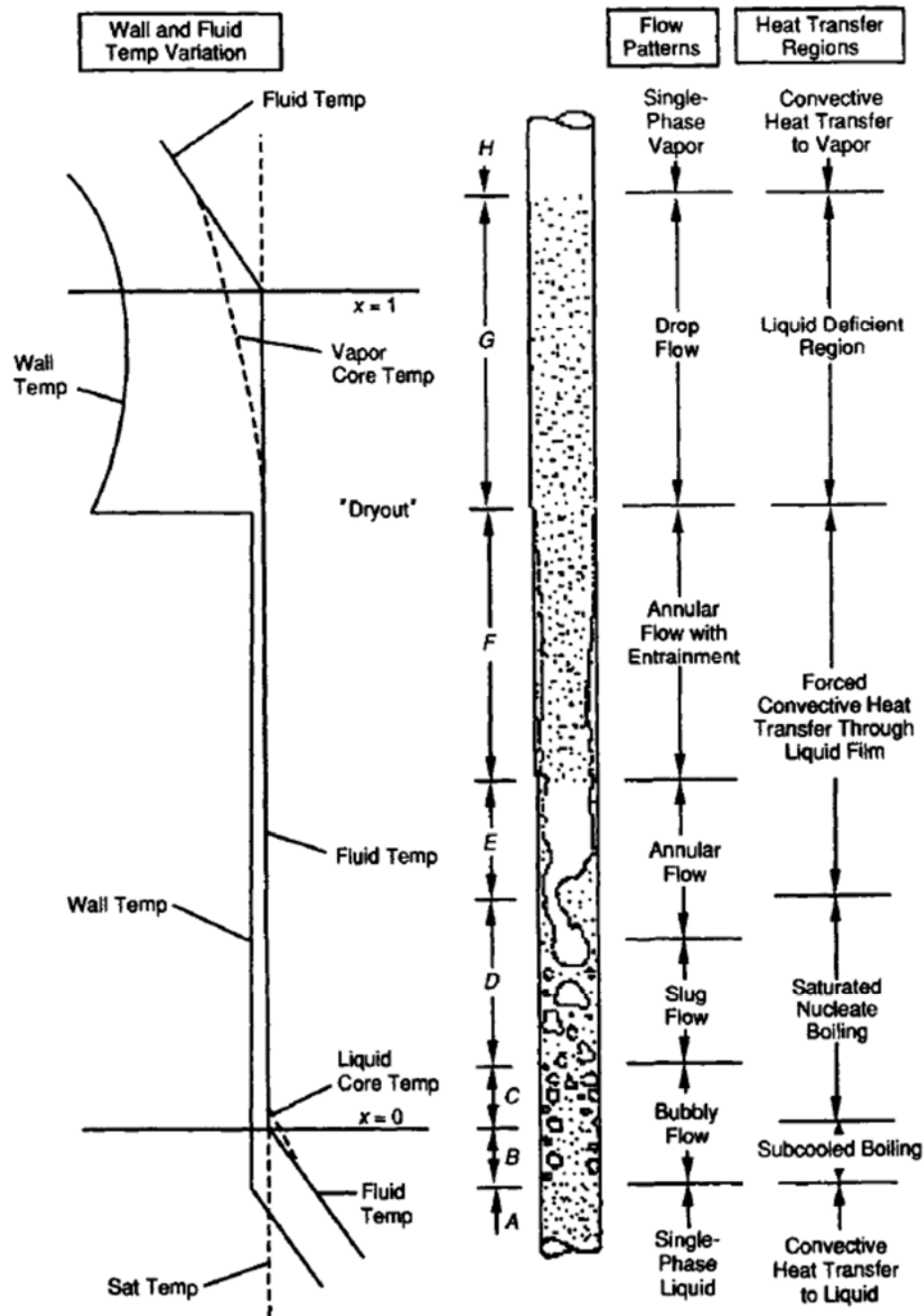


Figure 2.2 - Patterns of flow and heat transfer in vertically and upward convective boiling flows (Collier and Thome, 1994).

Vapor Quality

As can be seen in Figure 2.2, the vapor quality (x) varies between zero and one between the beginning and end of the saturated boiling process and it can also have values less than zero and greater than one. If the enthalpy of the fluid is smaller than the saturation enthalpy of the liquid phase, vapor quality will be lesser than zero, which means that the boiling process has not started yet. If the

enthalpy of the fluid is greater than the saturation enthalpy of the vapor phase, vapor quality will be greater than one and the boiling process is completed. The vapor quality is also called by some authors as simply *quality* or even as the *mass fraction of the vapor in the fluid*. The latter expression simply indicates what vapor quality really is: the amount of mass of vapor present in a given region of the flow, at a given instant of time. It can be defined as the ratio between the mass flux of vapor (G_v) over the total mass flux (G). Quality can be calculated from the enthalpy at a given point in a channel. For a channel of cross-sectional area A and perimeter S , the enthalpy i at a distance z from the channel entrance is given by (Rohsenow et al., 1998):

$$i = i_{ent} + \frac{S}{GA} \int_0^z q'' dz = xi_v + (1 - x)i_l \quad (2.5)$$

where i_{ent} is the enthalpy of the fluid at the entrance of the channel (J/kg), x is the vapor quality, G is the mass flux (kg/m².s), i_v and i_l are the saturation enthalpies of the vapor and liquid phases, respectively (J/ kg). For a channel with a circular cross section, the integral in Eq. (2.5) can be replaced by $q''z$. In this way, knowing the enthalpy in a certain point in the channel, it is possible to calculate the vapor quality from the following equation:

$$x = \frac{i - i_l}{i_v - i_l} = \frac{i - i_l}{i_{lv}} \quad (2.6)$$

As mentioned earlier, it is possible to observe from Eq. (2.6) that depending on the enthalpy of the fluid, the vapor quality can also be negative or greater than one. Nevertheless, it is also possible that the boiling process can began when the vapor quality is smaller than zero and still be ongoing when it is greater than one. This fact occurs because the quality given by Eq. (2.6) is related to the thermodynamic equilibrium condition, however, as can be seen below, the thermodynamic non-equilibrium effects exert an important influence over the boiling process in channels.

Vapor Volume Fraction

The volume fraction of the vapor phase, as its name reveals, corresponds to the volume fraction occupied by the vapor in relation to the total volume of the fluid in a given region of the flow. Some authors prefer to use the volume fraction instead of the quality because it conveys a better idea of the amount of vapor present in the fluid. For example, a given section of a water flow that has a volume fraction of vapor equal to 0.3 has 30 % of its space occupied by vapor, while another section that has a quality of vapor of 0.3 has a much higher amount of steam than 30% occupying this space

(depending on the flow pressure, it can be more than 90 %).

A problem present in the calculation of the quality and volume fraction of vapor in internal flows is that the quality determined by Eq. (2.6) refers only to the thermodynamic equilibrium condition, which is not normally the case in this type of problem. What actually happens in internal flows with boiling is that the appearance of vapor bubbles happens before $x = 0$, in the subcooled boiling region. Because of this, it is necessary to calculate the *actual quality* (x_a). Figure 2.3 demonstrates the variation of the vapor volume fraction along an internal boiling flow, comparing the real situation with that of thermodynamic equilibrium. It is possible to observe the point where the beginning of nucleate boiling occurs, defined as z_n , which can be calculated by the equation given by Davis and Anderson (1966):

$$z_n = \frac{G D c_{p,l}}{4} \left[\left(\frac{8 \sigma T_{sat} v_v}{k_l i_{lv} q''} \right)^{1/2} - \frac{1}{h_l} + \frac{T_{sat} - T_{in}}{q''} \right] \quad (2.7)$$

where D is the internal diameter of the channel (m), v_v is the specific volume of the vapor phase (m^3/kg), h_l is the convective heat transfer coefficient for the liquid phase ($\text{W}/\text{m}^2\cdot\text{K}$) and T_{in} is the temperature of the fluid at the inlet of the channel.

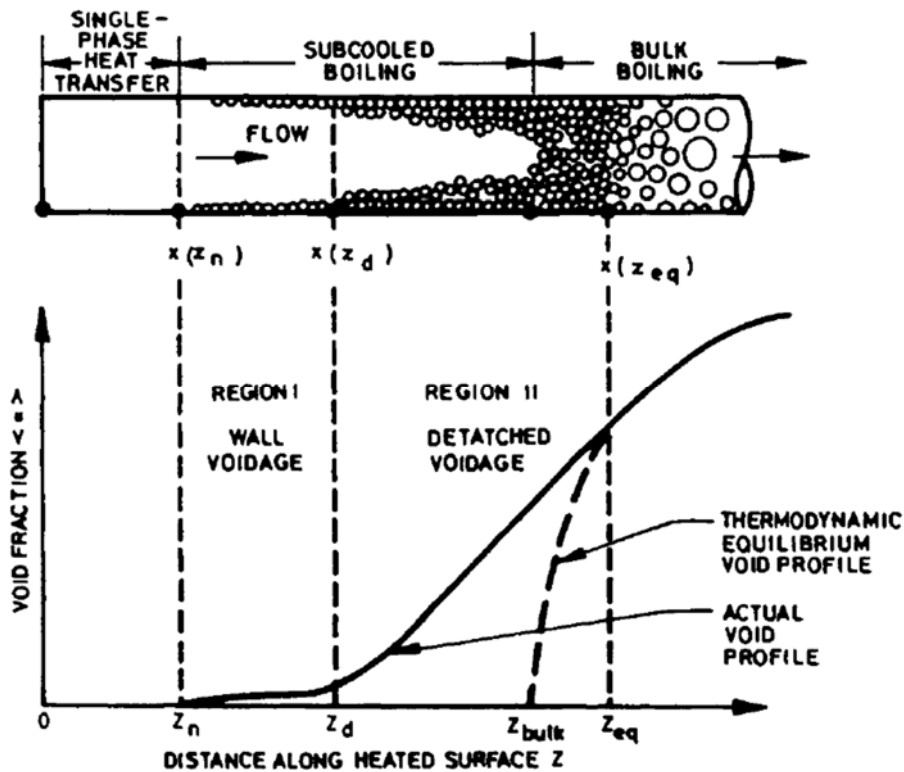


Figure 2.3 – Vapor volume fraction in subcooled boiling (Hewitt, 1978).

To calculate the actual quality, it is first necessary to determine the quality at point z_d , the point where vapor bubbles begin to detach from the channel surface. This can be calculated from the following correlations given by Saha and Zuber (1974):

$$x(z_d) = -0.0022 \frac{q'' D c_{p,l}}{i_{lv} k_l} \quad \text{for} \quad \frac{G D c_{p,l}}{k_l} < 70.000 \quad (2.8)$$

$$x(z_d) = -154 \frac{q''}{G i_{lv}} \quad \text{for} \quad \frac{G D c_{p,l}}{k_l} > 70.000 \quad (2.9)$$

With $x(z_d)$, it is possible to calculate the position z_d . For a circular channel of diameter D , one can rearrange Eq. (2.5) and z_d can be calculated as follows:

$$z_d = \frac{G D [i_{lv} x(z_d) + i_l - i_{ent}]}{4 q''} \quad (2.10)$$

From the determination of x given by Eq. (2.6) and from $x(z_d)$ by Eqs. (2.8) and (2.9), it is possible to calculate the actual quality with the correlation given by Levy (1966):

$$x_a = x - x(z_d) \exp \left[\frac{x}{x(z_d)} - 1 \right] \quad (2.11)$$

The vapor volume fraction at a given point in a channel can be calculated by knowing x_a . There are several correlations that relate volume fraction and quality in boiling convective flows but one of the most used is that of Zuber and Findlay (1965):

$$\alpha = \frac{x_a \rho_l}{C_o [x_a \rho_l + (1 - x_a) \rho_v] + \rho_l \rho_v u_\infty / G} \quad (2.12)$$

where C_o is a dimensionless parameter relative to the volume fraction distribution in the flow. For the bubbly flow region, Tong and Tang (1997) recommend a value C_o equal to 1.2. The velocity u_∞ is called the terminal velocity and, for the bubbly flow region, it is given by the equation of Lahey and Moody (1997):

$$u_\infty = 2,9 \left[\frac{(\rho_l - \rho_v) \sigma g}{\rho_l^2} \right]^{0,25} \quad (2.13)$$

The vapor volume fraction in the bubbly flow region corresponds to values of 0 % up to 30 %. Above this value, the flow enters the slug flow region (Rohsenow et al., 1998). The present work only deals with the bubbly flow region and, therefore, it is not addressed the other regions. However, it is important to mention that there are also specific methods and correlations to determine the quality and volume fraction of vapor in each of the different regions of an internal boiling flow.

Heat Transfer Coefficient

Several correlations have been developed over the last few decades with the intention of calculating the heat transfer coefficient in convective boiling flows, each with its particularities and limitations. Some of these use pool boiling correlations, such as Eq. (2.3), in combination with existing single-phase convection correlations, where an attempt is made to estimate the contribution portions of nucleated boiling and liquid phase convection to heat transfer. Another important approach is the use of empirical correlations, which are based on experimental data of forced convective boiling in internal flows. An important empirical correlation is that of Gungor and Winterton (1987), given by:

$$\frac{h}{h_l} = \left[1 + 3000Bo^{0,86} + 1,12 \left(\frac{x}{1-x} \right)^{0,75} \left(\frac{\rho_l}{\rho_g} \right)^{0,41} \right] \quad (2.14)$$

where Bo is a dimensionless called the boiling number and h_l is the heat transfer coefficient for the flowing of the liquid phase alone (W/m²K).

The boiling number Bo is defined by the following equation:

$$Bo = \frac{q''}{Gi_{lv}} \quad (2.15)$$

The heat transfer coefficient h_l can be calculated from the equation of Dittus and Boelter (1930):

$$Nu_l = \frac{h_l D}{k_l} = 0,023 Re_l^{0,8} Pr_l^n \quad (2.16)$$

where n is dimensionless and is equal to 0.4 for heating fluids, Re_l and Pr_l are the Reynolds number

and the Prandtl number, respectively, and in this case, they can be calculated by (Rohsenow et al., 1998):

$$Re_l = \frac{G(1-x)D}{\mu_l} \quad (2.17)$$

$$Pr_l = \frac{\mu_l c_{p,l}}{k_l} \quad (2.18)$$

Critical Heat Flux

As with in pool boiling, the phenomenon of critical heat flux also occurs in internal boiling flows and is characterized by an increase in the temperature of the channel walls for small incremental increases in the heat flux through the walls or, in cases where the wall temperature is controlled by the reduction of the heat flux for small incremental increases in temperature. This phenomenon is particularly important as it is a limiting condition for the operation of many equipment where boiling occurs (Hewitt, 1978).

Given the importance of the CHF, several authors have dedicated themselves to the development of correlations to determine the CHF point in boiling flows. One of the most used is that one proposed by Bowring (1972), which is valid for vertical and upward flow of water in pipes with a diameter between 2 and 45 mm, length between 150 and 3,700 mm, pressure between 0.2 and 19 MPa and mass fluxes between 136 and 18,600 kg/m².s. The CHF can be calculated by the following equation:

$$q''_{crit} = \frac{A' + 0.25DG\Delta i_{sub}}{C' + L} \quad (2.19)$$

where Δi_{sub} is the difference between the enthalpies of the saturated liquid and the liquid at the inlet of the channel (kJ/kg), L is the length of the tube (m). A' and C' are given by the following expressions:

$$A' = \frac{2.317(0.25i_{lv}DG)F_1}{(1+0.0143F_2D^{1/2}G)} \quad (2.20)$$

$$C' = \frac{0.077F_3DG}{[1+0.347F_4(G/1356)^n]} \quad (2.21)$$

where n is equal to:

$$n = 2,0 - 0,5P_R \quad (2.22)$$

and P_R equal to:

$$P_R = \frac{P}{6,895 \times 10^6} \quad (2.23)$$

being P the fluid pressure (Pa).

The parameters F_1 , F_2 , F_3 and F_4 depend on whether the value of P_R is less or greater than 1.0 and are given according to Table 2.1:

Table 2.1. Determination of the parameters F_1 , F_2 , F_3 and F_4 for the calculation of the critical heat flux (Bowring, 1972).

Parameter	$P_R < 1$	$P_R > 1$
F_1	$\frac{P_R^{18,492} \exp[20,8(1 - P_R)] + 0,917}{1,917}$	$P_R^{-0,368} \exp[0,648(1 - P_R)]$
F_1/F_2	$\frac{P_R^{1,316} \exp[2,444(1 - P_R)] + 0,309}{1,309}$	$P_R^{-0,448} \exp[0,245(1 - P_R)]$
F_3	$\frac{P_R^{17,023} \exp[16,658(1 - P_R)] + 0,667}{1,667}$	$P_R^{0,219}$
F_4/F_3	$P_R^{1,649}$	$P_R^{1,649}$

Wall Temperature

One way to calculate the channel wall temperature in boiling internal flow problems is through empirical correlations. Thom et al. (1965) proposed a correlation for water based on experimental results, which determines the wall superheat temperature and depends only on the heat flux and channel operating pressure:

$$\Delta T_e = 22.5 q''^{0.5} \exp(-P/87) \quad (2.24)$$

2.2. Coefficient of Performance

The Coefficient of Performance (COP) is a useful parameter that indicates the ratio of heating

(or cooling) provided to a system over the work required by this system. The higher the COP value of a given system is, the higher its efficiency. Also, high COP values lead to lower energy consumption. The COP can be summarized as:

$$COP = \frac{\text{useful heating (or refrigeration)}}{\text{net work}} \quad (2.25)$$

For a system consisted of the flow of a fluid inside a heated channel, the COP can be written as:

$$COP = \frac{q}{W} = \frac{q}{\Delta P \times \dot{m}} \quad (2.26)$$

where W is the net work rate done by the system (W), ΔP is the pressure drop along the channel (Pa) and $\dot{m}$ is the mass flow rate (kg/s).

2.3. Constructal Theory

Design is a fascinating topic and much explored in different areas of knowledge. It is present in biology, engineering, architecture, fashion and many other areas. In some of these areas, design is the result of a human creation and is something that can come from human emotions and feelings or the result of human intelligence and its work to solve problems. In much of the world around us, however, design is formed and established naturally. Whatever its origin, in many cases it is possible to identify that the design changes and evolves over time. A classic example of this evolution is from biology: Charles Darwin's theory of natural selection explains the evolution of species of living beings (and their design) based on the predominance and survival of the best adapted to their environments. In another area, engineering, it is easy to observe that certain designs of machines, equipment, vehicles, among others, evolve over the years, always with the intention of meeting some specific objective determined by humans. For example, the evolution of design in automobiles is noticeable over time. In the old days, until the 1970s and 1980s, cars with many straight lines and with a "square" appearance were common, while the cars launched today have much more curves and small details in their design because they are more aerodynamically efficient (as well as more attractive for consumers) and also because the manufacturing cost of producing more complex designs went down. From these examples, it is clear that systems design and its evolution are of great importance, but it is also common to think that there is no relationship between design in distant areas such as geology and economics or fluid dynamics and the evolution of species. The Constructal Theory, based on the

Constructal Law, seeks to explain that there is indeed a fundamental principle that can unify the formation and evolution of designs present in such different areas, in this case, the design of the so-called “flow systems”.

The Constructal Theory (Bejan and Lorente, 2008; Bejan and Zane, 2012; Bejan and Lorente, 2013; Bejan, 2020) was created by Adrian Bejan, a mechanical engineer with a PhD from MIT (Massachusetts Institute of Technology) born in 1948 in Romania (Lage et al., 2008). In the 1970s, Bejan began to work with research in the field of thermodynamics and heat transfer, having published several books and articles in the area.

The moment described as the discovery of the Constructal Law is remembered by Bejan as being at a conference on thermodynamics in France in 1995, when he heard a speech by a Nobel Prize winner who declared that tree-like structures found in nature (trees, river bases and deltas, lightning, circulatory system in living beings, among many others) had this shape randomly and that their similar design was purely coincidental, which was deemed a conventional thinking of the scientific community at the time (Bejan and Zane, 2012). In the second edition of his book *Advanced engineering Thermodynamics* (Bejan, 1997), Bejan introduced the world to Constructal Theory, postulating the famous Constructal Law:

“For a finite-sized fluid system to persist in time (to survive), its configuration needs to evolve freely in time in such a way that it facilitates access to the currents flowing through this system.”

Bejan established that the generation of configurations in flow systems, animate or inanimate, is a universal phenomenon of physics and that this phenomenon must occur according to the principles established by the Constructal Law. Of course, this statement has huge implications for our world. The existence of such a law allows us to say that there is a fundamental principle (a physical law) that governs, for example, from the formation of rivers and their tributaries to the way wealth is distributed in our society. Constructal Law, therefore, is about the evolution of flow systems design, not about the design itself. This is important because Constructal Theory should not be confused with a method of geometric optimization or of creation of geometries and systems.

The flow systems cited in the Constructal Law are literally everything that moves, whether animate or inanimate, physical or not (Bejan and Zane, 2012). They have two basic properties: the current that flows through this system and the design by which these currents flow. The definition of flow systems is independent of scale and includes a good part of the world around us and the clearest examples are the evolution of design in living beings over time, the formation of rivers and their tributaries, designs found in nature such as the of lightning, snowflakes, geological formations, the systems designs found in engineering, social systems such as the economy, the organization of

governments, and the meshes of streets and roads in a city (Bejan and Zane, 2012). Based on research in the most varied areas of knowledge, Constructal Theory has shown that the evolution of the design of all these flow systems follows a single law, the Constructal Law, always in the sense of facilitating the access of the currents that flow them.

A key point in Constructal Theory is the correct identification of the characteristics of a flow system: the geometry, its boundaries and constraints, the currents that flow through it and the resistances present to the flow of these currents. A typical flow system easily found around us are the streets and roads of a city. These can be any street that allows the transit of only pedestrians or roads designed to cars, trains, subways, bicycles, in short, any form that allows the movement of people or vehicles (the flowing currents). Logically, the movement of people in a city is not perfect because there are several difficulties that limit and hinder the perfect movement. Vehicle traffic, especially at certain times of the day, is a clear result of the inefficiency of a city's transit system. Some cities are able to deal with this problem much better than others and some factors, such as when a certain part of the city was planned and built also have a big influence on the flow of a city. Rome (Figure 2.4), for example, still has very narrow and cramped streets in the region of its historic center (inside the red square in Figure 2.4), which are better adapted to pedestrian traffic and small and slow vehicles, which causes great congestion of cars nowadays. The regions of the city further from the center were built much more recently and have wider streets and grid-shaped blocks (inside the blue square in Figure 2.4), which is much more efficient for the main means of transport used today: automobiles. This evolution was necessary for the city to adapt to the dynamics of the society that inhabited it, thus allowing the maintenance of its usefulness to people, as well as its own survival, and is a clear example of the existence of the Constructal Law in the society (Bejan et al. Zane, 2012).

In engineering, the Constructal Theory finds a clear application in problems of thermodynamics, heat transfer and fluid dynamics, although it is also applicable in several other areas. The nature of these problems, where a flow system, its currents and resistances are generally identifiable, make these problems ideal to be approached from the perspective of Constructal Theory. As seen in the bibliographic review section, the Constructal Theory has been applied in several works in these areas, such as in problems of conductive paths, convective flows, renewable energy devices, among others. Usually in these works, the Constructal Design methodology is applied, which is based on the definition of four steps:

- Identify the flow system and what is flowing;
- Degrees of freedom: the geometric variables of the system that have the freedom to vary within the established limits;

- Geometric constraints: the geometric variables of the system that are kept fixed;
- Identify the performance indicator which makes the flow system performs better, i.e., facilitates the flow: what is desired with the application of the method (for example: the reduction of the maximum temperature in excess in a problem of a heat conducting plate or the maximization of the heat transfer rate in a problem of an internal convective flow).

From the correct definition of these four steps, the Constructal Design can then be associated with a geometric optimization method such as exhaustive search or meta-heuristic methods such as genetic algorithms and simulated annealing in order to find the geometries that best meet the proposed objectives (Gonzales et al., 2015; Gonzales et al., 2021; Gonzales et al., 2023).



Figure 2.4 – Satellite view of the city of Rome, Italy (Google Earth, October 2021).

2.4. TOPSIS

In the present thesis, the geometric evaluation process was conducted in a multi-objective way, considering the thermal and fluid dynamic objectives. In this way, it is necessary to use an analysis methodology that allows the geometric evaluation to quantify the multi-objective performance of each geometry. In this thesis, the problem defines two conflicting objectives, which makes the application of a simple weighted average method inappropriate. Multiple-criteria decision analysis methods are methods designed specifically for problems of conflicting criteria in decision making and are used in many different areas such as military, finance, administration and science.

In this work, the TOPSIS (Technique for Order of Preference by Similarity to Ideal Solution) multi-criteria analytical decision method (Hwang and Yoon, 1981; Hwang et al., 1993) was employed for the multi-objective analysis. This method is based on the definition of criteria and on the concept that the alternative chosen in a multi-criteria problem with several alternatives must be the one with the smallest Euclidean distance from the ideal positive solution and the greatest distance from the ideal negative solution. Figure 2.5 graphically demonstrates an example of a multi-criteria problem where alternative A_1 has a smaller distance to the ideal positive solution A^* than A_2 , but A_1 also has a smaller distance than A_2 to the negative ideal solution, A^- , which makes it difficult to declare A_1 as the best alternative. The TOPSIS method eliminates this problem by considering both the distance to the ideal positive solution and the ideal negative solution.

The TOPSIS method evaluates a decision matrix containing m alternatives and n criteria (or attributes), like matrix D in the following example:

$$D = \begin{matrix} & x_1 & x_2 & \cdots & x_j & \cdots & x_n \\ \begin{matrix} Alt_1 \\ Alt_2 \\ \vdots \\ Alt_i \\ \vdots \\ Alt_m \end{matrix} & \begin{bmatrix} x_{11} & x_{12} & \cdots & x_{1j} & \cdots & x_{1n} \\ x_{21} & x_{22} & \cdots & x_{2j} & \cdots & x_{2n} \\ \vdots & \vdots & & \vdots & & \vdots \\ x_{i1} & x_{i2} & \cdots & x_{ij} & \cdots & x_{in} \\ \vdots & \vdots & & \vdots & & \vdots \\ x_{m1} & x_{m2} & \cdots & x_{mj} & \cdots & x_{mn} \end{bmatrix} \end{matrix}$$

where Alt_i is the i -th alternative to be considered and x_{ij} is the numerical value of the i -th alternative with respect to the j -th criterion.

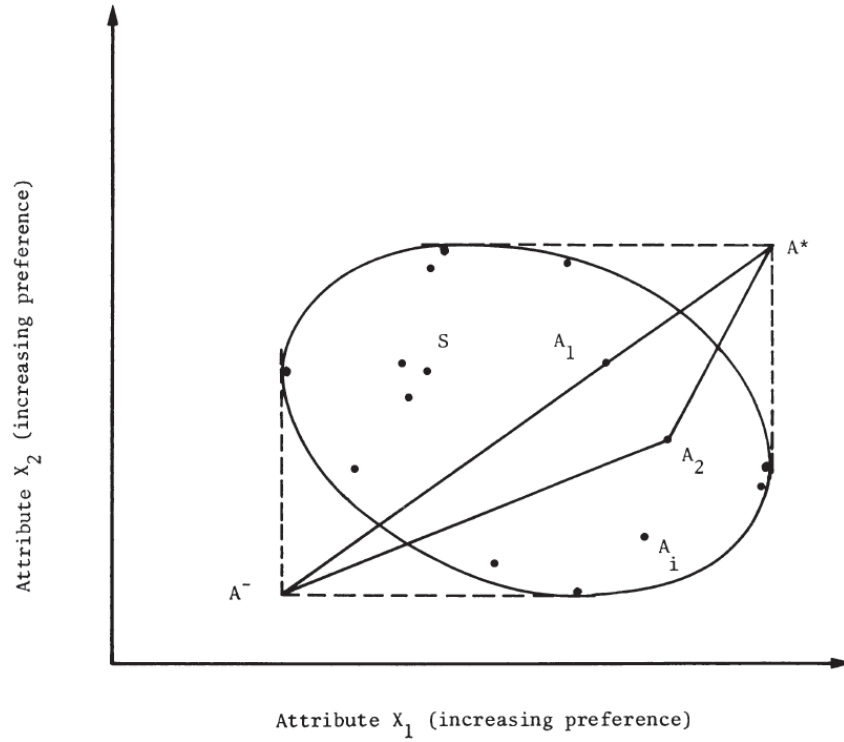


Figure 2.5 – Example of a multi-criteria problem (Hwang and Yoon, 1981).

The method considers that each attribute is increasing or decreasing as in a monotone function. The criteria are defined as “benefit” or “cost”, depending on their effect on the ideal solution. It is also allowed to define different weights for each criterion, according to their importance. After defining the decision matrix D with m alternatives and n criteria, the method is carried out following certain steps, which are:

1st step: Normalization of decision matrix D : Matrix D is normalized using the following formula for each element of the matrix:

$$r_{ij} = \frac{x_{ij}}{\sqrt{\sum_{i=1}^m x_{ij}^2}} \quad (2.27)$$

2nd step: Construction of the weighted normalized decision matrix: Each element of the matrix r_{ij} is multiplied by a weight w_j corresponding to a criterion j , generating a matrix V :

$$v_{ij} = r_{ij} \cdot w_j \quad (2.28)$$

where $\sum_{j=1}^n w_j = 1$.

3rd step: Determination of the positive and negative ideal solutions: The positive ideal solution is:

$$A^* = \{v_1^*, \dots, v_n^*\} = \{(max_i v_{ij} | j \in J), (min_i v_{ij} | j \in J')\} \quad (2.29)$$

and the negative ideal solution is given by:

$$A^- = \{v_1^-, \dots, v_n^-\} = \{(min_i v_{ij} | j \in J), (max_i v_{ij} | j \in J')\} \quad (2.30)$$

where $J = \{j = 1, 2, \dots, n | j\}$ is associated with the benefit criterion and $J' = \{j = 1, 2, \dots, n | j\}$ is associated with the cost criterion.

4th step: Calculate the distances between each alternative and the ideal solutions: The distance from an alternative i to the positive ideal solution is:

$$D_i^* = \sqrt{\sum_{j=1}^m (v_{ij} - v_j^*)^2}, \quad i = 1, \dots, m \quad (2.31)$$

and the distance to the negative ideal solution is:

$$D_i^- = \sqrt{\sum_{j=1}^m (v_{ij} - v_j^-)^2}, \quad i = 1, \dots, m \quad (2.32)$$

Step 5: Calculate the relative proximity between each alternative and the ideal solution:

$$C_{bi} = \frac{D_i^-}{D_i^- + D_i^*}, \quad i = 1, \dots, m \quad (2.33)$$

where b is related to a specific weight combination set.

6th step: Order the alternatives according to the values obtained from C_{bi} . The closer to 1 the value of C_i , the higher the alternative should be ranked.

the heat transfer and fluid dynamics were investigated and also is based on the limitations of the numerical model. After several tests performed, it was observed that the numerical model was not able to simulate, for the proposed conditions, smaller distances between the blocks, higher heights for blocks as well as the addition of more blocks to the channel other than those proposed for the channel's geometry in this section.

In this thesis, a geometric evaluation of the channel is conducted with the aim of evaluating the influence of the geometry over the performance indicators. Thus, three methods are used with the intention of assisting in this process. The first one is the Constructal Design, which is used to define the search space geometric possibilities (degrees of freedom and constraints) and the objectives (performance indicators) of the problem. For the geometric optimization, the Exhaustive Search method is used, with the definition of a certain step of variation between the possible geometries of the search space. The third method applied is the TOPSIS, which is applied in the multi-objective analysis of the results obtained, in order to determine the best multi-objective geometry. Table 3.1 presents the dimensions of the channel that are kept fixed for all cases and Table 3.2 shows the conditions applied to the problem. The Reynolds number in Table 3.2 is calculated based in the thermophysical properties of the liquid phase at the inlet of the channel:

Table 3.1 – Constant dimensions of the geometry of the channel

L (mm)	250.00
D (mm)	20.00
L_{in} (mm)	100.00
L_{out} (mm)	54.00
L_{ocu} (mm)	96.00

Table 3.2 – Conditions applied to the studied cases

$Re_D = \rho V_{in} D / \mu$	~ 92.634
V_{in} (m/s)	0.65
T_{in} (K)	510
T_w (K)	536
T_{sat} (K)	530.59
P_{out} (Pa)	0
P_{op} (MPa)	4.5

Table 3.3 presents the thermofluidic properties of the liquid and vapor phases at the saturation temperature (530.59 K).

Table 3.3 - Thermofluidic properties of the phases at the saturation temperature.

Properties	Water (liquid)	Water (vapor)
Density (kg/m ³)	787.61	22.6967
Dynamic Viscosity (Pa.s)	1.0278×10^{-4}	1.7778×10^{-5}
Specific Heat (kJ/kg.K)	4.9492	4.2279
Thermal Conductivity (W/m.K)	0.6122	0.05330
Specific Enthalpy (kJ/kg)	1,121.404	2,797.997
Boiling Temperature (K)	530.5894	

One of the objectives of this study is to analyze the influence of boiling over the geometric evaluation process. For this, similar cases are simulated both with and without the occurrence of boiling in the turbulent flow regime. In this way, two similar geometric evaluation processes are performed for a direct comparison.

3.2. Constructal Design

The Constructal Design method is applied for the geometric evaluation of the channel. Firstly, the geometrical constraints are defined. In the present thesis, there are three constraints: the channel area (A), the area of occupation of each block (A_1 , the same for both blocks) and the area of the blocks (A_2 , also the same for both blocks):

$$A = D \cdot L \quad (3.1)$$

$$A_1 = D \cdot (L_{c1} + L^3/2) = D \cdot (L_{c2} + L^3/2) \quad (3.2)$$

$$A_2 = H_1 \cdot L_1 = H_2 \cdot L_2 \quad (3.3)$$

The ratio between the area of each block over their area of occupation (A_2/A_1) is called ϕ_1 and ϕ_2 , and, in this study, these ratios are considered constant and equal to 0.2, that is, each block occupies 20% of its occupation area. The objectives of the geometric evaluation are to maximize the heat transfer rate (q) from the heated region of the channel (colored in red in Figure 3.1) to the

surrounding flow and to minimize the pressure difference (ΔP) between the inlet and outlet of the channel, analyzing the various geometry arrangements of the two blocks. For this purpose, two degrees of freedom are defined: the ratio between the height and the length of each block, H_1/L_1 and H_2/L_2 . Also, the effect of the ratio between the distance of center-to-center of the blocks and the length of occupation, L_3/L_{ocu} , over the objectives will be investigated. The heat transfer rate can be calculated as (Chen, 1966):

$$q = A_s h_l (T_w - T_{in}) + A_s h_{NB} (T_w - T_{sat}) \quad (3.4)$$

where A_s is the surface area of the channel (m^2), h_l is the heat transfer coefficient for the liquid phase flowing alone in the channel (W/m^2K) and h_{NB} is the nucleate boiling heat transfer coefficient (W/m^2K).

The pressure drop along the channel is the difference between the pressures of the inlet and the outlet of the channel:

$$\Delta P = P_{in} - P_{out} \quad (3.5)$$

where P_{in} and P_{out} are the averaged pressures in the inlet and outlet surfaces of the channel (Pa), respectively.

Figure 3.2 illustrates the schematic representation of the methodology applied to the geometric evaluation and optimization of the present study.

3.3. Geometrical Evaluation

The Exhaustive Search method is used for the geometric optimization process, according to the three constraints and the two degrees of freedom defined by the Constructal Design method. The geometries of the two blocks are varied by changes of the ratios H_1/L_1 and H_2/L_2 in the range between 0.0625 and 1.55 (with 12 different values for each ratio) while L_3/L_{ocu} is analyzed for three different magnitudes (0.375, 0.5 and 0.625). Thus, in each of the two cases (with and without boiling) 432 simulations are performed, with 864 simulations in total.

It is important to mention that the choice of the maximum value of 1.55 for H_1/L_1 and for H_2/L_2 , where the blocks have a height slightly greater than half the diameter of the channel. This restriction is imposed due to observations carried out in other flow studies that found that a block height very close to the channel diameter results in a drop not only in fluid dynamic performance, but also in heat transfer (Prajapati, 2019).

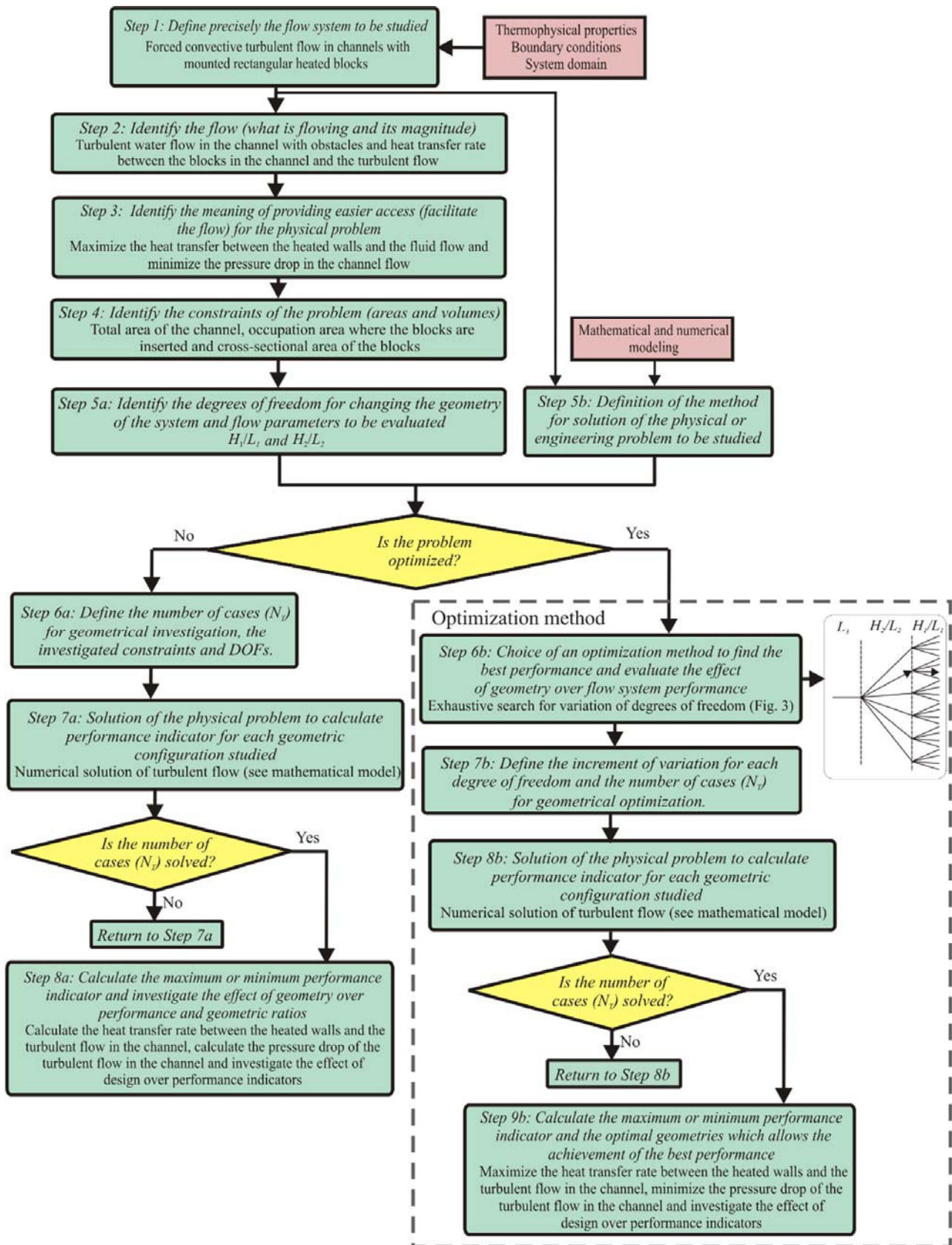


Figure 3.2 – Schematic representation of the methodology applied to the geometric evaluation and optimization of the present study.

In a work carried out in laminar flows with boiling in microchannels with trapezoidal blocks, Feijó et al. (2022) found that a block height close to the value of the microchannel diameter generates a stricture effect, significantly reducing the overall performance of the system. Also, Abdous et al. (2016) found that smaller fin heights in microchannels with boiling flows led to better performance in the cases studied by them. For these reasons, it was preferred not to explore an upper limit of H_1/L_1 and H_2/L_2 in which the blocks would have a height closer to the diameter of the channel.

The geometric evaluation process carried out in this work follow the principles of Constructal Design, already used in past works (Feijó et al., 2018). In this process, the thermal and fluid dynamic objectives are initially analyzed separately, but the steps for their realization are the same, one for each degree of freedom:

1st step: H_2/L_2 and L_3/L_{ocu} are kept constant at a given value and H_1/L_1 is simulated within its search space (considering the 12 different magnitudes). Among the results obtained in this step, the maximum value for the heat transfer rate is called the once maximized rate $(q)_{1,max}$, and, similarly, the minimum pressure drop value is called the pressure drop once minimized, $(\Delta P)_{1,min}$. The geometries corresponding to these values are called the once thermally optimized ratio of H_1/L_1 , $(H_1/L_1)_{o,T}$, and once fluid dynamically optimized ratio of H_1/L_1 , $(H_1/L_1)_{o,F}$.

2nd step: Now the first step is repeated, but for the other values of H_2/L_2 . Again, among all the results obtained, a maximum value for the heat transfer rate is now the twice maximized heat transfer rate $(q)_{2,max}$, and a minimum value for the pressure drop is the twice minimized pressure drop, $(\Delta P)_{2,min}$. The corresponding geometries for each performance indicators are $(H_1/L_1)_{oo,T}$ (now twice optimized) and $(H_2/L_2)_{o,T}$ (once optimized) and also $(H_1/L_1)_{oo,F}$ and $(H_2/L_2)_{o,F}$.

3rd step: The two previous steps are now repeated for the different L_3/L_{ocu} values. Thus, among all the results, a three times maximized heat transfer rate $(q)_{3,max}$ and a three times minimized pressure drop, $(\Delta P)_{3,min}$, are obtained. These correspond to the optimal thermal geometry $(H_1/L_1)_{ooo,T}$, $(H_2/L_2)_{oo,T}$ and $(L_3/L_{ocu})_{o,T}$ and the optimal fluid dynamics geometry $(H_1/L_1)_{ooo,F}$, $(H_2/L_2)_{oo,F}$ and $(L_3/L_{ocu})_{o,F}$.

Figure 3.3 demonstrates the representation of the steps of the geometric evaluation with Constructal Design.

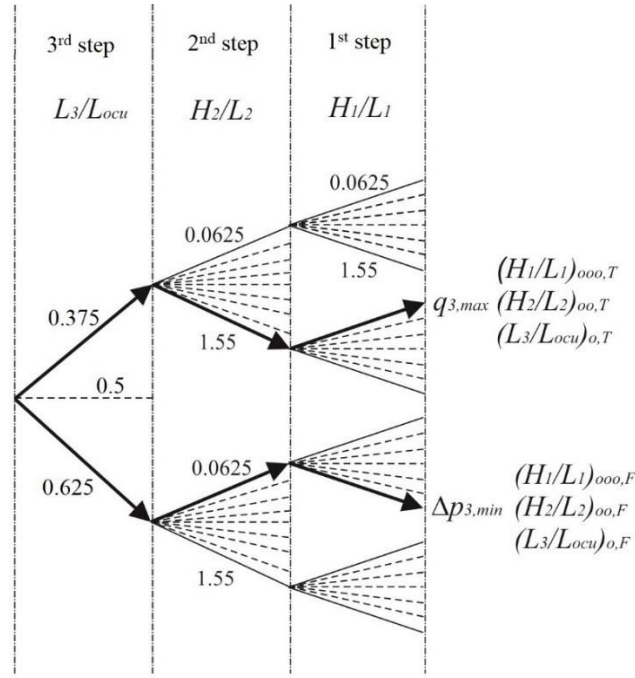


Figure 3.3 – Geometrical evaluation process applied to the channel.

Subsequently, a geometric evaluation is carried out taking into account the proposed multi-objective (thermal and fluid dynamics, considered simultaneously), similarly to the three steps described, in order to determine the optimal multi-objective geometry: $(H_1/L_1)_{ooo}$, $(H_2/L_2)_{oo}$ and $(L_3/L_{ocu})_o$. In this step, the TOPSIS method is used, where the possible geometries are considered as the alternatives to be chosen and the two objectives are considered as the choice criteria, with the thermal objective being a positive impact criterion and the fluid dynamic objective a negative impact criterion. In the present study, both criteria are considered to have identical weight over the ideal solution. Likewise, a similar geometric evaluation is performed afterwards considering the Coefficient of Performance (COP) as the objective of the geometric optimization.

3.4. The Eulerian Multiphase Model

The mathematical modeling of the flow is based on the Eulerian multiphase model (Drew, 1983; Ishii and Mishima, 1984). The Eulerian model solves the conservation equations for each of the phases present and considers the concept of volume fraction of the phases, so that the multiphase flow is a continuous and interpenetrating medium, unlike the Volume of Fluid model (VOF), widely used in multiphase flow simulations, which reproduces the exact boundary between the phases present in the flow. Because of this simplification, the use of Eulerian model brings a significant advantage in processing time in numerical simulations in relation to the VOF model (Guerrero et al., 2017). Figure 3.4 illustrates the main difference between the VOF and the Eulerian models, where

the volume vapor fraction fields for the same flow, subjected to different heat fluxes, are illustrated for comparison. Table 3.4 presents a comparison between the main characteristics of the VOF and Eulerian models based on the authors Guerrero et al. (2017) and ANSYS (2017a).

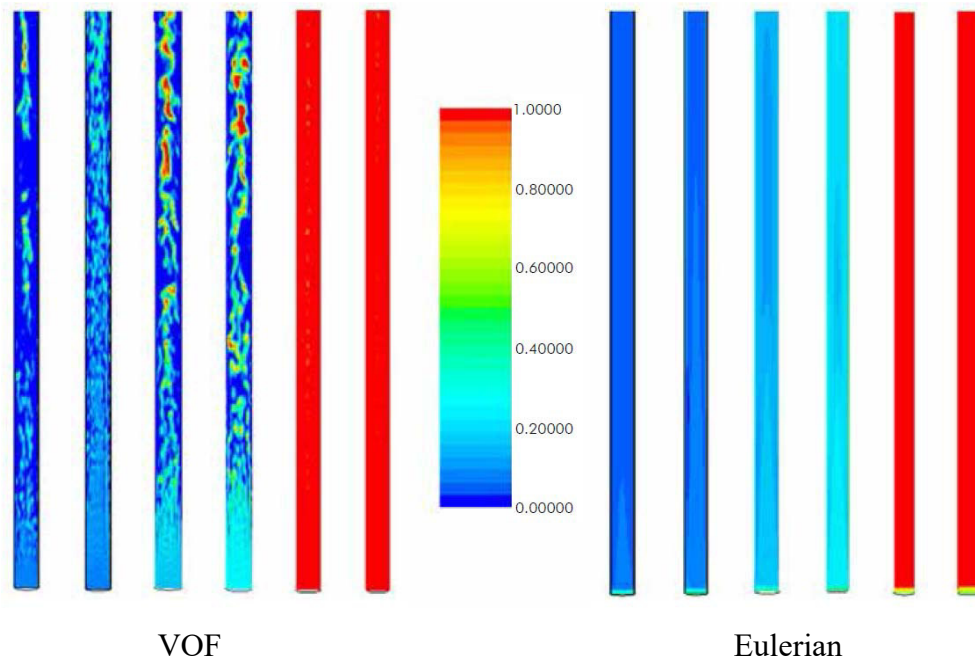


Figure 3.4 – Comparison of phase volume fraction fields between the VOF and Eulerian models for the same multiphase flow subjected to different heat fluxes. Adapted from Guerrero et al. (2017).

Table 3.4 – Comparison between the Volume of Fluid and Eulerian models.

Volume of Fluid	Eulerian
Tracks the interface of the phases: It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest.	Doesn't track the interface of the phases but has less computational cost and is more precise for bubbly flows than the VOF.
A single set of momentum equations is shared by the fluids.	It solves a set of momentum and continuity equations for each phase; Coupling is achieved through the pressure and interphase exchange coefficients.
Applications: stratified flows, free-surface flows, filling, sloshing, the motion of large bubbles in a liquid, the motion of liquid after a dam break, the prediction of jet breakup (surface tension), and the steady or transient tracking of any liquid-gas interface.	Applications: bubble columns, bubbly flows, risers, particle suspension, and fluidized beds.

For a given volume of the flow, the sum of the volume fractions of all phases must be equal

to 1. Thus, for a fluid with n phases:

$$\sum_{q=1}^n \alpha_q = 1 \quad (3.6)$$

where α_q is the volume fraction of phase q .

The equations of conservation of mass, momentum and energy of the Eulerian model are given by Ishii and Mishima (1984). For a given phase q , the mass conservation equation can be written as:

$$\frac{\partial}{\partial t} (\overline{\alpha_q \rho_q}) + \nabla \cdot (\overline{\alpha_q \rho_q \vec{v}_q}) = \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) \quad (3.7)$$

where $\vec{v}_q$ is the phase q velocity (m/s), $\dot{m}_{pq}$ and $\dot{m}_{qp}$ are, respectively, the mass transfers from phases p to q and from q to p (kg/s).

The momentum conservation equation for a phase q is given by:

$$\begin{aligned} \frac{\partial}{\partial t} (\overline{\alpha_q \rho_q \vec{v}_q}) + \nabla \cdot (\overline{\alpha_q \rho_q \vec{v}_q \vec{v}_q}) = & -\overline{\alpha_q} \nabla \overline{P} + \nabla \cdot \overline{\vec{\tau}_q} + \overline{\alpha_q \rho_q} \vec{g} + \sum_{p=1}^n (\psi_{pq} (\overline{\vec{v}_p} - \overline{\vec{v}_q}) + \dot{m}_{pq} \overline{\vec{v}_{pq}} - \\ & \dot{m}_{qp} \overline{\vec{v}_{qp}}) + \vec{F}_q + \vec{F}_{lift,q} + \vec{F}_{vm,q} + \vec{F}_{td,q} \end{aligned} \quad (3.8)$$

where $\overline{\vec{\tau}_q}$ is the phase q stress-strain tensor (N/m³), ψ_{pq} is the interphase momentum exchange coefficient, $\vec{F}_q$ is an external force (N), $\vec{F}_{lift,q}$ is the lift force (N), $\vec{F}_{vm,q}$ is the virtual mass force (N) and $\vec{F}_{td,q}$ is the turbulent dispersion force (N). $\vec{v}_{pq}$ and $\vec{v}_{qp}$ are the interphase speeds (m/s) where when $\dot{m}_{pq} > 0$, $\vec{v}_{pq}$ is equal to $\vec{v}_p$ and when $\dot{m}_{pq} < 0$, $\vec{v}_{pq}$ is equal to $\vec{v}_q$. Likewise, when $\dot{m}_{qp} > 0$, $\vec{v}_{qp}$ is equal to $\vec{v}_q$ and when $\dot{m}_{qp} < 0$, $\vec{v}_{qp}$ is equal to $\vec{v}_p$.

The stress-strain tensor from phase q is given by (Lopez de Bertodano et al., 1994):

$$\overline{\vec{\tau}_q} = \overline{\alpha_q} \mu_{t,q} (\nabla \overline{\vec{v}_q} + \nabla \overline{\vec{v}_q}^T) + \overline{\alpha_q} \left(\zeta_q - \frac{2}{3} \mu_{t,q} \right) \nabla \cdot \overline{\vec{v}_q} \overline{\vec{I}} \quad (3.9)$$

where $\mu_{t,q}$ is the turbulent dynamic viscosity of the phase q (N.s /m²), ζ_q is the bulk viscosity of the phase q (N.s/m²) and $\overline{\vec{I}}$ is the identity tensor.

The energy conservation equation for a phase q can be expressed as:

$$\frac{\partial}{\partial t} (\bar{\alpha}_q \rho_q \bar{i}_q) + \nabla \cdot (\bar{\alpha}_q \rho_q \bar{\vec{v}}_q \bar{i}_q) = \bar{\alpha}_q \frac{d\bar{P}_q}{dt} + \bar{\vec{q}}_q \cdot \nabla \bar{\vec{v}}_q - (k + k^T) \nabla \cdot \bar{\vec{T}} + \sum_{p=1}^n (Q_{pq} + \dot{m}_{pq} i_{pq} - \dot{m}_{qp} i_{qp}) \quad (3.10)$$

where i_q is the phase q enthalpy (J/kg), P_q is the phase q pressure (Pa), $\vec{q}_q$ is the phase q heat flux (W/m²), Q_{pq} is the volumetric rate of convection heat transfer between the phases p and q (W/m³), i_{pq} and i_{qp} are the relative enthalpies between the phases p and q (J/kg).

Interphase Exchange Coefficient

The interphase momentum exchange coefficient plays an important role in the interaction between the phases of a flow and, as already seen, it is present in the momentum conservation equation, Eq. (3.8). In a mixture of two fluids, the secondary phase, p in this case, can be considered to be in the form of bubbles. The exchange coefficient can be determined by the equation:

$$\psi_{pq} = \frac{\rho_p f_{drag}}{6\tau_p} d_p A_i \quad (3.11)$$

where f_{drag} is the default drag function, A_i is the concentration of the interfacial area (m²/m³), τ_p is the particle relaxation time (s) and d_p is the phase p bubble diameter (m).

The concentration of the interfacial area can be obtained by:

$$A_i = \frac{6\alpha_p}{d_p} \quad (3.12)$$

This equation is derived from the volume fraction of phase p , α_p , multiplied by the ratio between the surface area and the volume of a circular particle with diameter d_p , called A_p :

$$A_p = \frac{\pi d_p^2}{\frac{1}{6}\pi d_p^3} = \frac{6}{d_p} \quad (3.13)$$

In Eq. (3.11), particle relaxation time can be calculated as follows:

$$\tau_p = \frac{\rho_p d_p^2}{18\mu_q} \quad (3.14)$$

In the present study, the Eulerian model uses the Ishii model for the drag function (Ishii and Kim, 2001). The Ishii drag function can be calculated from the equation:

$$f_{drag} = \frac{C_D Re_{rel}}{24} \quad (3.15)$$

where C_D is the drag coefficient and Re_{rel} is the relative Reynolds number.

The relative Reynolds number is calculated based on the relative velocity, $\vec{V}_{pq}$ (Manninen *et al.*, 1996):

$$Re_{rel} = \frac{d_p \rho_q |\vec{V}_{pq}|}{\mu_q} \quad (3.16)$$

In the model proposed by Ishii, the drag coefficient can be obtained from the minimum value between the coefficient of the viscous regime (C_D^{vis}) and that of the distorted regime (C_D^{dis}):

$$C_D = \min(C_D^{vis}, C_D^{dis}) \quad (3.17)$$

where C_D^{vis} and C_D^{dis} can be calculated as:

$$C_D^{vis} = \frac{24}{Re_{rel}} (1 + 0.15 Re_{rel}^{0.75}) \quad (3.18)$$

$$C_D^{dis} = \frac{2}{3} \frac{d_p}{\sqrt{\frac{\sigma}{g|\rho_q - \rho_p|}}} \quad (3.19)$$

Lift Force

The lift force, present in the momentum conservation equation, Eq. (3.8), acts on the particles of the secondary phase (dispersed phase) mainly due to the velocity gradient of the primary phase. For a primary phase q and a secondary phase p , the lift force can be calculated as (Drew and Lahey, 1993):

$$\vec{F}_{lift,q} = -C_l \rho_q \alpha_p (\vec{v}_q - \vec{v}_p) \times (\nabla \times \vec{v}_q) \quad (3.20)$$

where C_l is the lift coefficient.

The present study applied the model developed by Moraga et al. (1999) to determine the lift coefficient, which combines the action of lift forces resulting from the interaction between the phase particles and those induced by turbulence vortices. Thus, it is necessary to calculate both the relative Reynolds number and the vorticity Reynolds number. The vorticity Reynolds number can be given by:

$$Re_{\nabla} = \frac{\rho_q |\nabla \times \vec{v}_q| d_p^2}{\mu_q} \quad (3.21)$$

In this way, the Moraga lift coefficient can be obtained from the following definition:

$$C_l = \begin{cases} 0.0767 & \varphi \leq 6000 \\ -\left(0.12 - 0.2e^{-\frac{\varphi}{3.6} \times 10^{-5}}\right) e^{\frac{\varphi}{3} \times 10^{-7}} & 6000 < \varphi < 5 \times 10^7 \\ -0.6353 & \varphi \geq 5 \times 10^7 \end{cases} \quad (3.22)$$

where $\varphi = Re_{rel} Re_{\nabla}$.

Virtual Mass Force

To represent the effect of the acceleration of the secondary phase particles in relation to the primary phase, the effect of virtual mass force is included. This force can be calculated by the formula (Drew and Lahey, 1993):

$$\vec{F}_{vm,q} = C_{vm} \alpha_p \rho_q \left(\frac{d_q \vec{v}_q}{dt} - \frac{d_p \vec{v}_p}{dt} \right) \quad (3.23)$$

where C_{vm} is the virtual mass coefficient and in this study is considered constant and equal to 0.5.

Turbulent Dispersion Force

The turbulent dispersion force, also present in the momentum conservation equation, is calculated in the present study as proposed by Lopez de Bertodano (1991):

$$\vec{F}_{td,q} = -\vec{F}_{td,p} = C_{td} \rho_q k_{t,q} \nabla \alpha_p \quad (3.24)$$

where $k_{t,q}$ is the turbulent kinetic energy of the phase q (m^2/s^2) and C_{td} is a constant and considered

equal to 1.0.

Convective Heat Transfer Between Phases

From the energy conservation equation, Eq. (3.10), it is necessary to determine the volumetric rate of convection heat transfer between the phases (Q_{pq}). This is a function of the temperature difference between the phases p and q and the concentration of the interfacial area A_i , determined by the Eq. (3.12). Q_{pq} can be calculated from the following formula:

$$Q_{pq} = h_{pq} A_i (T_p - T_q) \quad (3.25)$$

where h_{pq} is the convection heat transfer coefficient between the phases p e q (W/m².K), which can be determined by:

$$h_{pq} = \frac{k_q Nu_p}{d_p} \quad (3.26)$$

where Nu_p is the Nusselt number of the secondary phase p .

To calculate the Nusselt number of the phase p , the present study used the Ranz and Marshall (1952) correlation, given by:

$$Nu_p = 2.0 + 0.6 Re_{rel}^{1/2} Pr_q^{1/3} \quad (3.27)$$

where Pr_q is the Prandtl number of the primary phase q , defined by:

$$Pr_q = \frac{c_{p,q} \mu_q}{k_q} \quad (3.28)$$

3.5. The RPI Boiling Model

The RPI boiling model (Kurul and Podowski, 1991) was employed in this study in the multiphase cases with phase change. This model was developed in the early 1990s in the *Rensselaer Polytechnic Institute* and has become one of the most used in the numerical solution of internal boiling flow problems. According to this model, the total heat flux from the walls of a channel to a flow can be divided into three distinct parts: liquid phase convective heat flux ($\dot{q}_C$), quenching heat flux ($\dot{q}_Q$) and evaporative heat flux ($\dot{q}_E$):

$$\dot{q}_W = \dot{q}_C + \dot{q}_Q + \dot{q}_E \quad (3.29)$$

The RPI model assumes that the surface area of the channel walls can be divided into a portion called the area of influence (A_b), covered by the forming vapor bubbles, and another portion equal to $(1 - A_b)$, occupied only by the liquid phase. From this, it is possible to define the equations of the three heat fluxes. The convective heat flux can be written as follows:

$$\dot{q}_C = h_l(T_w - T_l)(1 - A_b) \quad (3.30)$$

where T_l is the temperature of the liquid (K).

The quenching heat flux represents the average energy transfer that occurs cyclically due to the presence of liquid in the vicinity of the channel wall, in a region after the formation and detachment of bubbles, with a period of t , and can be expressed as the follow:

$$\dot{q}_Q = C_{wt} \frac{2k_l}{\sqrt{\pi\lambda_l t}} (T_w - T_l) \quad (3.31)$$

where t is the time (s), λ_l is the diffusivity of the liquid (m^2/s) and C_{wt} is the bubble waiting time coefficient, introduced to correct the waiting time between the departure of successive vapor bubbles.

The evaporative heat flux is calculated by the following equation:

$$\dot{q}_E = V_d N_w \rho_v i_{lv} f \quad (3.32)$$

where V_d is the volume of the bubbles with reference to the departure diameter (m^3), N_w is the nucleate site density area and f is the frequency of the vapor bubbles departure (1/s).

Area of Influence

The area A_b , present in the convective heat flux equation, Eq. (3.30), is known as the area of influence and can be defined as:

$$A_b = \min \left(1, K_i \frac{N_w \pi D_w^2}{4} \right) \quad (3.33)$$

where K_i is an empirical coefficient and D_w is the bubble departure diameter (m).

In the present study, the empirical coefficient K_i is determined by the correlation proposed by Del Valle and Kenning (1985):

$$K_i = 4.8e^{(-Ja_{sub}/80)} \quad (3.34)$$

where Ja_{sub} is the subcooled Jacob number, defined as:

$$Ja_{sub} = \frac{\rho_l c_{p_l} \Delta T_{sub}}{\rho_v l_{lv}} \quad (3.35)$$

where ΔT_{sub} is the degree of subcooling (K), given by:

$$\Delta T_{sub} = T_{sat} - T_l \quad (3.36)$$

Nucleate Site Density

The nucleate site density (N_w) is defined based on the correlation proposed by Lemmert and Chawla (1977):

$$N_w = C^n (T_w - T_{sat})^{n_{nsd}} \quad (3.37)$$

where C and n_{nsd} are empirical parameters given by $C = 210$ and $n_{nsd} = 1.805$.

Frequency of Bubble Departure

The frequency of separation of the vapor bubbles (f) is defined by the correlation of Cole (1960):

$$f = \sqrt{\frac{4g(\rho_l - \rho_v)}{3\rho_l D_w}} \quad (3.38)$$

Bubble Departure Diameter

The bubble departure diameter is determined by the empirical correlation of Tolubinski and Kostanchuk (1970):

$$D_w = \min\left(0.0014, 0.0006e^{-\frac{\Delta T_{sub}}{45}}\right) \quad (3.39)$$

3.6. Turbulence Modeling – The $\kappa - \omega$ SST model

The present study applied the $\kappa - \omega$ SST model in turbulence modeling (Menter, 1994; Menter et al., 2003; Menter, 2009), both in single-phase and multiphase cases where turbulence was applied. This model is a refinement of the $\kappa - \omega$ model (Wilcox, 2006) and differs in modifying the turbulent viscosity formulation, taking into account the transport effects of shear stress turbulence. The $k - \omega$ SST model is a hybrid model between the $k - \omega$ model, which represents adequately the turbulent flow in the near wall region, and the $k - \varepsilon$ model, that represents the turbulent flow in a more properly form in isotropic regions of the flow (far from the wall).

In the $\kappa - \omega$ SST model, the turbulent viscosity is given by:

$$\mu_t = \frac{\rho a_1 k_t}{\max(a_1 \omega, I_{sr} F_2)} \quad (3.40)$$

where a_1 is a constant equal to 0.31, ω is the specific turbulence dissipation rate (1/s) and I_{sr} is the invariant measure of the strain rate (1/s). F_2 is given by:

$$F_2 = \tanh\left[\left[\max\left(\frac{2\sqrt{k_t}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega}\right)\right]^2\right] \quad (3.41)$$

where β^* is a constant equal to 0.09 and y is the distance to the nearest wall (m).

The turbulent diffusivity can be calculated by:

$$\lambda_t = \frac{\mu_t}{Pr_t} \quad (3.42)$$

where Pr_t is the turbulent Prandtl number, given by:

$$Pr_t = \frac{c_p \mu_t}{k_t} \quad (3.43)$$

From Eq. (3.40), one can define the equations for the turbulent kinetic energy (k_t) and for the specific turbulence dissipation rate (ω):

$$\frac{\partial(\rho k_t)}{\partial t} + \frac{\partial(\rho u_i k_t)}{\partial x_i} = \tilde{P}_k - \beta^* \rho k_t \omega + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_k \mu_t) \frac{\partial k_t}{\partial x_i} \right] \quad (3.44)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_i \omega)}{\partial x_i} = \frac{1}{a} \tilde{P}_k - \beta \rho \omega^2 + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] + 2(1 - F_1) \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k_t}{\partial x_i} \frac{\partial \omega}{\partial x_i} \quad (3.45)$$

where u_i is the flow velocity (m/s) in the direction i . a , β , σ_k and σ_ω are constants determined by the equation of the type:

$$\Phi = \Phi_1 F_1 + \Phi_2 (1 - F_1) \quad (3.46)$$

where Φ represents the constants a , β , σ_k and σ_ω . The model constants are $a_1 = 0.31$, $\beta_1 = 0.075$, $\sigma_{k1} = 1.176$, $\sigma_{\omega 1} = 2.0$, $a_2 = 0$, $\beta_2 = 0.0828$, $\sigma_{k2} = 1.0$ and $\sigma_{\omega 2} = 1.168$.

$\tilde{P}_k$, present in Eqs. (3.44) and (3.45), is defined by:

$$\tilde{P}_k = \min(P_k, 10 \cdot \beta^* \rho k_t \omega) \quad (3.47)$$

being P_k given by:

$$P_k = \mu_t \frac{\partial u_i}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3.46)$$

F_1 is determined according to the following equation:

$$F_1 = \tanh \left\{ \left\{ \min \left[\max \left(\frac{\sqrt{k_t}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\rho \sigma_{\omega 2} k_t}{CD_{k\omega} y^2} \right] \right\}^4 \right\} \quad (3.47)$$

where $CD_{k\omega}$ is given by:

$$CD_{k\omega} = \max \left(2\rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k_t}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right) \quad (3.48)$$

4. NUMERICAL MODELING

The present thesis carried out a numerical study of a computational fluid dynamics (CFD) problem where several cases were simulated. CFD is a powerful engineering tool capable of simulating fluid dynamics problems and situations in a virtual environment reproducing all its main characteristics and in a fully customizable way by the user, through methods and mathematical models inserted into computer codes. It began to be disseminated around the 1970s and 1980s with the evolution of computer processing power, which made it possible to address increasingly complex problems. This evolution of computers continues to influence CFD to this day and will likely continue into the future (Maliska, 2004; Versteeg and Malalasekera, 2007; Moukalled et al., 2016)

One of the great advantages provided by CFD is the practicality when compared to real experiments. CFD allows performing a large number of simulations of a given problem with a relative low cost, which is mainly the time required to process the simulations. This is a big advantage from a financial point of view, in addition to the physical effort required to conduct experiments. Other advantages are the ease of recording and storing data, the ability to reproduce systems that are very complex, difficult or even dangerous to be experimentally conducted, and the high level of detail possible in both the problem definition and the results (Versteeg and Malalasekera, 2007). Despite these advantages, CFD is still not perfect and is in a continuous process of improvement. Numerical models need to be validated and verified with reliable data and this is still a challenge for many types of complex problems.

4.1. Numerical Procedures

In the present thesis, the Ansys Workbench software was used to perform numerical simulations. This software includes a collection of computer modeling softwares including those needed to model and simulate the problem under study. To create the computational domain, the Ansys DesignModeler was used and to generate the mesh from this domain, the Ansys Meshing was employed. After the mesh was generated, it was imported by Ansys Fluent 18.1, where it was possible to set up the problem parameters, perform the numerical simulations and generate the results. A total of 864 simulations were performed in the present study. To perform several simulations in sequence, Ansys Fluent 18.1 allows the use of scripts for process automation.

The software Ansys Fluent 18.1 is based on the finite volume method to solve its problems (ANSYS, 2017a) and has several different mathematical models for multiphase and phase change simulations. It is important to note that in the cases where there was no phase change, the only

difference in the numerical model for the boiling cases was the disabling of the phase change process. After an investigation on the literature of the main models used in boiling problems and after the execution of several tests in Ansys Fluent 18.1 using its different models (Volume of Fluid, Mixture and Eulerian for the multiphase and RPI Boiling, Lee and Thermal Phase Change for the boiling), the choice for the selected models, Eulerian and RPI Boiling, was made following the conditions of validation of the model and of the shortest time of processing. That is, once the tested models were properly validated with reliable data present in other works, preference was given to those that presented the shortest processing time. This preference for shorter processing time is due to the fact that the more simulations that are possible to be performed, for a given period of time, the more degrees of freedom can be explored (and more precisely), thus increasing the level of possible geometric analysis. It is important to mention that this stage of the study was considerably time consuming and required intense effort due to the complexity of the models being tested and the lack of previous works with a similar approach.

The application of the TOPSIS method was carried out with the use of the spreadsheet software Microsoft Excel and the implementation of the method into the spreadsheets was verified through the website Decision Radar (<https://decision-radar.com/>), which provides an online and totally free TOPSIS calculator.

4.1.1. Finite Volume Method

The finite volume method (FVM) is one of the most used numerical techniques in CFD. It is a discretization method that transforms partial differential equations, which represent conservation laws, into discrete algebraic equations over finite volumes. The flow leaving one face of a given volume is identical to that entering the adjacent volume, making this a conservative method. In FVM, the geometry of the domain under study is first discretized into finite volumes. Then, the differential equations representing the conservation laws are integrated and transformed into algebraic equations in each of the finite volumes. Afterwards, it is possible to generate an algebraic system of equations that can be solved by a computer so that the values of the dependent variables are found (Maliska, 2004).

One of the advantages of the FVM is that it is considerably flexible and can be used in both structured and unstructured meshes. The meshes created in the FVM have some characteristics such as covering the entire domain, leaving no free spaces between cells and also no cell of the mesh simultaneously occupies the space of another. Figure 4.1 shows some examples of FVM meshes, such as one-dimensional (a), structured two-dimensional Cartesian (b), structured two-dimensional

curvilinear (c) and unstructured two-dimensional (d).

In a generic way, considering a variable ϕ (which can be the temperature or another property for example), the conservative form of the equations of a flow in the transient regime can be written as (Veersteeg and Malalasekera, 2007):

$$\frac{\partial(\rho\phi)}{\partial t} + \text{div}(\rho\phi\vec{v}) = \text{div}(\Gamma.\text{grad}\phi) + S_\phi \quad (4.1)$$

where ϕ is the scalar quantity being transported, Γ is the diffusion coefficient, $\vec{v}$ is the velocity field vector, and S_ϕ is the energy source or sink term per unit volume (W/m^3).

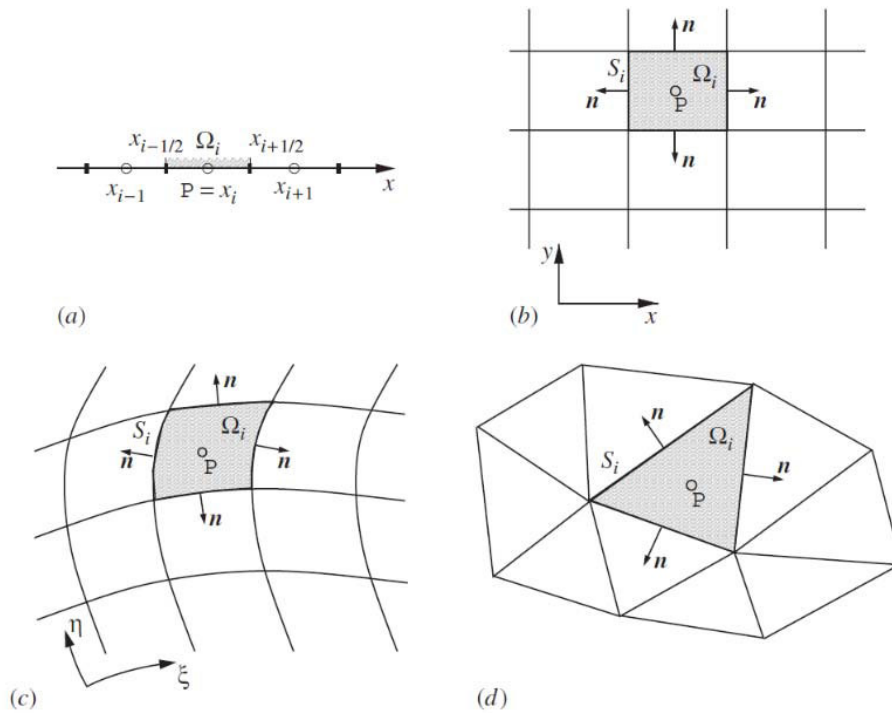


Figure 4.1 – Examples of meshes of the finite volume method (Zikanov, 2010).

Equation 4.1 is known as the transport equation and is used for incompressible, transient and convective heat transfer flows. Integrating Eq. 4.1 over a control volume we obtain:

$$\int_V \frac{\partial(\rho\phi)}{\partial t} dV + \int_A \vec{n} \cdot (\rho\phi\vec{v}) dA = \int_A \vec{n} \cdot (\Gamma.\text{grad}\phi) dA + \int_V S_\phi dV \quad (4.2)$$

where A is the area of the control volume (m^2), V is the volume of the control volume (m^3) and $\vec{n}$ is a

unit vector normal to the surface of the control volume.

Equation 4.2 represents the flow balance of ϕ in a control volume and is applied to all control volumes of a computational domain. Ansys Fluent 18.1 applies a technique to convert a scalar transport equation, such as Eq. 4.2, into an algebraic equation that can be solved numerically by the computer. The discretized form of Eq. 4.2 for a control volume can be written by (ANSYS, 2017a):

$$\frac{\partial(\rho\phi)}{\partial t} V + \sum_f^{N_{faces}} (\rho_f \vec{v}_f \phi_f \vec{A}_f) = \sum_f^{N_{faces}} \Gamma_\phi \nabla \phi_f \vec{A}_f + S_\phi V \quad (4.3)$$

where f is the sub-index referring to a given face of the control volume, N_{faces} is the number of faces around the control volume in question, ϕ_f is the value of ϕ on the face f of the control volume and $\nabla \phi_f$ is the gradient of ϕ on the face f .

The next step is to linearize the discretized equation given by Eq. 4.3, in the form of:

$$a\phi = \sum_{nb} a_{nb} \phi_{nb} + b \quad (4.4)$$

where nb refers to the neighboring control volumes a and b and are the linearized coefficients of ϕ .

This equation applies to each of the control volumes and in this way, it is possible to generate a matrix solvable by a computer to calculate the value ϕ in all the control volumes.

Spatial Discretization

In convective problems with the finite volume method, it is necessary to know the values of ϕ on the faces of a control volume, as seen in Eq. 4.3. Since the value of ϕ is defined at the center of the cell, it is necessary to use interpolation methods to calculate the value on the faces of the control volumes. This is necessary in convective problems, since in advection there is a distribution of the transported quantities only in the direction of the flow, differently from diffusive problems, where the distribution occurs in all directions (Maliska, 2004).

Ansys Fluent 18.1 has some interpolation schemes for spatial discretization and the present study applied the first-order Upwind and second-order Upwind schemes.

- ***First Order Upwind:*** This scheme assumes that the value of a variable ϕ in the center of the cell also represents the average value of ϕ across the entire cell. In this way, the value ϕ_f of a given face f will be defined as the value ϕ of the cell upstream of this face:

$$\phi_f = \phi \quad (4.5)$$

- **Second Order Upwind:** This scheme is more accurate than the first order Upwind. It achieves greater accuracy through a multidimensional linear reconstruction method (Barth and Jespersen, 1989). In this, the value of ϕ_f can be calculated with the following equation:

$$\phi_f = \phi + \nabla\phi \cdot \vec{r} \quad (4.6)$$

where $\vec{r}$ is the displacement vector from the centroid of the face of the control volume to the centroid of the upstream control volume.

Pressure-Velocity Coupling

In convective problems, generally the velocity and pressure fields in a computational domain are not known at first. They must be calculated using conservation equations. In a compressible flow this is not a big problem as the pressure can be obtained as a function of the temperature and density of the fluid. However, in incompressible flows the pressure is independent of these, which generates the need for an alternative way of calculating the pressure field. The software Ansys Fluent 18.1 solves this problem by creating an additional condition for the pressure, reformulating the continuity conservation equation. For this, algorithms are used, which can calculate the flow fields in two ways: segregated and coupled (Moukalled et al., 2016).

- **SIMPLE Algorithm:** The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) is a segregated algorithm. In it, the momentum conservation equation is calculated from an initially guessed pressure field. A correction of this pressure field is calculated using a multi-mesh algebraic method to satisfy the continuity conservation equation.
- **COUPLED Algorithm:** As the name implies, COUPLED is an algorithm that treats velocity and pressure in a coupled way and that in this study will be used throughout all cases. While segregated algorithms calculate conservation equations and pressure correction separately, which makes convergence more time consuming, COUPLED solves all equations simultaneously. This is done through an implicit discretization of the pressure gradient terms

in the momentum conservation equation.

4.1.2. Numerical Parameters

The definition of the simulation parameters applied in the present study was carried out after consultations in reference works that used the same multiphase and phase change models and also after an extensive number of tests carried out with the intention of obtaining the results that best reproduced flows with boiling (through validations and verifications, presented below). Table 4.1 lists the main numerical parameters applied in the simulations of the present study. It is important to mention that in Ansys Fluent 18.1, the spatial discretization of the volume fraction is only possible with first-order and not second-order *Upwind*. Using an Intel i7-11700K processor with 3.60 GHz, the simulations where the boiling process was enabled took between one and four hours, depending on the geometry complexity (the greater the block's height, the longer it took for the simulation to perform). The single-phase simulations took much less time to simulate because of the much simpler models, taking between five and fifteen minutes each.

Table 4.1 – Numerical parameters applied in the present study.

Time	Transient
Pressure-velocity coupling	COUPLED
Spatial discretization – momentum	Second order upwind
Spatial discretization – volume fraction	First order upwind
Spatial discretization – turbulent kinetic energy	Second order upwind
Spatial discretization – specific dissipation rate	Second order upwind
Spatial discretization – energy	Second order upwind
Residuals	1×10^{-6} (continuity) 5×10^{-6} (velocity, energy, κ , ω and volume fraction)

4.2. Mesh Independence Study

To evaluate the independence of the results from the refinement of the meshes used, mesh independence tests were performed in this thesis. Two different types of meshes were used: the first type was applied to the cases without boiling while the second type was applied to the cases with boiling. The use of different meshes between the cases was necessary due to the RPI boiling model and its characteristics in relation to mesh refinement in the region close to the walls.

The mesh independence tests were applied to the problem described in section 3.1 of this thesis, with degrees of freedom defined at $H_1/L_1 = 1.55$, $H_2/L_2 = 1.55$ and $L_3/L_{ocu} = 0.375$. This geometry corresponds to the most demanding case from the computational point of view among all the proposed ones, since it is the geometry in which the flow encounters the greatest obstruction to its displacement. To compare the results obtained from each mesh, the value of the total heat transfer rate (q) from the heated region to the fluid was considered. The following formula was applied to calculate the acceptance criteria for the meshes:

$$\left| \frac{(q^j - q^{j+1})}{q^j} \right| < CC \quad (4.7)$$

where q^j represents the heat transfer rate of a given mesh (W), q^{j+1} the heat transfer rate of the mesh with refinement one level above q^j (W) and CC is the convergence criterion.

As convergence criterion for the meshes of the different cases, it was adopted an acceptance criteria of $CC = 3 \times 10^{-3}$.

4.2.1. Mesh Independence Test - Cases without Boiling

Figure 4.2 shows the mesh used in the cases without boiling. The mesh uses a mixture of triangular and rectangular volumes, with rectangular volumes used predominantly. It is possible to observe that it also has a refinement near the channel walls. Such refinement was included with the intention of better capturing the effects of turbulence and heat transfer. With this refinement, the value of y^+ was kept around 1.0 throughout all simulated cases. This parameter is critical for turbulent flows and represents how well the turbulence model reproduces the effects of turbulence in regions close to the walls. In the model $\kappa - \omega$ (and its variations, such as the SST) it is important to keep the value of y^+ as close to 1.0 as possible (Menter, 1994; Menter et. al, 2003). A mesh with 91,429 volumes satisfactorily met the established criteria.

Table 4.2 presents the results obtained in the mesh independence test for the cases without boiling.

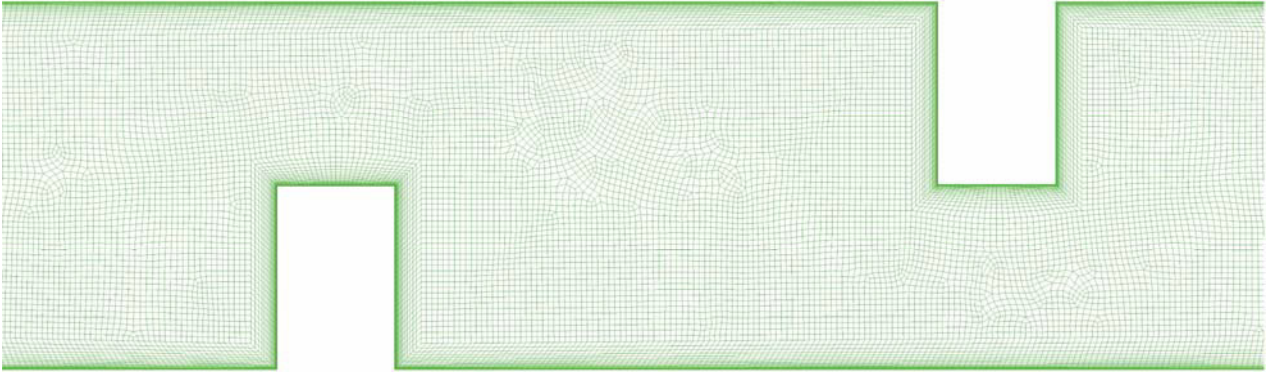


Figure 4.2 - Mesh with triangular and rectangular finite volumes employed in the cases without boiling.

Table 4.2 - Mesh independence test - cases without boiling

Number of Volumes	q (W)	Relative Deviation
3,852	77,365.77	0.04934
7,817	81,182.85	0.13916
15,284	92,479.85	0.03848
30,352	88,921.02	0.01635
60,555	87,467.08	0.00329
91,429	87,178.51	0.00275
121,142	86,938.67	-

4.2.2. Mesh Independence Test – Cases with Boiling

The mesh used in the turbulent cases with boiling differs significantly from the mesh used in the cases without boiling. This is because the RPI model has some of its constitutive relationships derived from one-dimensional correlations, with its terms expressed in relation to average rather than local temperature and velocity. When this model is applied to two-dimensional or three-dimensional geometries, these correlations need to be revised, otherwise they would generate solutions dependent on mesh refinement. A way to solve this problem was proposed by Koncar et al. (2005), where the value of y^+ is fixed at 250. This is a counterintuitive approach and works for meshes with a considerably coarse refinement. After a certain degree of refinement, the results start to deviate from the correct. This proposal by Koncar et al. (2005) has been shown to be able to correctly reproduce turbulent boiling flows in two and three-dimensional geometries, which has been proven by other works (Zhang et al, 2015; Das and Punekar, 2016). The Ansys Fluent 18.1 implements this solution in the RPI model and allows fixing the value of y^+ (ANSYS, 2017b). Figure 4.3 shows the mesh used

in turbulent boiling cases. It is possible to observe that it does not have refinement close to the walls as in the other cases and follows a structured pattern throughout the entire computational domain, only with rectangular volumes. In this case, a mesh with 30,450 finite volumes satisfactorily met the established convergence criterion. Table 4.3 presents the results of the mesh independence test for the cases with boiling.

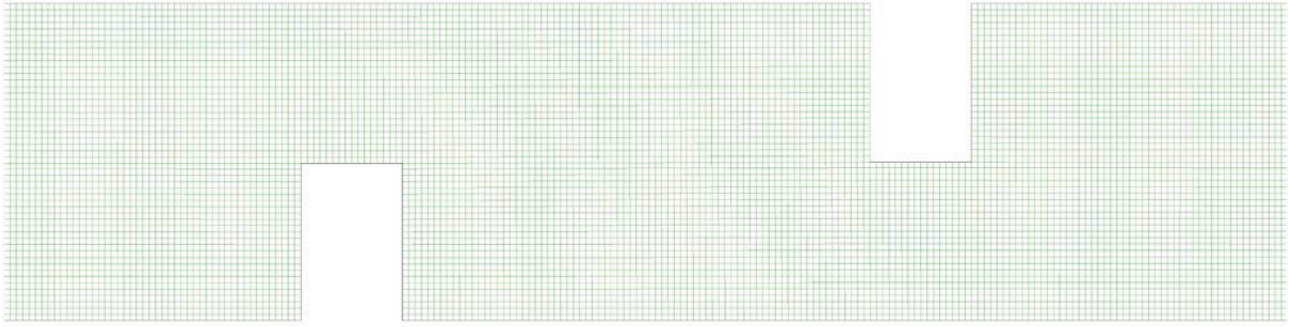


Figure 4.3 - Mesh with triangular and rectangular finite volumes employed in the cases with boiling.

Table 4.3 - Mesh independence test - cases with boiling.

Number of Volumes	q (W)	Relative Deviation
3,852	95,706.27	0.03039
7,658	98,615.26	0.00733
15,948	99,337.64	0.00388
30,450	98,951.31	0.00146
61,747	99,095.72	-

5. RESULTADOS

Este capítulo apresenta os resultados obtidos no desenvolvimento desta tese. Primeiramente, são mostrados os resultados da validação do modelo numérico para escoamentos internos convectivos com ebulição aplicado no presente estudo. Em seguida, são apresentados os resultados da avaliação geométrica aplicada através das análises térmica, fluidodinâmica, multi-objetivo e do Coeficiente de Desempenho (*COP*), onde foram determinadas as geometrias ótimas dos casos com e sem ebulição em cada uma das análises.

5.1. Validação do Modelo Numérico Aplicado

O modelo numérico utilizado no presente estudo foi validado a partir de um teste realizado em um canal, com o mesmo domínio computacional da Figura 5.1. Os resultados foram comparados com resultados experimentais e numéricos e com correlações empíricas de outros trabalhos. O teste foi realizado em um canal axissimétrico bidimensional, com diâmetro $D = 15.4$ mm e comprimento $L = 2.0$ m. O fluido entra no canal com uma velocidade aproximada $V_{in} = 1.0$ m/s, $T_{in} = 470.44$ K e um fluxo de calor igual à $q'' = 570$ kW/m² é aplicado uniformemente distribuído nas paredes do canal. O número de Reynolds do escoamento é igual a 104,420. Além disso, o canal tem uma pressão de operação de 4.5×10^6 Pa e é posicionado na vertical com o fluido escoando para cima e, portanto, com a gravidade atuando na direção oposta ao eixo x .

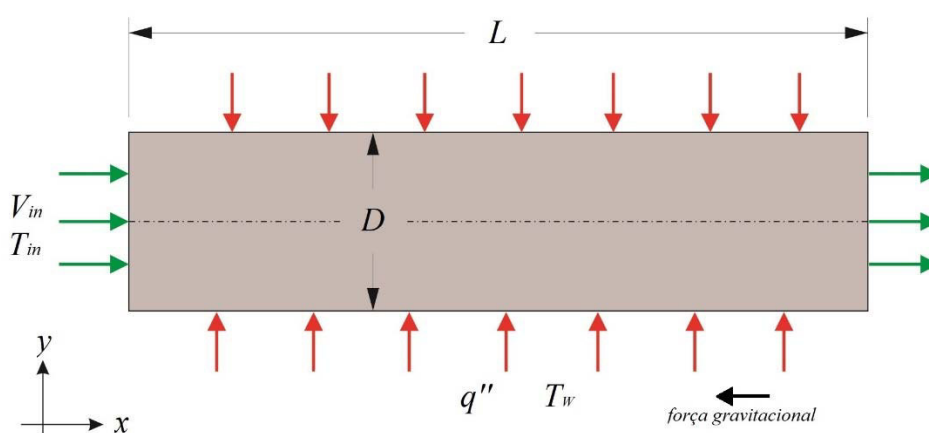


Figura 5.1 – Domínio computacional utilizado para validar o modelo numérico aplicado no presente estudo.

A Figura 5.2 apresenta os resultados obtidos da fração de volume de vapor ao longo do canal, em comparação com o resultado experimental obtido por Bartolomej e Chanturiya (1967), o resultado

numérico obtido por Huiying Li et al. (2011) e a correlação empírica de Zuber e Findlay (1965), dada pela Eq. (2.12). A fração de volume do vapor demonstrada na figura é correspondente a média da fração de volume em diferentes seções transversais ao longo do canal.

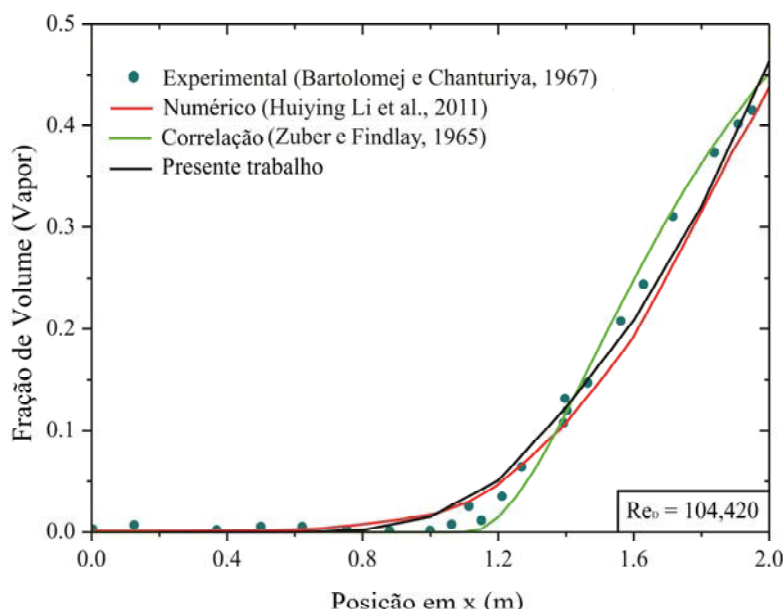


Figura 5.2 – Comparação da fração de volume de vapor ao longo do canal obtida no presente estudo em relação a outros trabalhos.

A Figura 5.3 apresenta a comparação dos resultados obtidos no presente estudo para a temperatura média do líquido ao longo do canal e os resultados obtidos por Bartolomej e Chanturiya (1967) e por Huiying Li et al. (2011). As temperaturas observadas na Figura 5.3 correspondem a média da temperatura em diferentes seções transversais ao longo do canal.

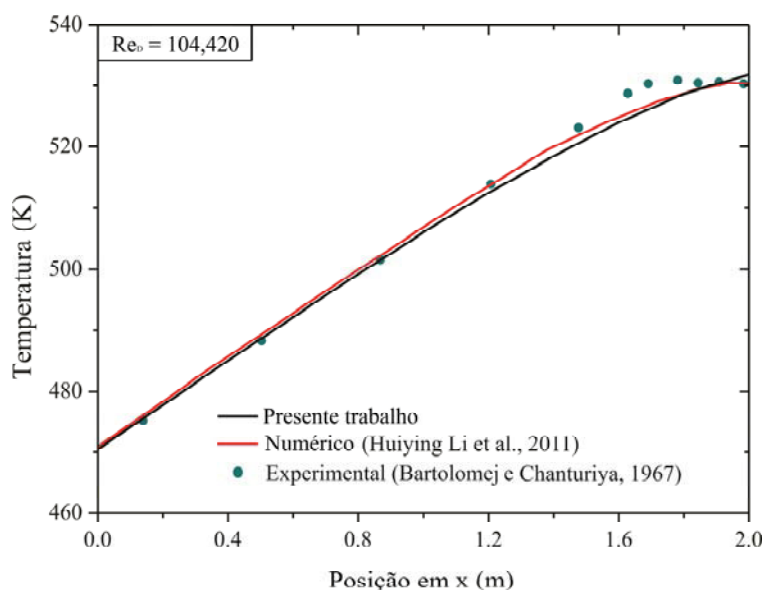


Figura 5.3 – Comparação da temperatura média do líquido ao longo do canal.

A Figura 5.4 mostra a temperatura da parede do canal obtida no presente estudo em comparação com os resultados de Bartolomej e Chanturiya (1967), Huiying Li et al. (2011) e também com a correlação empírica de Thom et al. (1965), dado pela Eq. (2.24). É importante salientar que esta correlação representa um valor único para a temperatura da parede ao longo do canal e é válida apenas a uma certa distância da entrada do canal, o que explica a diferença de temperatura observada entre esta correlação e os demais resultados na faixa $0.0 \text{ m} \leq x \leq 0.4 \text{ m}$.

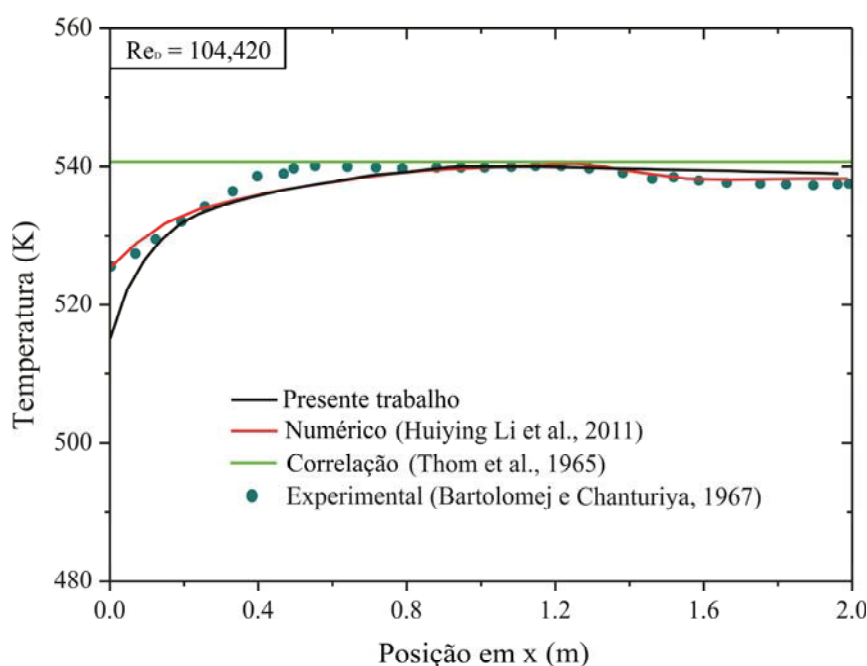


Figura 5.4 – Comparação da temperatura da parede ao longo do canal.

A partir dos resultados apresentados nas Figuras 5.2, 5.3 e 5.4, é possível observar que os resultados obtidos no presente estudo para o processo de ebulição e também das temperaturas ao longo do canal estão de acordo com os resultados obtidos em experimentos, correlações empíricas e em outros modelos numéricos e, assim, concluiu-se que o modelo numérico utilizado no presente estudo é adequado para simular escoamentos convectivos com ebulição no regime turbulento.

5.2. Avaliação Geométrica de um Canal com Dois Blocos Alternados Aquecidos Sujeito a Escoamentos Turbulentos Convectivos com e sem Ebulição

Os resultados obtidos nas simulações realizadas no presente trabalho são apresentados a seguir em uma análise de três etapas, onde os princípios do Design Construtal apresentados na seção 2.2 são

aplicados na avaliação e otimização geométrica do problema proposto. Em todas as etapas são apresentados os resultados obtidos para os casos com e sem ebulição, de forma a ser possível comparar o efeito do processo de ebulição na otimização geométrica. Primeiramente, é realizada uma análise térmica para avaliar a influência da geometria dos blocos sobre o objetivo térmico, a maximização da taxa de transferência de calor, e encontrar a geometria ótima térmica que corresponde à maior taxa de transferência de calor encontrada entre todos os casos simulados. Em segundo lugar, uma análise fluidodinâmica é conduzida da mesma forma que a análise térmica, agora considerando o objetivo fluidodinâmico, a minimização da queda de pressão ao longo do canal, para determinar a geometria ótima fluidodinâmica que corresponde à menor queda de pressão encontrada entre todos os casos. Após, uma análise multi-objetiva, onde os objetivos térmico e fluidodinâmico são considerados simultaneamente, a partir da aplicação do método TOPSIS, com o objetivo de determinar a geometria ótima multi-objetivo, a geometria que satisfaz ambos os objetivos da melhor maneira possível em comparação com as outras geometrias. Nesta etapa, foram definidas cinco combinações diferentes de pesos para os objetivos térmico e fluidodinâmico. Por fim, é realizada uma análise considerando o Coeficiente de Desempenho (COP) como objetivo da otimização geométrica.

5.2.1. Análise Térmica

A Figura 5.5 mostra o efeito de H_1/L_1 sobre a taxa de transferência de calor (q) para diferentes valores de H_2/L_2 , $L_3/L_{ocu} = 0.5$ e para casos com e sem ebulição. Como esperado, os maiores valores de H_1/L_1 e H_2/L_2 produziram uma maior taxa de transferência de calor, devido à maior intrusão dos blocos no canal, o que aumenta a interação do calor entre os blocos aquecidos e o escoamento mais frio. Pode-se observar um aumento quase linear da taxa de transferência de calor com o aumento dos dois graus de liberdade. Também é possível observar que os casos com ebulição têm uma maior taxa de transferência de calor do que os casos sem ebulição, porém em ambos os casos o efeito de H_1/L_1 sobre (q) é semelhante. Da mesma forma, a Figura 5.6 mostra que a taxa transferência de calor uma vez maximizada (q)_{1,max} aumenta com o aumento de H_2/L_2 e também é maior para os casos com ebulição com um efeito similar de H_2/L_2 sobre (q) para os dois casos. Para todos os valores de H_2/L_2 analisados, a geometria ótima térmica do primeiro bloco, $(H_1/L_1)_{o,T}$, resultante foi igual a 1.55, o maior valor possível definido neste trabalho, como mostra a Figura 5.7.

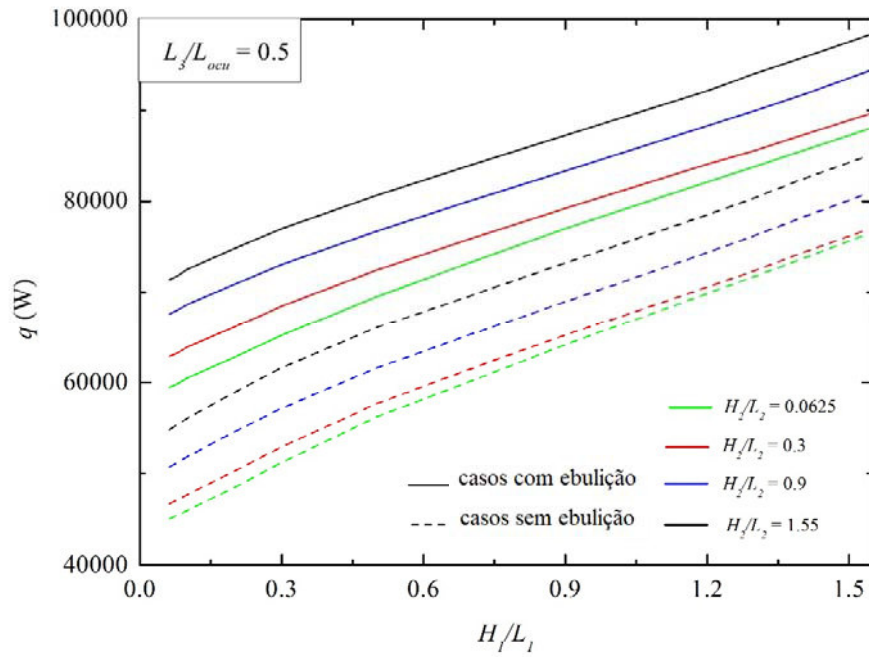


Figura 5.5 – Efeito de H_1/L_1 sobre a taxa de transferência de calor (q) para diferentes valores de H_2/L_2 .

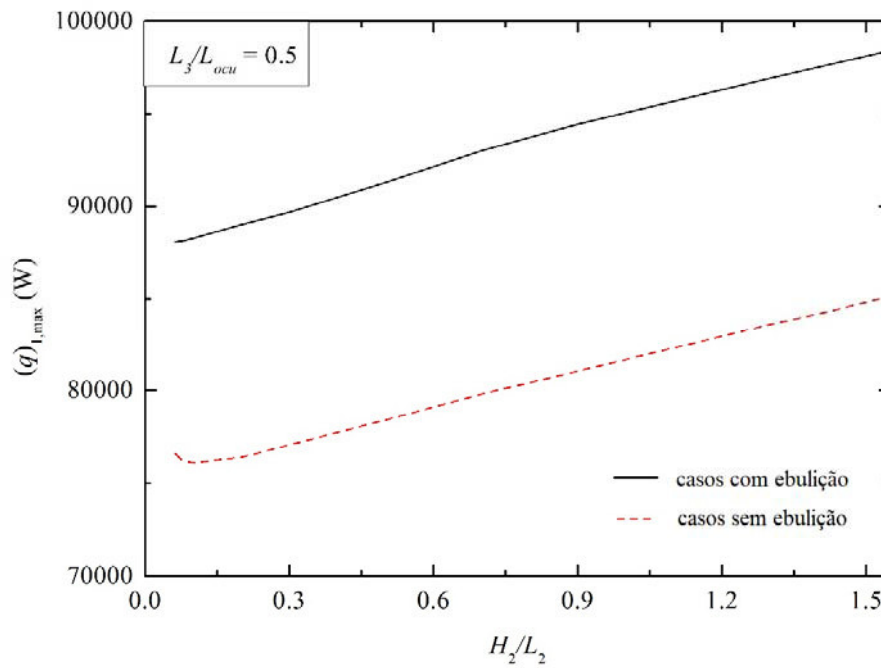


Figura 5.6 – Efeito de H_2/L_2 sobre a taxa de transferência de calor uma vez maximizada $(q)_{1,max}$.

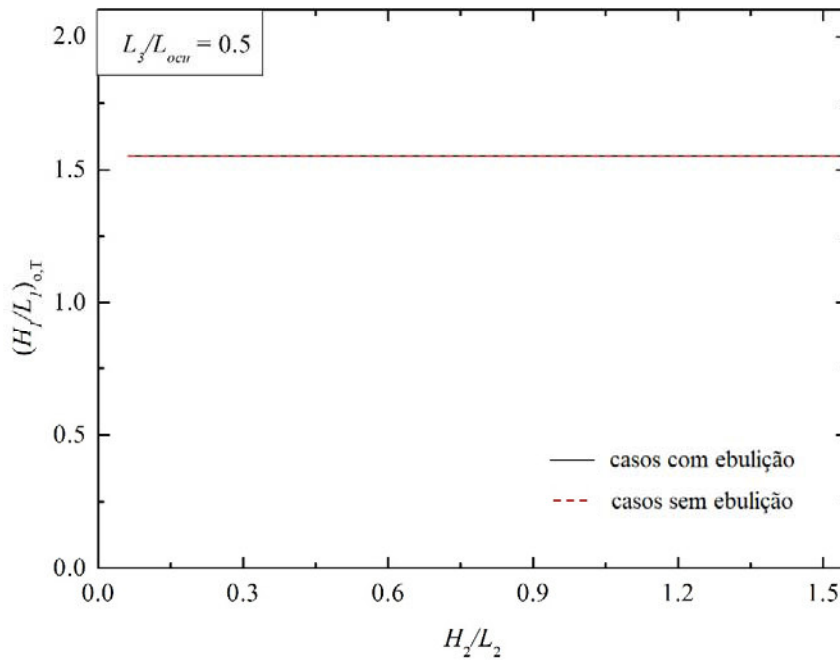


Figure 5.7 – Efeito de H_2/L_2 sobre a razão H_1/L_1 uma vez termicamente otimizada, $(H_1/L_1)_{0,T}$.

Em relação ao efeito de L_3/L_{ocu} , a Figura 5.8 demonstra que a variação entre os três valores definidos para esta dimensão (0.375, 0.5 e 0.625) praticamente não altera o valor da taxa de transferência de calor duas vezes maximizada, $(q)_{2,max}$, tanto para os casos com ebulição como para os casos sem ebulição. Também, mais uma vez é possível observar que os casos com ebulição produzem uma maior taxa de transferência de calor do que os casos sem ebulição. As Figuras 5.9 e 5.10 mostram que para todos os valores de L_3/L_{ocu} , as melhores geometrias térmicas obtidas foram sempre as geometrias com os maiores valores das razões H_1/L_1 e H_2/L_2 , ou seja, a razão H_1/L_1 duas vezes otimizada termicamente, $(H_1/L_1)_{00,T}$, e a razão H_2/L_2 uma vez otimizada termicamente, $(H_2/L_2)_{0,T}$, se mantiveram constantes e igual a 1.55 para os três valores de L_3/L_{ocu} . Este comportamento se mostrou semelhante para os casos com e sem ebulição.

As Figuras 5.11 e 5.12 apresentam os campos de temperatura, fração de volume (apenas para caso com ebulição), pressão, velocidade e linhas de fluxo para as geometrias ótimas térmicas dos casos com e sem ebulição. Pode-se observar que em ambas as situações os dois blocos têm um papel importante na interação com o escoamento e que a maior altura dos blocos na geometria ótima força o escoamento a interagir com as paredes dos blocos, que se encontram a uma temperatura superior, o que aumenta a taxa de transferência de calor naquela região. De forma geral, é possível observar através dos campos das Figuras 5.11 e 5.12 que não existem grandes diferenças entre os casos com e

sem ebulição, porém é possível verificar através dos campos de temperatura que o caso com ebulição possui uma dissipação de calor ao redor dos dois blocos superior comparado ao caso sem ebulição, o que pode ser atribuído ao processo de ebulição e o calor latente necessário para a mudança de fase. A geometria ótima térmica tanto para os casos com ebulição como para os casos sem ebulição foi a mesma e igual a $(H_1/L_1)_{000,T} = 1.55$, $(H_2/L_2)_{00,T} = 1.55$ e $(L_3/L_{ocu})_{0,T} = 0.375$, os maiores valores possíveis para as razões H_1/L_1 e H_2/L_2 e o menor valor possível de L_3/L_{ocu} entre as configurações estudadas. Para os casos com ebulição, esta geometria ótima gerou uma taxa de transferência de calor três vezes maximizada de $(q)_{3,max} = 98,629.32$ W. Para os casos sem ebulição, $(q)_{3,max}$ foi igual a 85,619,89 W. Esta diferença significa um desempenho térmico da geometria ótima 15.19 % superior para o caso com ebulição. Para comparação, nos casos com ebulição a geometria ótima gerou uma taxa de transferência de calor 65.85 % maior do que a pior geometria térmica, que teve uma taxa de transferência de calor igual de $q = 59,468.27$ W. Nos casos sem ebulição esta diferença foi maior e igual a 90.05 %, sendo a menor taxa de transferência de calor observada igual a 45,050.49 W.

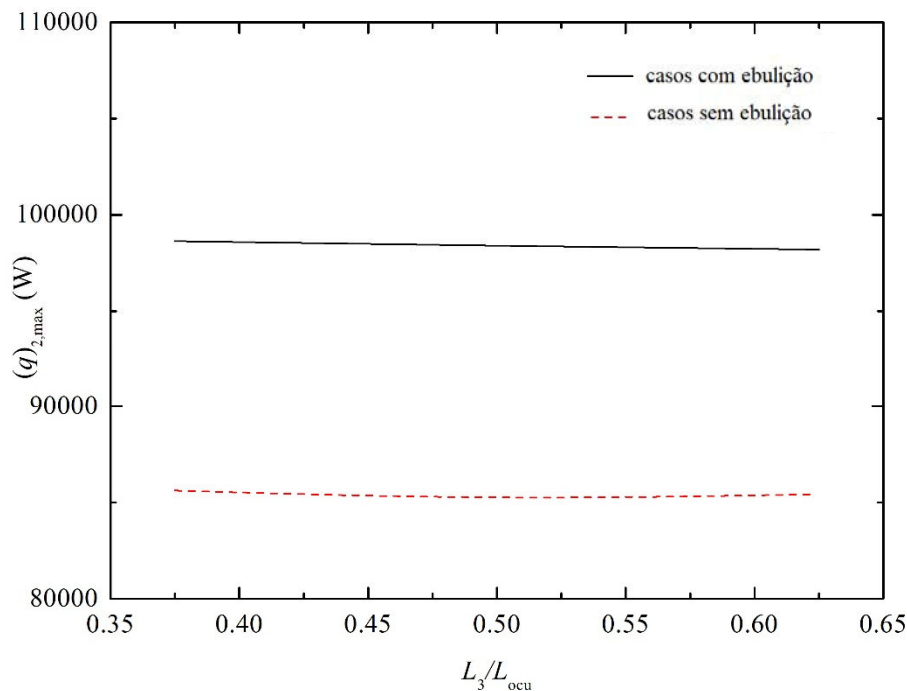


Figura 5.8 – Efeito de L_3/L_{ocu} sobre a taxa de transferência de calor duas vezes maximizada $(q)_{2,max}$.

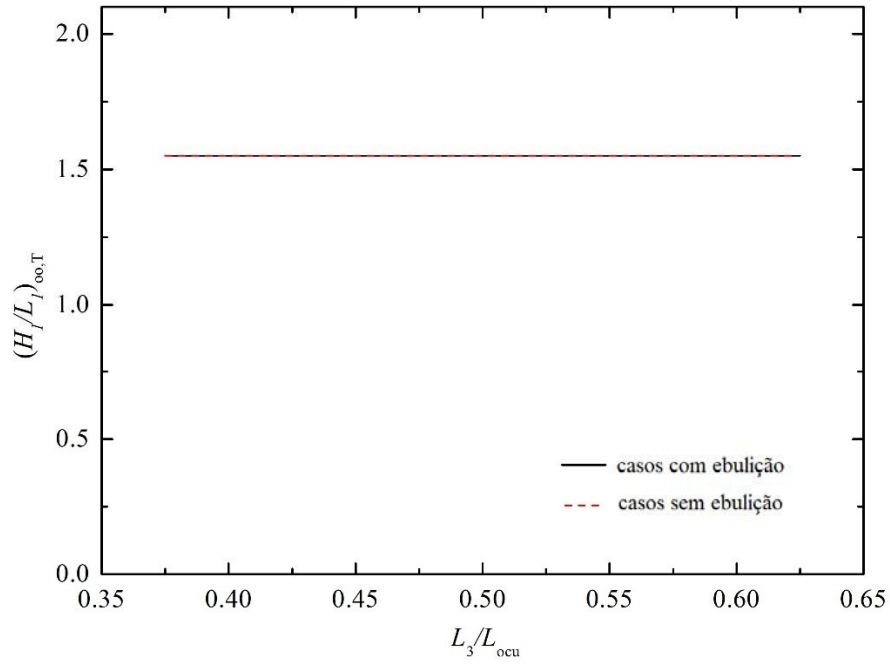


Figura 5.9 – Efeito de L_3/L_{ocu} sobre a razão H_1/L_1 duas vezes otimizada termicamente, $(H_1/L_1)_{oo,T}$.

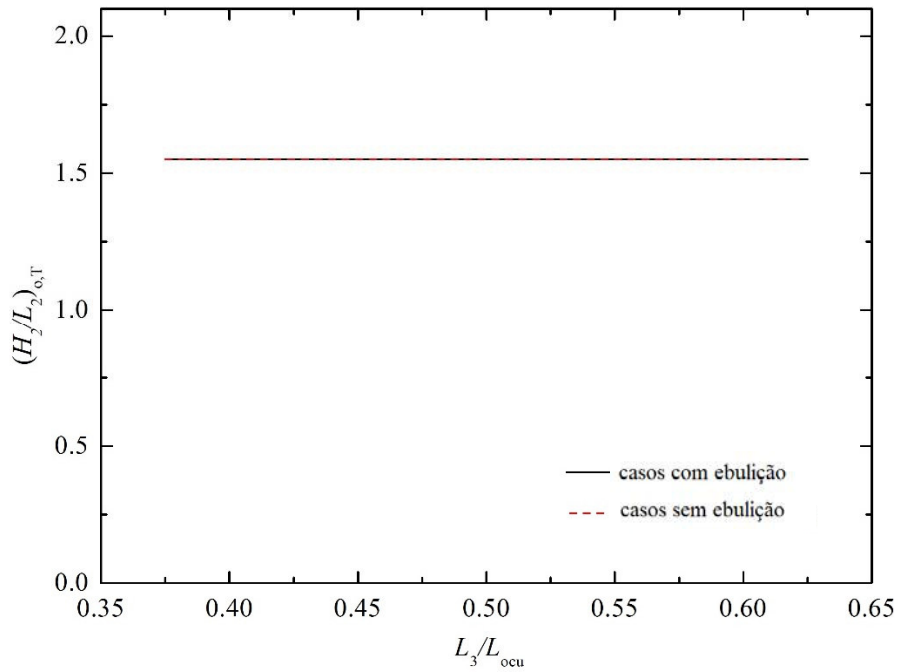


Figura 5.10 – Efeito de L_3/L_{ocu} sobre a razão H_2/L_2 uma vez otimizada termicamente, $(H_2/L_2)_{oo,T}$.

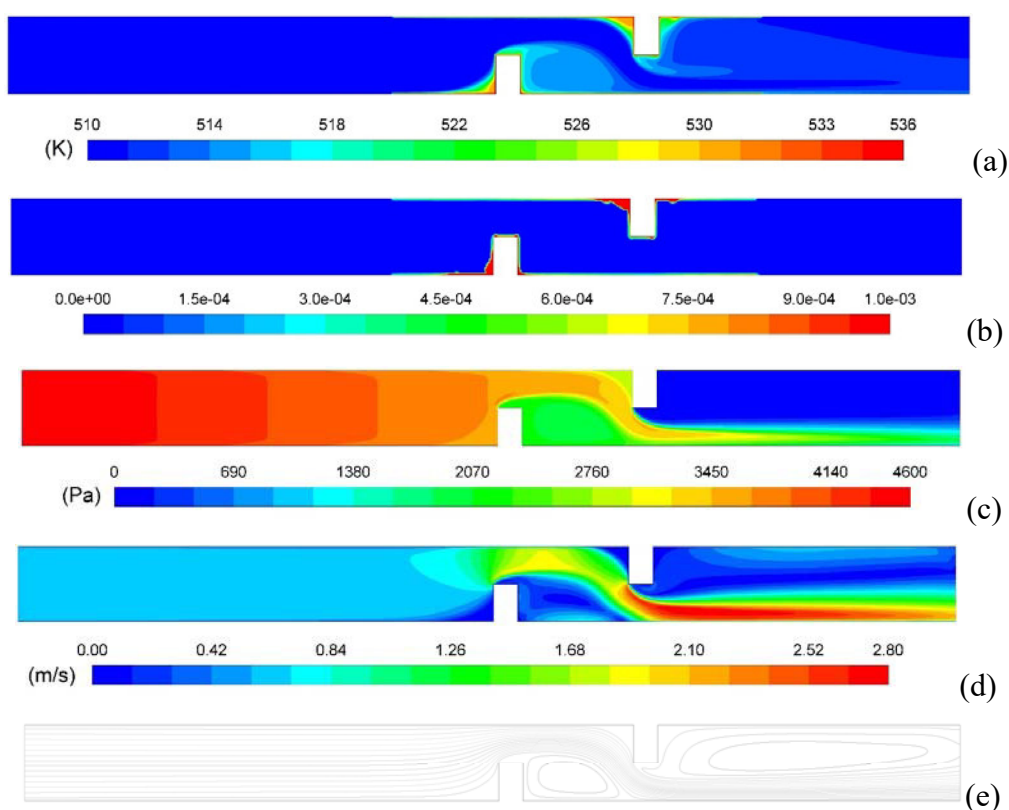


Figura 5.11 – Campos de temperatura (K) (a), fração de volume (b), pressão (Pa) (c), velocidade (m/s) (d) e linhas de fluxo (e) para as geometrias ótimas térmicas com ebulição, $(H_1/L_1)_{oo,T} = 1.55$, $(H_2/L_2)_{oo,T} = 1.55$ e $(L_3/L_{ocu})_{o,T} = 0.375$.

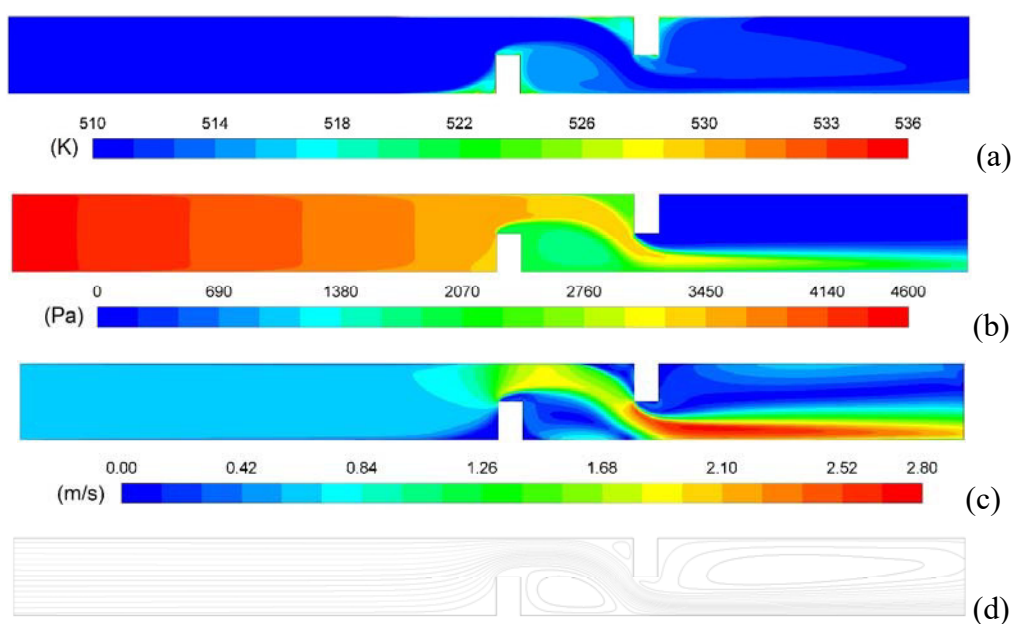


Figura 5.12 – Campos de temperatura (K) (a), pressão (Pa) (b), velocidade (m/s) (c) e linhas de fluxo (d) para as geometrias ótimas térmicas sem ebulição, $(H_1/L_1)_{oo,T} = 1.55$, $(H_2/L_2)_{oo,T} = 1.55$ e $(L_3/L_{ocu})_{o,T} = 0.375$.

5.2.2. Análise Fluidodinâmica

A partir dos resultados obtidos para a diferença de pressão entre a entrada e a saída do canal, é possível analisar o problema do ponto de vista fluidodinâmico e determinar quais geometrias proporcionam um melhor desempenho nesse quesito, ou seja, quais apresentam a menor queda de pressão ao longo do canal. A Figura 5.13 mostra o efeito de H_1/L_1 sobre a queda de pressão ao longo do canal (ΔP) para diferentes valores de H_2/L_2 , $L_3/L_{ocu} = 0.5$ e para casos com e sem ebulição, enquanto a Figura 5.14 mostra o efeito de H_2/L_2 sobre a queda de pressão uma vez minimizada (ΔP)_{1,min}. Como esperado, os menores valores de H_1/L_1 e H_2/L_2 fornecem os melhores resultados fluidodinâmicos. O aumento dessas razões provoca um aumento da queda de pressão. Além disso, a Fig. 5.13 mostra que na região onde as geometrias são ótimas, para menores razões de H_1/L_1 , o desempenho fluidodinâmico dos casos com ebulição foi superior aos casos sem ebulição. Para as maiores razões de H_1/L_1 observou-se um comportamento contrário. A Figura 5.14 mostra que para todas razões de H_2/L_2 o desempenho dos casos com ebulição foi superior aos casos sem ebulição. A Figura 5.15 mostra que para cada H_2/L_2 , o valor ótimo de H_1/L_1 é constante e igual ao menor valor possível, $(H_1/L_1)_{o,F} = 0.0625$. É possível observar nas Figuras 5.13, 5.14 e 5.15 que apesar de os casos com ebulição proporcionarem uma menor queda de pressão, não há diferenças consideráveis dos efeitos de H_1/L_1 e H_2/L_2 sobre a queda de pressão ao longo do canal ao comparar-se os casos com e sem ebulição.

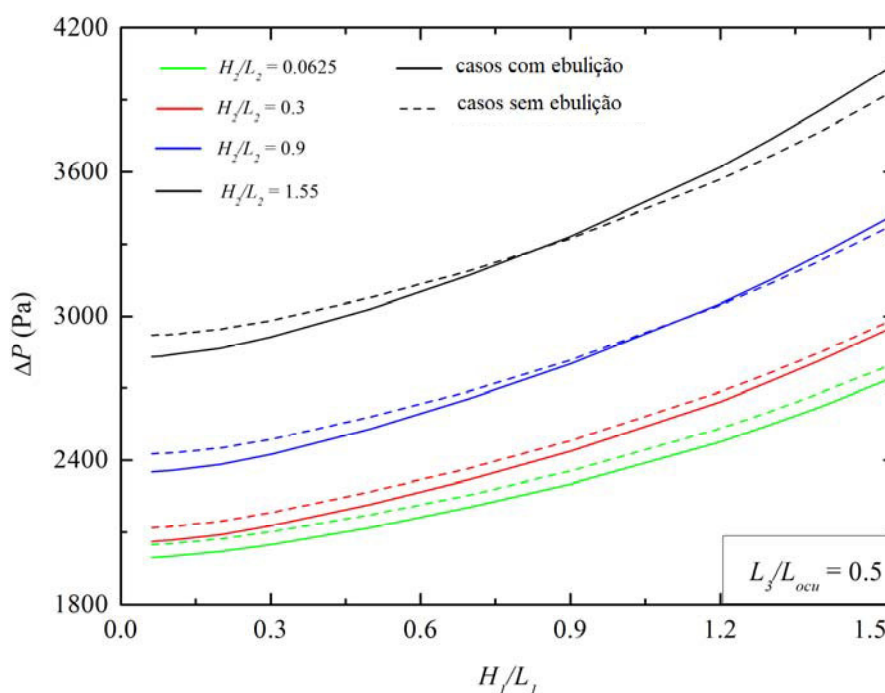


Figura 5.13 – Efeito de H_1/L_1 sobre a queda de pressão (ΔP) para diferentes valores de H_2/L_2 .

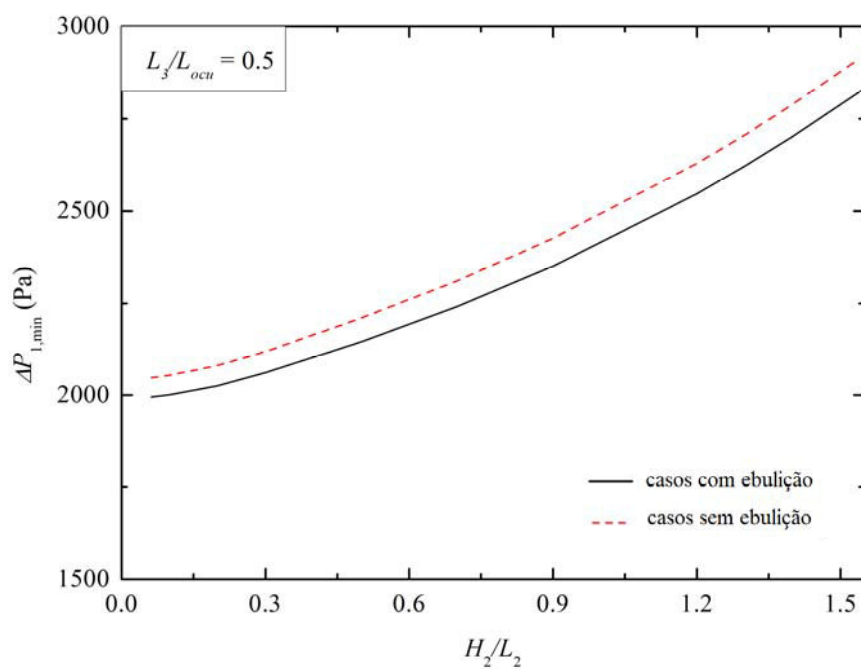


Figura 5.14 – Efeito de H_2/L_2 sobre a queda de pressão uma vez minimizada $(\Delta P)_{1,min}$.

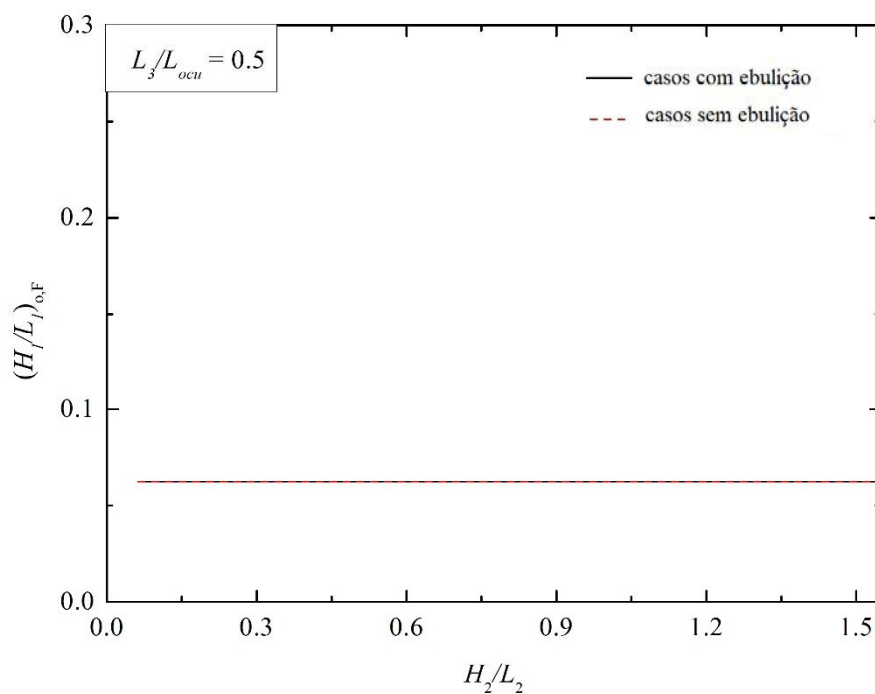


Figura 5.15 – Efeito de H_2/L_2 sobre a razão H_1/L_1 uma vez otimizada fluidodinamicamente, $(H_1/L_1)_{o,F}$.

É importante notar na Figura 5.13 que os casos sem ebulição têm uma menor queda de pressão para valores de H_1/L_1 maiores que 0.9 quando H_2/L_2 é igual a 0.9 e 1.55. A explicação disto pode ter relação com o observado por Abdous et al. (2016), onde o processo de ebulição apresentou um maior desempenho térmico e fluidodinâmico para blocos com menores alturas comparadas as de maiores alturas.

As Figuras 5.16, 5.17 e 5.18 mostram o efeito da razão L_3/L_{ocu} sobre a queda de pressão duas vezes minimizada, $(\Delta P)_{2,min}$, e as geometrias ótimas fluidodinâmicas, $(H_1/L_1)_{oo,F}$ e $(H_2/L_2)_{o,F}$, respectivamente. É possível observar na Figura 5.16 que o aumento da distância entre os dois blocos (o aumento da razão L_3/L_{ocu}) mantém a diferença de pressão entre a entrada e a saída do canal (ΔP) praticamente constante, o que ocorre de forma semelhante para os casos com e sem ebulição, sendo que os casos com ebulição apresentaram um desempenho fluidodinâmico superior aos casos sem ebulição. As Figuras 5.17 e 5.18 mostram que as razões H_1/L_1 duas vezes fluidodinâmico otimizada, $(H_1/L_1)_{oo,F}$, e H_2/L_2 uma vez fluidodinâmico otimizada, $(H_2/L_2)_{o,F}$, se mantiveram constantes e iguais a 0.0625 para os três valores de L_3/L_{ocu} , igualmente para os casos com e sem ebulição.

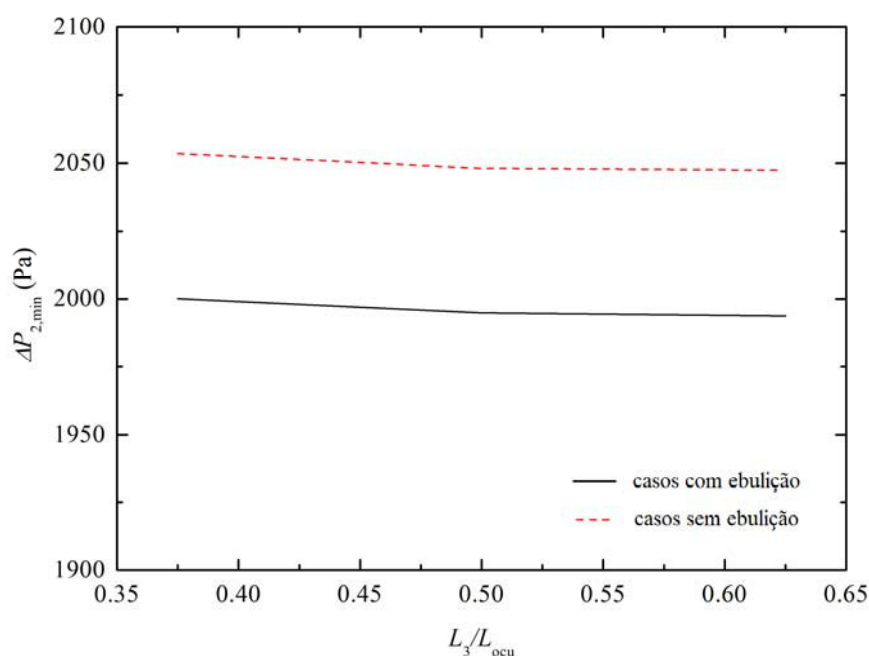


Figura 5.16 – Efeito de L_3/L_{ocu} sobre a queda de pressão duas vezes minimizada $(\Delta P)_{2,min}$.

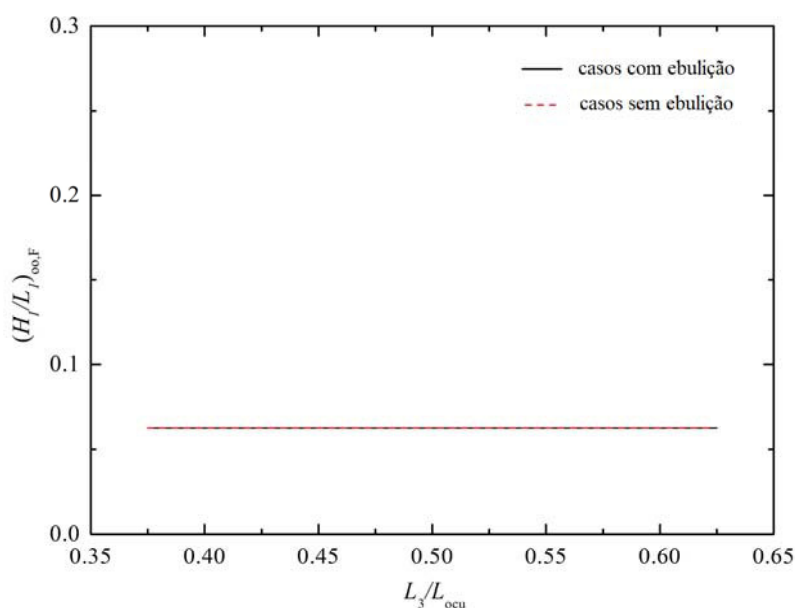


Figura 5.17 – Efeito de L_3/L_{ocu} sobre a razão H_1/L_1 duas vezes otimizada fluidodinamicamente, $(H_1/L_1)_{oo,F}$.

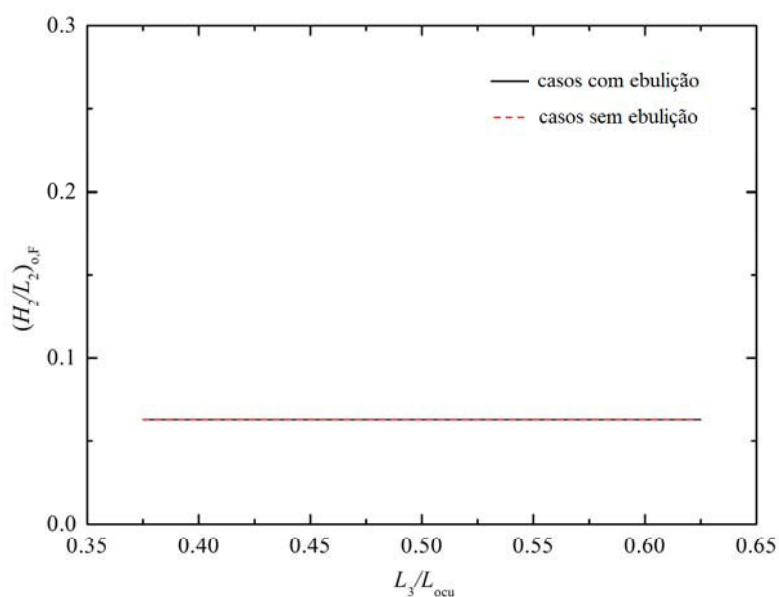


Figura 5.18 – Efeito de L_3/L_{ocu} sobre a razão H_2/L_2 uma vez otimizada fluidodinamicamente, $(H_2/L_2)_{o,F}$.

Nas Figuras 5.19 e 5.20 são apresentados os campos das geometrias ótimas fluidodinâmicas com e sem ebulição, respectivamente. Os resultados mais uma vez foram os esperados e a geometria onde os blocos têm a menor intrusão no canal e a maior distância entre eles sendo a que demonstrou o melhor desempenho fluidodinâmico para os dois casos, com e sem ebulição. Esta geometria é a que

melhor facilita o escoamento do fluido e é completamente oposta a geometria ótima térmica (Figuras 5.11 e 5.12), como pode ser visto nos campos das Figuras 5.19 e 5.20. Em comparação aos campos da geometria ótima térmica, é possível observar que a geometria ótima fluidodinâmica apresenta uma velocidade máxima consideravelmente menor e também uma transição da pressão ao longo do canal muito mais suave porém ao custo de uma troca de calor menor entre o fluido e a região dos blocos. A geometria ótima fluidodinâmica para os casos com e sem ebulição é $(H_1/L_1)_{oo,F} = 0.0625$, $(H_2/L_2)_{oo,F} = 0.0625$ e $(L_3/L_{ocu})_{o,T} = 0.625$ e teve uma queda de pressão três vezes minimizada igual à $(\Delta P)_{3,min} = 1,993.80$ Pa no caso com ebulição e igual à $(\Delta P)_{3,min} = 2,047.48$ Pa no caso sem ebulição. Estes resultados significam que a geometria ótima fluidodinâmica teve uma queda de pressão 2.62 % menor quando a ebulição foi considerada em comparação quando ela não foi considerada. Em comparação com as piores geometrias, nos casos com ebulição a geometria ótima teve uma queda de pressão 52.43 % menor enquanto que nos casos sem ebulição a geometria ótima teve uma queda de pressão 49.68 % menor, tendo as piores geometrias obtido uma queda de pressão igual à $\Delta P = 4.191,41$ Pa nos casos com ebulição e igual à $\Delta P = 4.068,80$ Pa nos casos sem ebulição.

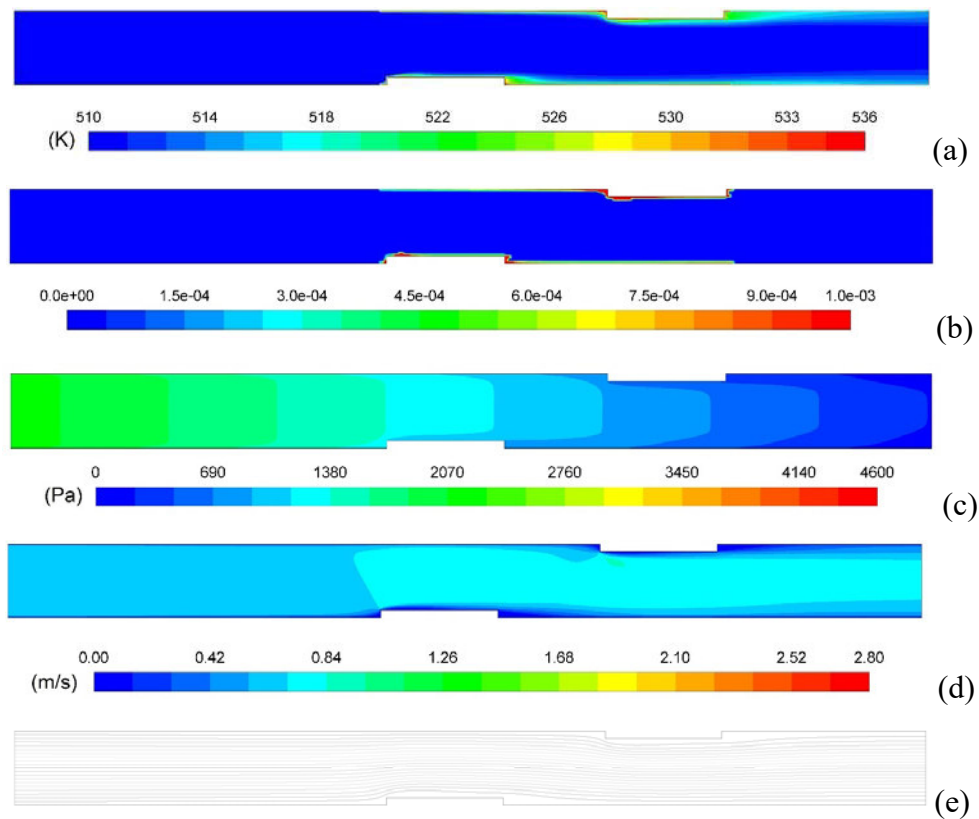


Figura 5.19 - Campos de temperatura (K) (a), fração de volume (b), pressão (Pa) (c), velocidade (m/s) (d) e linhas de fluxo (e) para as geometrias ótimas fluidodinâmicas com ebulição, $(H_1/L_1)_{oo,F} = 0.0625$, $(H_2/L_2)_{oo,F} = 0.0625$ e $(L_3/L_{ocu})_{o,F} = 0.625$.

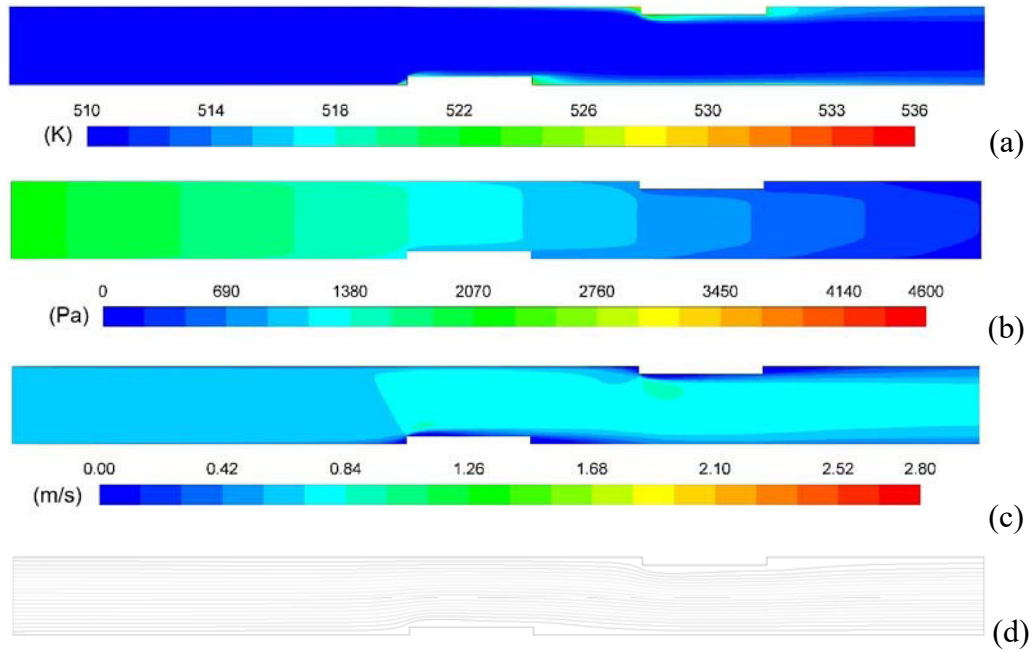


Figura 5.20 - Campos de temperatura (K) (a), pressão (Pa) (b), velocidade (m/s) (c) e linhas de fluxo (d) para as geometrias ótimas fluidodinâmicas sem ebulição, $(H_1/L_1)_{\text{oo},F} = 0.0625$, $(H_2/L_2)_{\text{oo},F} = 0.0625$ e $(L_3/L_{\text{ocu}})_{\text{o},F} = 0.625$.

5.2.3. Análise Multi-Objetivo

A realização das análises térmica e fluidodinâmica demonstraram que o processo de otimização geométrica resultou em geometrias ótimas completamente opostas nas duas análises. Em face a isto, fica clara a necessidade de uma abordagem multi-objetivo no processo de avaliação e otimização geométrica de canais com escoamentos convectivos. No presente estudo, a análise multi-objetivo é conduzida a partir da aplicação do método multi-critério TOPSIS, onde foram considerados pesos para os objetivos térmico (w_T) e fluidodinâmico (w_F). Para cada uma das combinações de pesos (b) aplicadas é calculado o parametro C_b , a proximidade relativa da solução ideal, de cada uma das geometrias simuladas, sendo que quanto mais próximo de 1.0 este valor for mais próxima a geometria estará da solução ideal. Através de C_b é possível comparar as geometrias, as alternativas possíveis, em relação ao multi-objetivo proposto (estabelecido pelos pesos w_T e w_F) e definir a geometria ótima deste ponto de vista. Na análise a seguir foram considerados cinco combinações de pesos diferentes, $w_T = 0.9$ e $w_F = 0.1$ (C_1), $w_T = 0.7$ e $w_F = 0.3$ (C_2), $w_T = w_F = 0.5$ (C_3), $w_T = 0.3$ e $w_F = 0.7$ (C_4) e $w_T = 0.1$ e $w_F = 0.9$ (C_5). É importante notar que o valor de C_b depende da combinação dos pesos, portanto não é possível comparar o desempenho multi-objetivo de geometrias (sejam iguais ou diferentes) em que houverem sido utilizadas combinações dos pesos diferentes (comparar C_1 com

C_4 por exemplo).

A Figura 5.21 mostra o efeito de H_1/L_1 sobre C_b para diferentes H_2/L_2 , $L_3/L_{ocu} = 0.5$ e para as cinco combinações de pesos definidas. É possível observar a evolução do efeito à medida que se aumenta o valor do peso considerado para o objetivo fluidodinâmico (w_F) e reduz-se o peso do objetivo térmico (w_T). Note-se que nos dois casos onde os pesos são considerados 0.1 e 0.9 os resultados são próximos aos obtidos nas análises térmica e fluidodinâmica, porém desta vez é possível observar a influência da inclusão de um outro objetivo ao problema, mesmo que ainda pequena.

Ao comparar-se os cinco gráficos, é possível observar que quando se aumenta o valor de w_F e se reduz w_T , a melhor geometria para o primeiro bloco, H_1/L_1 , deixa de ser a geometria com o maior valor, 1.55, como observado na Fig. 5.21 (a) e (b), obtém-se $(H_1/L_1)_{o,M} = 1.3$ para $w_T = w_F = 0.5$ e cai para $(H_1/L_1)_{o,M} = 0.0625$ para $w_T = 0.1$ e $w_F = 0.9$. Nota-se que os casos com ebulição tiveram um desempenho superior aos casos sem ebulição em todas as combinações de pesos definidas. Também, é possível observar que a vantagem dos casos com ebulição diminui consideravelmente à medida que se aumenta o valor do peso do objetivo fluidodinâmico. Provavelmente, esse comportamento se deve a maior sensibilidade da consideração da ebulição sobre a taxa de transferência de calor do que sobre a perda de carga. Também é possível observar que a consideração ou não da ebulição altera as razões ótimas de H_1/L_1 e o efeito dessa razão sobre C_b para as ponderações intermediárias. Uma das possíveis causas das diferenças existentes no efeito da geometria dos blocos sobre C_b entre os casos com e sem ebulição é a existência do calor latente no processo de ebulição, que pode ter proporcionado o maior desempenho multi-objetivo e térmico observado para as geometrias ótimas nos casos com ebulição, onde estas obtiveram menores valores das razões dos blocos nos casos com w_T menor e igual a 0.5.

O efeito de H_2/L_2 sobre $(C_b)_{1,max}$ para $L_3/L_{ocu} = 0.5$ é mostrado na Figura 5.22 para os diferentes pesos considerados. É possível observar a evolução do efeito de H_2/L_2 a medida que são variados os pesos w_F e w_T . Nota-se uma diferença considerável no efeito de H_2/L_2 sobre C_b e nas melhores geometrias (pontos máximos) ao comparar-se os casos com e sem ebulição quando $w_T = 0.9$ e $w_F = 0.1$ e também em $w_T = 0.7$ e $w_F = 0.3$. Também é possível observar a vantagem dos casos com ebulição em relação aos casos sem ebulição. A Figura 5.23 mostra o efeito de H_2/L_2 sobre a razão H_1/L_1 uma vez otimizada, $(H_1/L_1)_o$, também para $L_3/L_{ocu} = 0.5$ e para os diferentes pesos considerados e para os casos com e sem ebulição. É possível verificar que há diferenças consideráveis no efeito de H_2/L_2 sobre $(H_1/L_1)_o$ em $w_T = 0.7$ e $w_F = 0.3$ quando H_2/L_2 varia entre 0.6 e 1.55. Também, pequenas diferenças são observadas para $w_T = 0.5$ e $w_F = 0.5$ e para $w_T = 0.3$ e $w_F = 0.7$ e quando H_2/L_2 é menor do que 0.45 para $w_T = 0.1$ e $w_F = 0.9$.

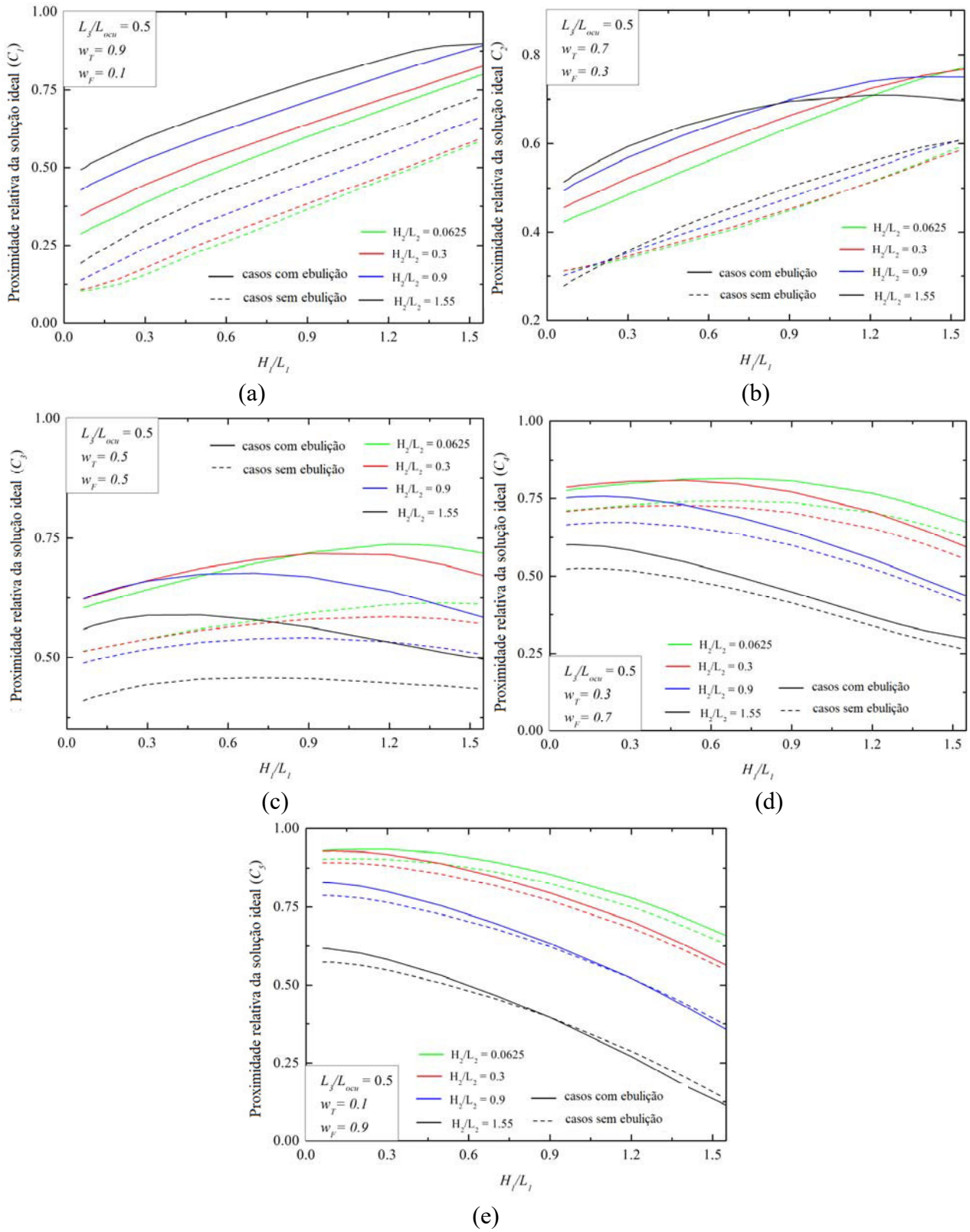


Figura 5.21 – Efeito de H_1/L_1 sobre a proximidade relativa da solução ideal (C_b) para diferentes H_2/L_2 , $L_3/L_{ocu} = 0.5$ e para: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$.

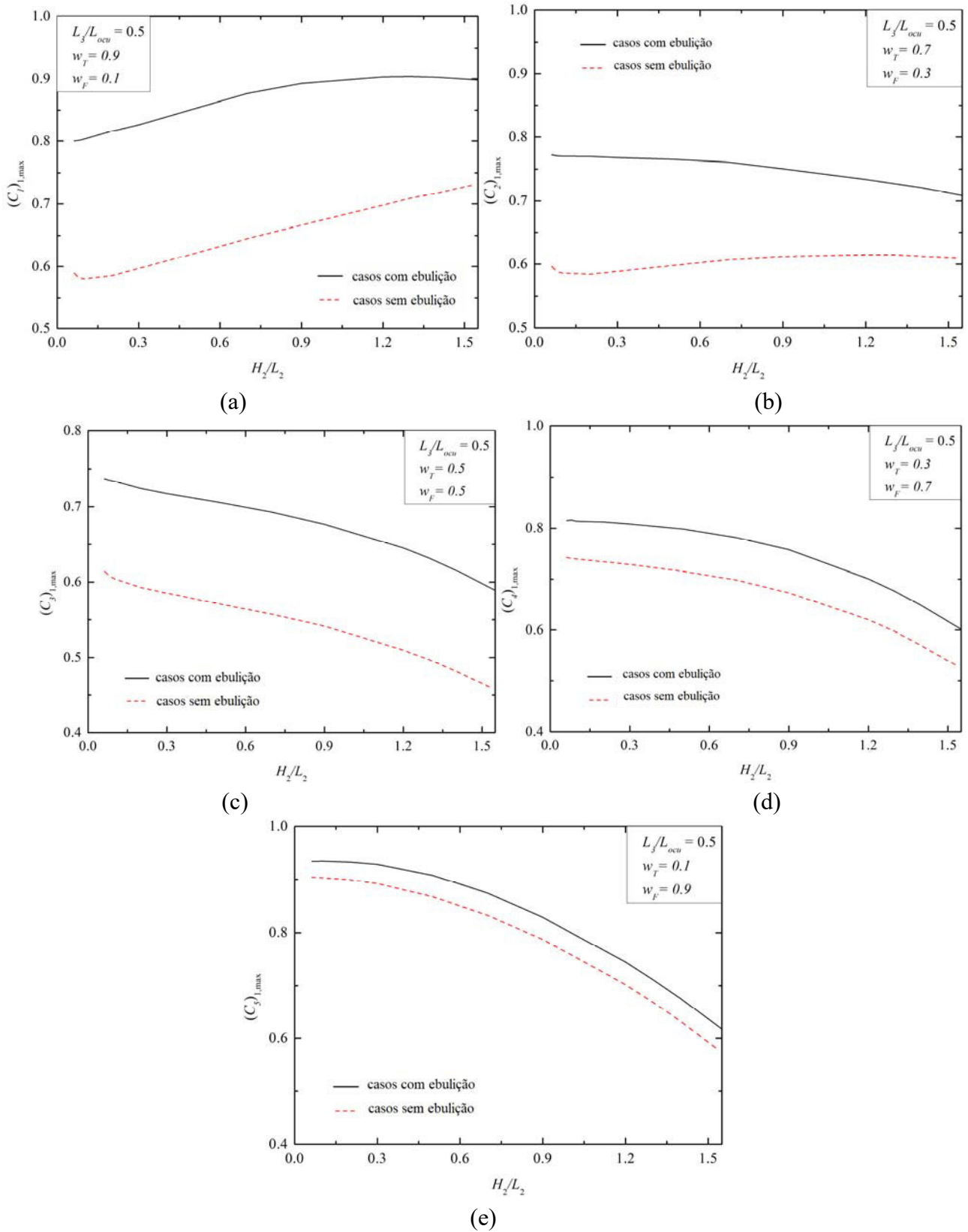


Figura 5.22 – Efeito de H_2/L_2 sobre a proximidade relativa da solução ideal uma vez maximizada $(C_b)_{1,\max}$, para $L_3/L_{ocu} = 0.5$ e para: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$.

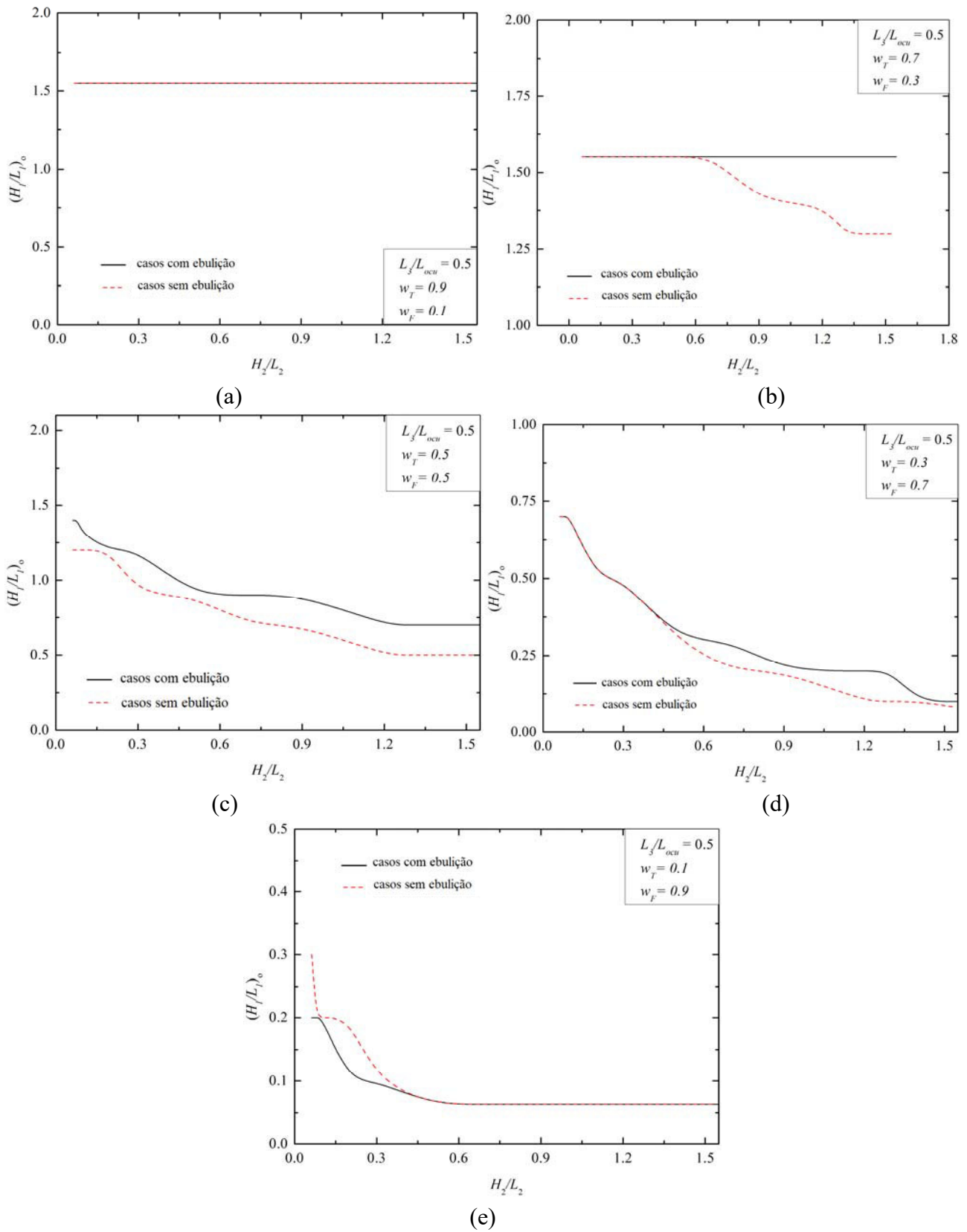


Figura 5.23 – Efeito de H_2/L_2 sobre a razão H_1/L_1 uma vez otimizada, $(H_1/L_1)_o$, para $L_3/L_{ocu} = 0.5$ e para: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$.

A Figura 5.24 mostra o efeito de L_3/L_{ocu} sobre $(C_b)_{2,max}$, a proximidade relativa da solução ideal duas vezes maximizada. Em geral é possível observar que L_3/L_{ocu} causa um efeito maior em $(C_b)_{2,max}$ quando $w_T = w_F = 0.5$, sendo que $L_3/L_{ocu} = 0.625$ possui um $(C_b)_{2,max}$ aproximadamente 3.2 % maior do que $L_3/L_{ocu} = 0.375$. Esta diferença tende a diminuir quando se aumenta um dos pesos e se diminui o outro. Também é possível observar que o processo de ebulição influi praticamente apenas na magnitude do efeito de L_3/L_{ocu} sobre $(C_b)_{2,max}$. As Figuras 5.25 e 5.26 mostram os efeitos de L_3/L_{ocu} sobre as geometrias ótimas multi-objetivo, $(H_1/L_1)_{oo}$ e $(H_2/L_2)_o$, respectivamente. É possível observar que não houveram diferenças consideráveis entre os casos com e sem ebulição com apenas uma exceção, o efeito de L_3/L_{ocu} sobre $(H_2/L_2)_o$ quando $w_T = 0.7$ e $w_F = 0.3$ (Figura 5.26 b). Isto pode significar que, em geral, o processo de ebulição não altera drasticamente o efeito da geometria sobre o desempenho multi-objetivo, porém pode causar mudanças pontuais, dependendo do peso atribuído para cada objetivo, além de causar alterações de magnitude no desempenho. Para $w_T = 0.7$ e $w_F = 0.3$, houve uma grande diferença entre a geometria ótima observada para o segundo bloco. Nos casos sem ebulição houve uma preferência por um segundo bloco com razão H_2/L_2 próxima da máxima definida ($H_2/L_2 = 1.55$), enquanto que nos casos com ebulição a geometria ótima do segundo bloco resultante foi próxima do oposto, a geometria com o menor valor de H_2/L_2 ($H_2/L_2 = 0.0625$).

As Figuras 5.27-31 mostram os campos das geometrias ótimas multi-objetivo dos casos com ebulição para as diferentes combinações de pesos em análise enquanto que as Figuras 5.32-35 mostram os campos das geometrias ótimas multi-objetivo dos casos sem ebulição. É possível verificar que em ambos os casos as geometrias se transformam de modo que o aumento do valor de w_F (e a subsequente diminuição de w_T) faz com que as geometrias ótimas dos dois blocos resultem em uma diminuição das razões H_1/L_1 e H_2/L_2 , em conformidade com os resultados obtidos nas análises térmica e fluidodinâmica, onde as geometrias ótimas obtidas foram as que possuíam os maiores e os menores valores das duas razões, respectivamente. Através dos campos das geometrias ótimas, também é possível observar que não houveram diferenças significativas no comportamento dos campos comparando-se os casos com e sem ebulição, apesar de ser possível observar diferenças na magnitude da temperatura, velocidade e pressão entre as duas situações.

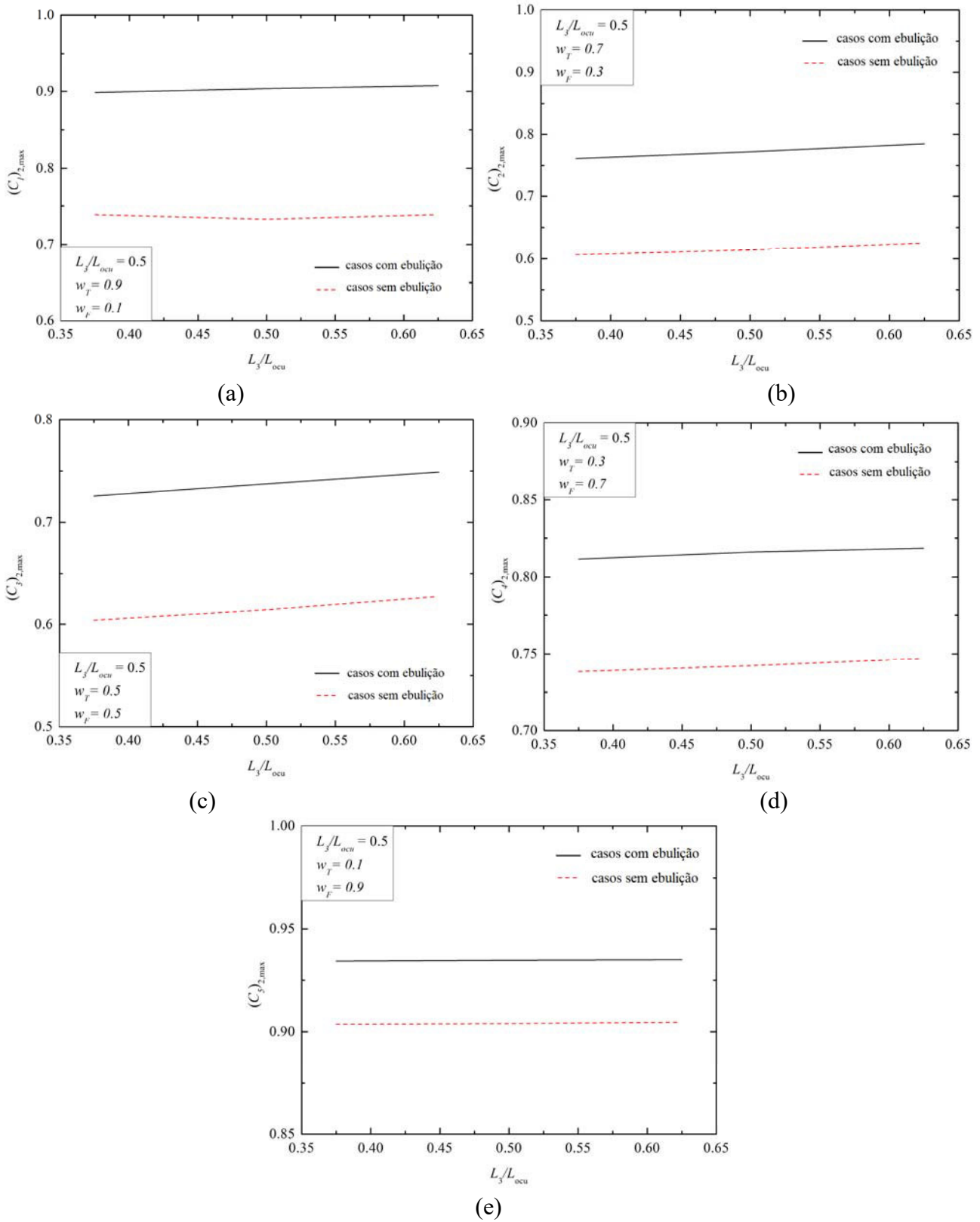


Figura 5.24 – Efeito de L_3/L_{ocu} sobre a proximidade relativa da solução ideal duas vezes maximizada $(C_b)_{2,max}$, $L_3/L_{ocu} = 0.5$ e para: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$.

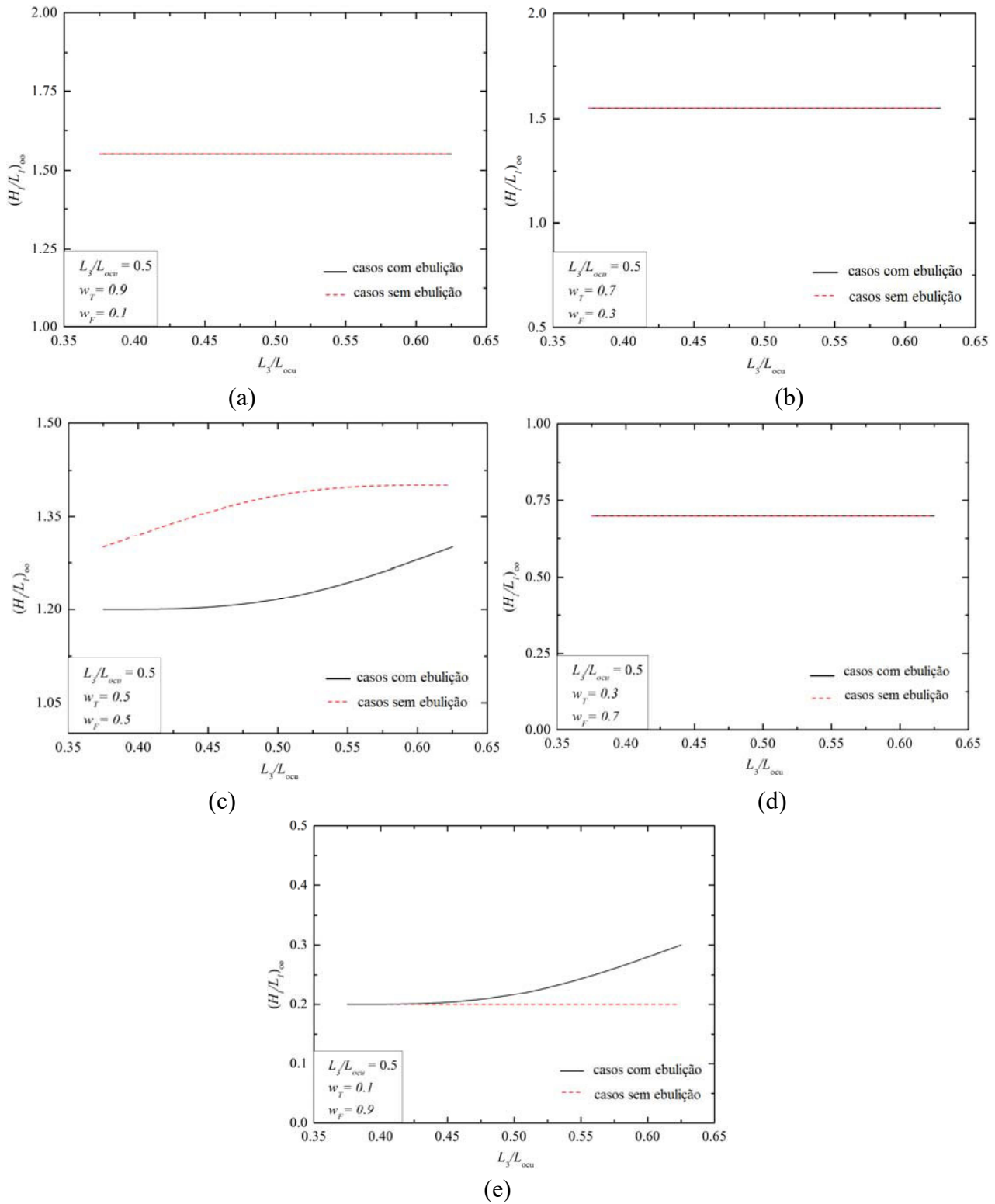


Figura 5.25 – Efeito de L_3/L_{ocu} sobre a razão H_1/L_1 duas vezes otimizada, $(H_1/L_1)_{\infty}$, $L_3/L_{ocu} = 0.5$ e para: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$.

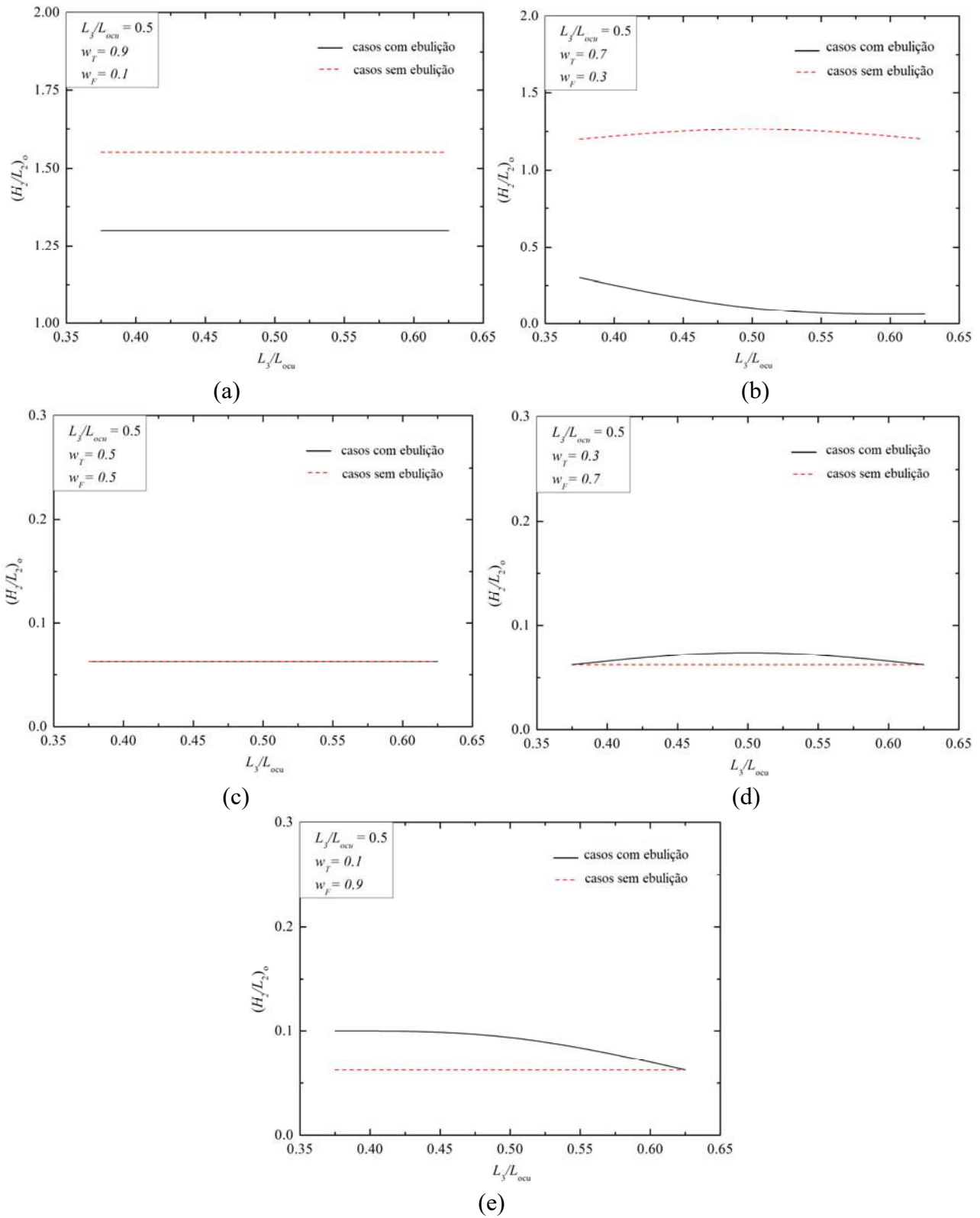


Figura 5.26 – Efeito de L_3/L_{ocu} sobre a razão H_2/L_2 duas vezes otimizada, $(H_2/L_2)_o$, L_3/L_{ocu} e: a) $w_T = 0.9$ e $w_F = 0.1$; b) $w_T = 0.7$ e $w_F = 0.3$; c) $w_T = w_F = 0.5$; d) $w_T = 0.3$ e $w_F = 0.7$; e) $w_T = 0.1$ e $w_F = 0.9$.

A Tabela 5.1 apresenta um resumo das geometrias ótimas obtidas na análise multi-objetivo e a Tabela 5.2 apresenta o resumo dos resultados obtidos. A Tabela 5.1 mostra a diferença observada entre as geometrias ótimas para $w_T = 0.7$ e $w_F = 0.3$ e também que para as demais combinações de pesos dos objetivos as geometrias ótimas dos casos com e sem ebulição foram iguais ou relativamente próximas. É possível verificar pela Tabela 5.2 que a proximidade relativa da solução ideal três vezes maximizada, $(C_b)_{3,\max}$, é 22.87 % superior para o caso com ebulição em relação ao caso sem ebulição quando $w_T = 0.9$ e $w_F = 0.1$, sendo que esta superioridade aumenta e chega a 25.55 % quando $w_T = 0.7$ e $w_F = 0.3$ porém diminui à medida que se aumenta w_F e é igual a 3.35 % quando $w_T = 0.1$ e $w_F = 0.9$. Além disso, é notável que em todas as combinações de pesos dos objetivos apresentadas, os casos com ebulição tiveram não só um melhor desempenho multi-objetivo, mas também um melhor desempenho térmico (maior taxa de transferência de calor) e um melhor desempenho fluidodinâmico (menor diferença de pressão).

Tabela 5.1 – Geometrias ótimas multi-objetivo.

Combinações de pesos dos objetivos	Casos com ebulição			Casos sem ebulição		
	$(H_1/L_1)_{000}$	$(H_2/L_2)_{00}$	$(L_3/L_{ocu})_0$	$(H_1/L_1)_{000}$	$(H_2/L_2)_{00}$	$(L_3/L_{ocu})_0$
$w_T = 0.9$ e $w_F = 0.1$	1.55	1.3	0.625	1.55	1.55	0.625
$w_T = 0.7$ e $w_F = 0.3$	1.55	0.0625	0.625	1.55	1.2	0.625
$w_T = w_F = 0.5$	1.3	0.0625	0.625	1.4	0.0625	0.625
$w_T = 0.3$ e $w_F = 0.7$	0.7	0.0625	0.625	0.7	0.0625	0.625
$w_T = 0.1$ e $w_F = 0.9$	0.3	0.0625	0.625	0.2	0.0625	0.625

Tabela 5.2 – Resultados obtidos das geometrias ótimas na análise multi-objetivo.

Combinações de pesos dos objetivos	Casos com ebulição			Casos sem ebulição		
	q (W)	ΔP (Pa)	$(C_b)_{3,\max}$	q (W)	ΔP (Pa)	$(C_b)_{3,\max}$
$w_T = 0.9$ e $w_F = 0.1$	96,817.46	3,685.25	0.9074	85,431.45	3,868.94	0.7385
$w_T = 0.7$ e $w_F = 0.3$	88,727.97	2,713.42	0.7848	83,338.67	3,554.32	0.6251
$w_T = w_F = 0.5$	84,441.17	2,519.30	0.7488	74,459.89	2,653.57	0.6273
$w_T = 0.3$ e $w_F = 0.7$	73,464.57	2,193.75	0.8185	60,839.53	2,248.50	0.7469
$w_T = 0.1$ e $w_F = 0.9$	65,344.92	2,046.23	0.9349	49,028.16	2,073.59	0.9046

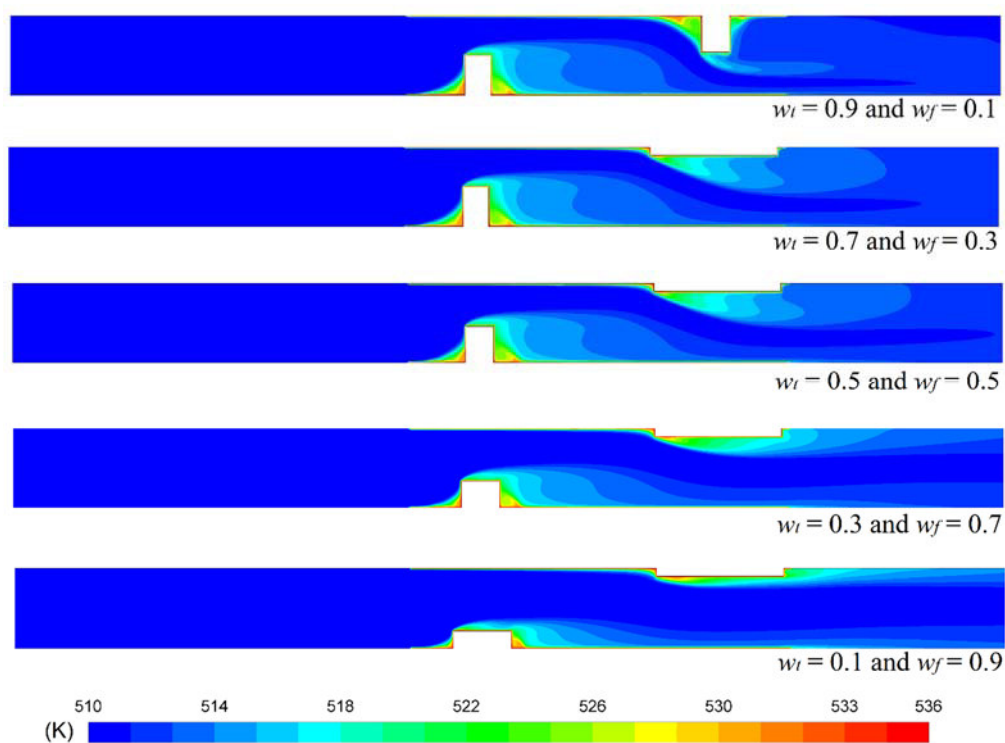


Figura 5.27 – Campos de temperatura (K) para as geometrias ótimas multi-objetivo com ebulição para diferentes combinações de pesos dos objetivos.

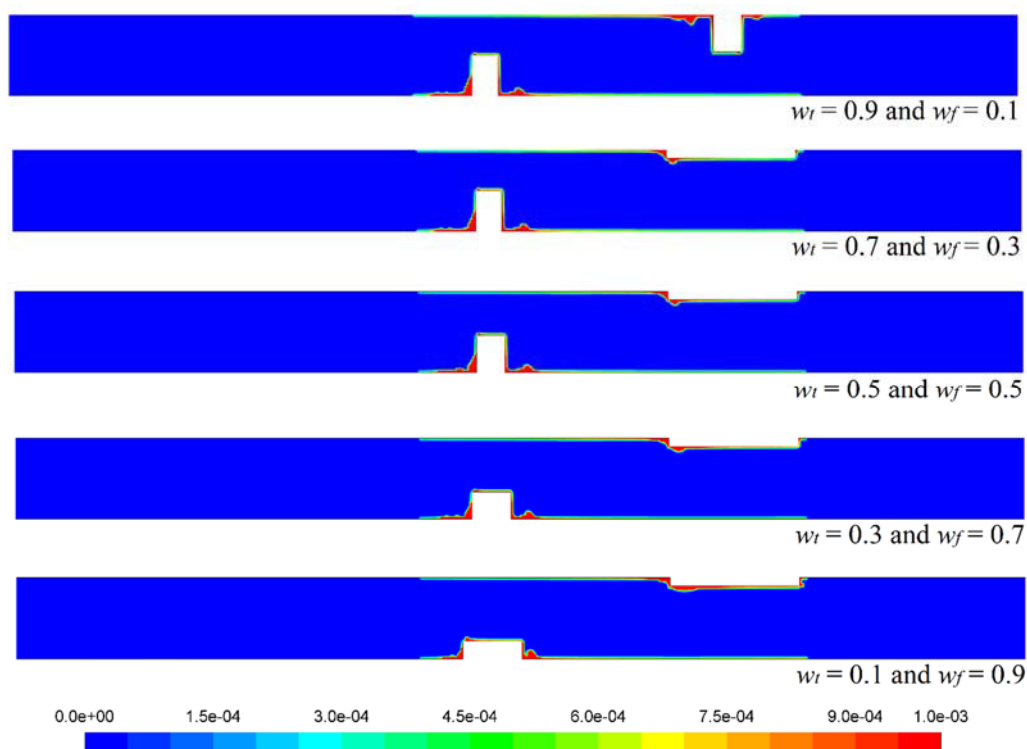


Figura 5.28 – Campos de fração de volume para as geometrias ótimas multi-objetivo com ebulição para diferentes combinações de pesos dos objetivos.

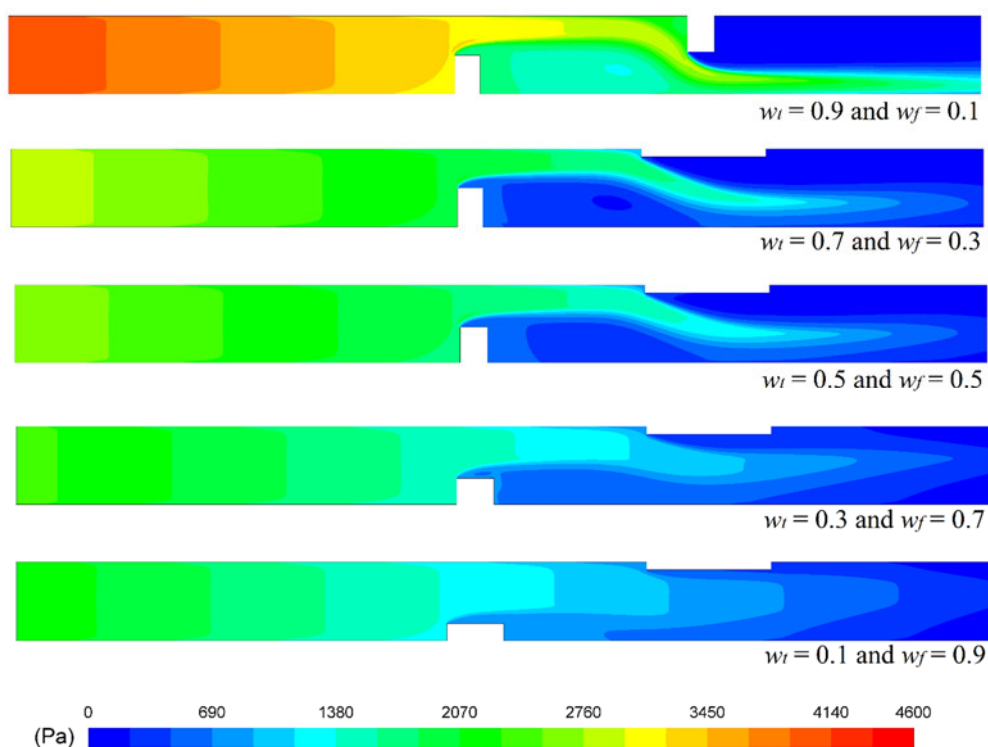


Figura 5.29 – Campos de pressão (Pa) para as geometrias ótimas multi-objetivo com ebulição para diferentes combinações de pesos dos objetivos.

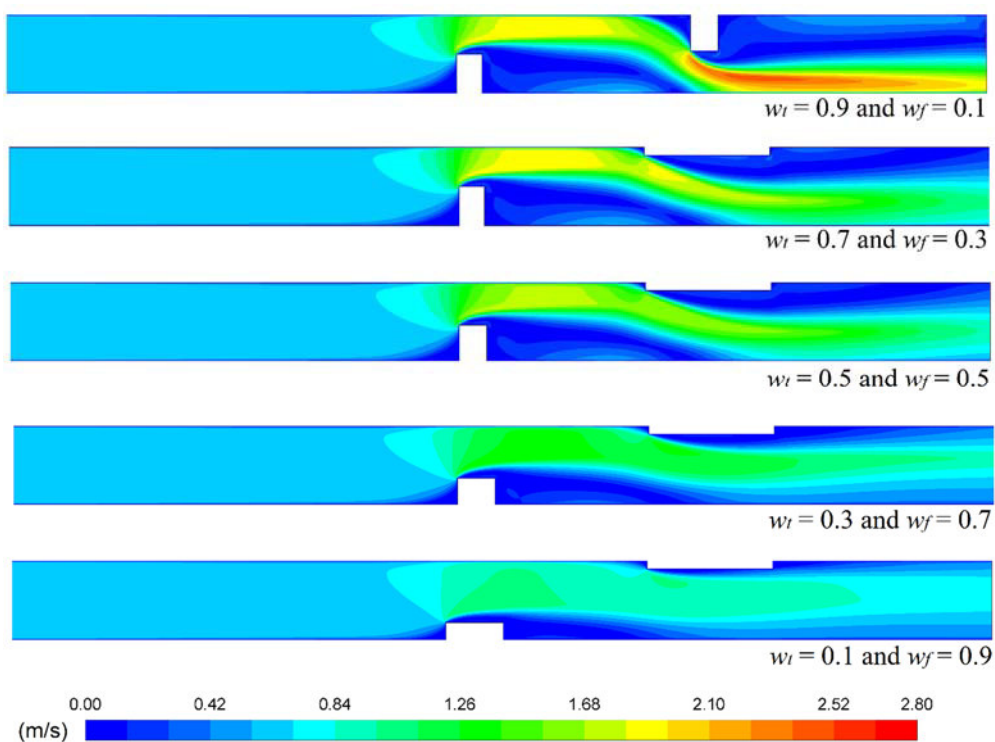


Figura 5.30 – Campos de velocidade (m/s) para as geometrias ótimas multi-objetivo com ebulição para diferentes combinações de pesos dos objetivos.



Figura 5.31 – Campos de linhas de fluxo para as geometrias ótimas multi-objetivo com ebulição para diferentes combinações de pesos dos objetivos.

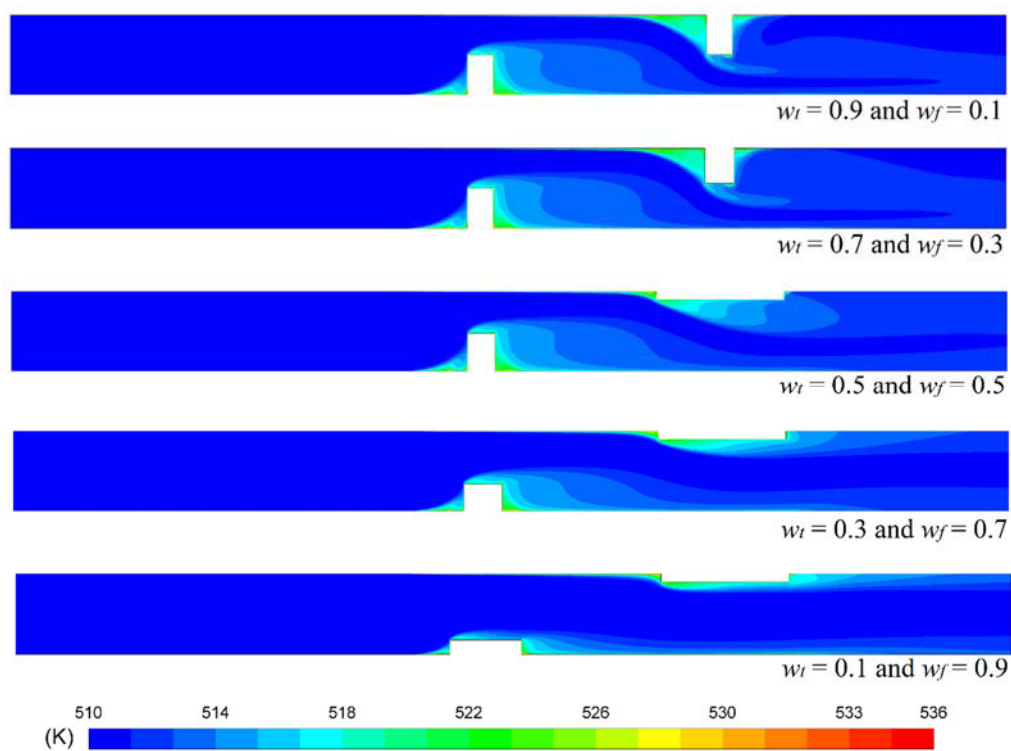


Figura 5.32 – Campos de temperatura (K) para as geometrias ótimas multi-objetivo sem ebulição para diferentes combinações de pesos dos objetivos.

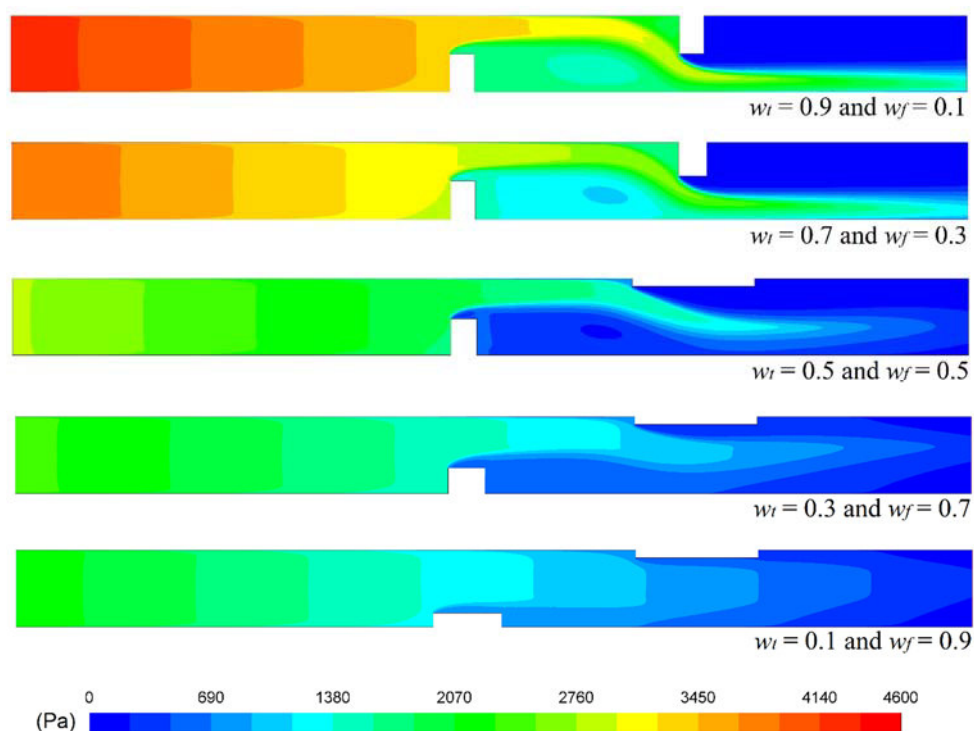


Figura 5.33 – Campos de pressão (Pa) para as geometrias ótimas multi-objetivo sem ebulição para diferentes combinações de pesos dos objetivos.

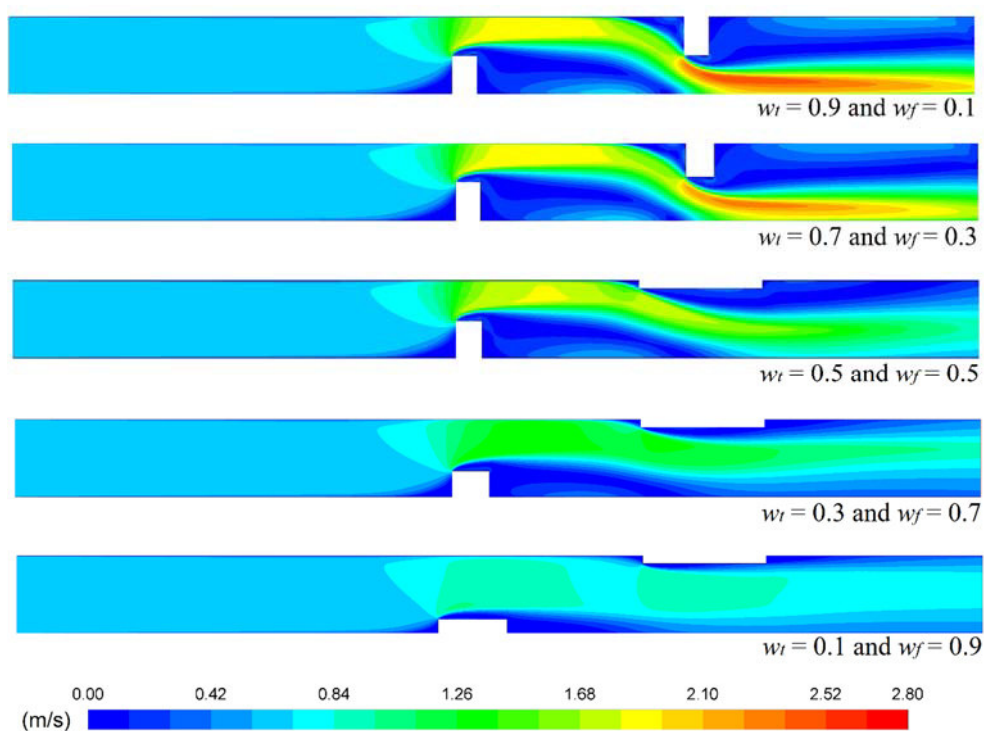


Figura 5.34 – Campos de velocidade (m/s) para as geometrias ótimas multi-objetivo sem ebulição para diferentes combinações de pesos dos objetivos.



Figura 5.35 – Campos de linhas de fluxo para as geometrias ótimas multi-objetivo sem ebulição para diferentes combinações de pesos dos objetivos.

5.2.4. Análise do Coeficiente de Desempenho (*COP*)

Além da análise multi-objetivo realizada na seção anterior através da aplicação do método TOPSIS, foi realizada uma avaliação geométrica considerando-se como objetivo a maximização do coeficiente de desempenho (*COP*), que é um indicador de desempenho comumente utilizado em equipamentos térmicos como aquecedores, refrigeradores e sistemas de ar condicionado. Esta análise seguiu os mesmos passos das análises anteriores apresentadas neste trabalho. A Figura 5.36 apresenta o efeito de H_1/L_1 sobre o coeficiente de desempenho (*COP*) para diferentes valores de H_2/L_2 . É possível verificar que H_1/L_1 possui diferentes comportamentos para H_2/L_2 diferentes e também que o processo de ebulição altera o efeito de H_1/L_1 sobre o *COP*. Além disso, assim como observado nas análises anteriores, observa-se que o processo de ebulição causa um aumento considerável no coeficiente de desempenho ao longo de todo o intervalo de H_1/L_1 . A Figura 5.37 mostra o efeito de H_2/L_2 sobre o coeficiente de desempenho uma vez maximizado $(COP)_{1,max}$, onde pode-se observar que o aumento do valor de H_2/L_2 causa uma queda no $(COP)_{1,max}$, de forma semelhante para os casos com e sem ebulição. O efeito da geometria do segundo bloco, H_2/L_2 , sobre a geometria uma vez otimizada do primeiro bloco, $(H_1/L_1)_o$, é apresentada na Figura 5.38, sendo visível que os casos sem ebulição tem um $(H_1/L_1)_o$ maior para todos os valores de H_2/L_2 comparando com os casos com ebulição.

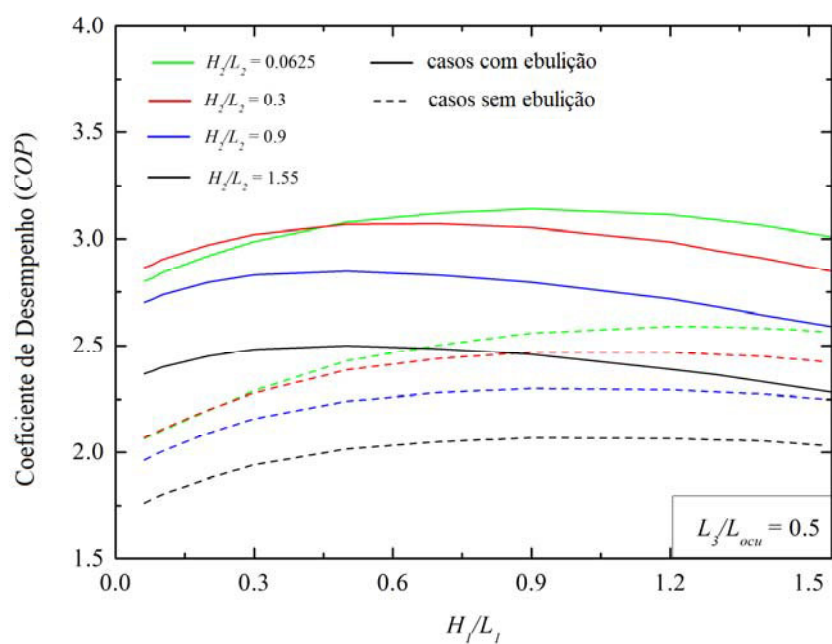


Figura 5.36 – Efeito de H_1/L_1 sobre o coeficiente de desempenho (COP) para diferentes valores de H_2/L_2 .

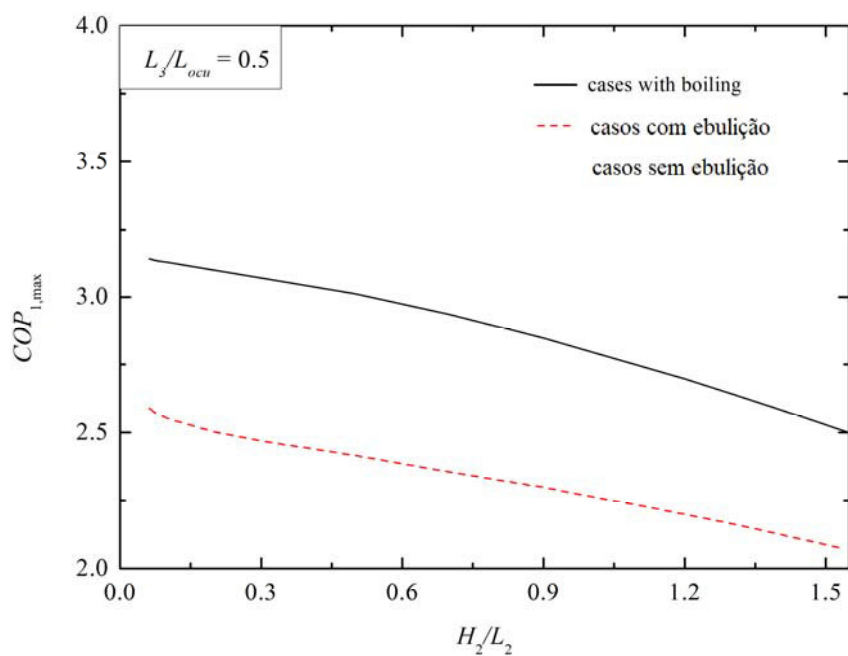


Figura 5.37 – Efeito de H_2/L_2 sobre o coeficiente de desempenho uma vez maximizado $(COP)_{1,max}$.

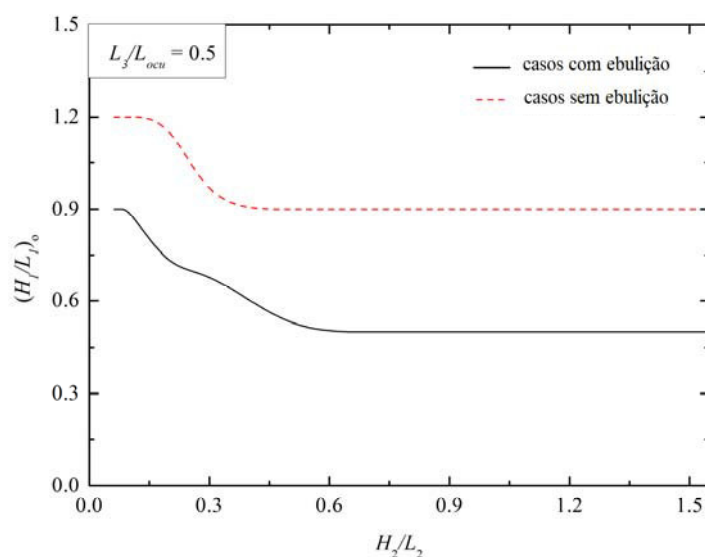


Figure 5.38 – Efeito de H_2/L_2 sobre a razão H_1/L_1 uma vez otimizada, $(H_1/L_1)_0$.

O efeito de L_3/L_{ocu} sobre o coeficiente de desempenho duas vezes maximizado, $(COP)_{2,max}$, pode ser visto na Figura 5.39. A variação do valor de L_3/L_{ocu} não causou alterações significativas no $(COP)_{2,max}$. É possível observar a vantagem considerável dos casos com ebulição em relação aos casos sem ebulição. Apesar desta diferença na magnitude do COP , o processo de ebulição manteve o efeito de L_3/L_{ocu} sobre $(COP)_{2,max}$ com a tendência de um ligeiro aumento do COP com o aumento de L_3/L_{ocu} .

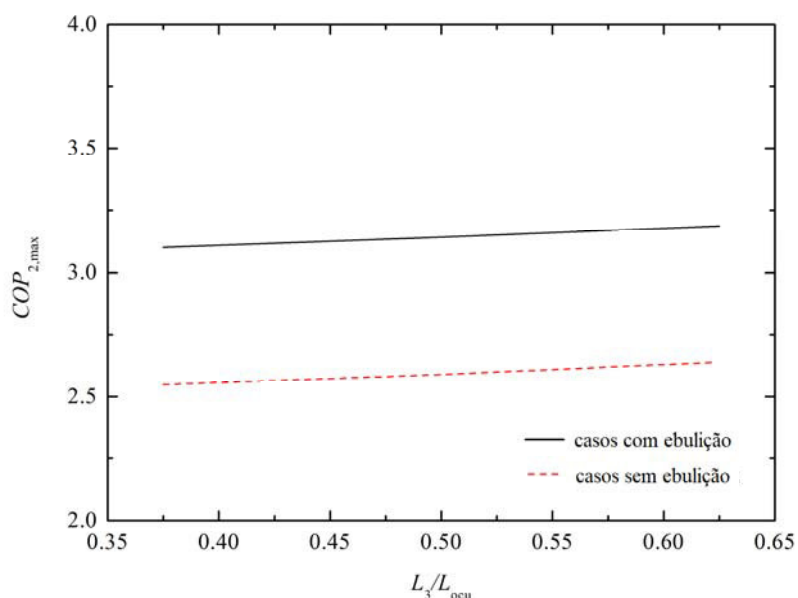


Figura 5.39 – Efeito de L_3/L_{ocu} sobre o coeficiente de desempenho duas vezes maximizado $(COP)_{2,max}$.

Os campos das geometrias ótimas são apresentados na Figura 5.40 (casos com ebulição) e Figura 5.41 (casos sem ebulição). Assim como verificado na análise multi-objetivo apresentada na seção anterior, é possível observar que no caso com ebulição há uma maior temperatura do fluido próximo a região das paredes dos blocos comparando ao caso sem ebulição. Esta maior temperatura observada nos casos com ebulição condiz com o maior coeficiente de desempenho obtido nestes casos. Entre os casos simulados com ebulição, a geometria ótima obtida foi igual a $(H_1/L_1)_{ooo} = 0.9$, $(H_2/L_2)_{oo} = 0.0625$ e $(L_3/L_{ocu})_o = 0.625$ e gerou um coeficiente de desempenho três vezes maximizado igual a $(COP)_{3,max} = 3.185$. Nos casos sem ebulição, a geometria ótima foi $(H_1/L_1)_{ooo} = 1.3$, $(H_2/L_2)_{oo} = 0.0625$ e $(L_3/L_{ocu})_o = 0.625$ e resultou num $(COP)_{3,max} = 2.64$. Portanto, a melhor geometria entre os casos com ebulição foi capaz de gerar um coeficiente de rendimento 20.64 % maior do que a geometria ótima dos casos sem ebulição. Ao comparar os resultados obtidos nesta avaliação geométrica através do coeficiente de desempenho, é possível verificar que os resultados apresentam uma proximidade considerável dos resultados obtidos na análise multi-objetivo quando os dois pesos são iguais ($w_T = w_F = 0.5$), tanto no efeito da geometria sobre o COP e sobre C_b como nas diferenças observadas pela presença do processo de ebulição. Isto pode ser um resultado da forma como o Coeficiente de Desempenho é calculado, uma vez que a equação que calcula o COP (equação 2.26) é diretamente proporcional a taxa de transferência de calor (o objetivo a ser maximizado) e inversamente proporcional a queda de pressão ao longo do canal (objetivo a ser minimizado), além de também conter o fluxo de massa que entra no canal, que é constante.

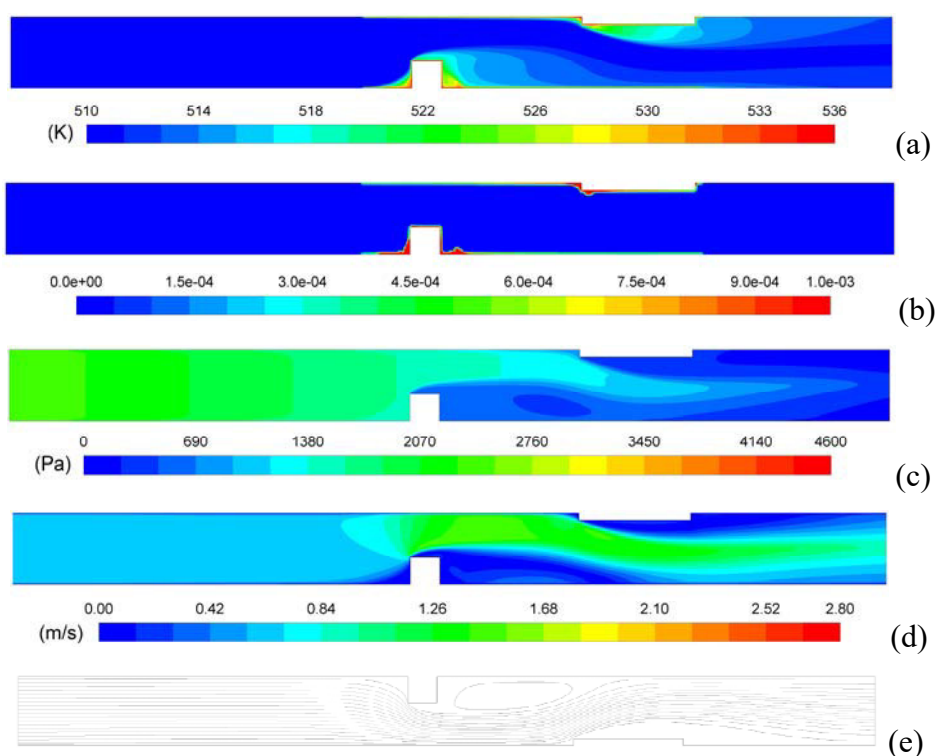


Figura 5.40 – Campos de temperatura (K) (a), fração de volume (b), pressão (Pa) (c), velocidade (m/s) (d) e linhas de fluxo (e) para as geometrias ótimas com ebulição, $(H_1/L_1)_{ooo} = 0.9$, $(H_2/L_2)_{oo} = 0.0625$ e $(L_3/L_{ocu})_o = 0.625$.

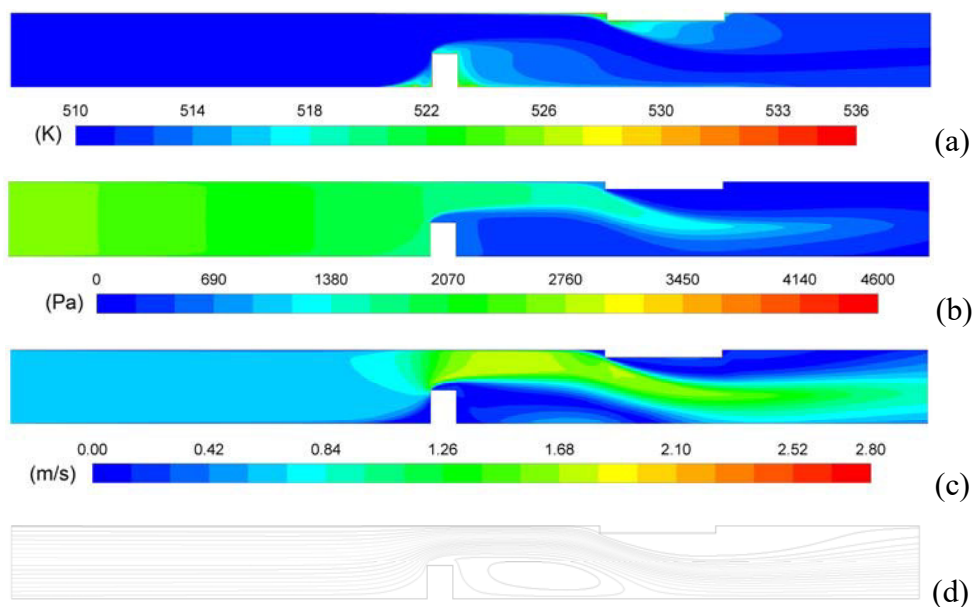


Figura 5.41 – Campos de temperatura (K) (a), pressão (Pa) (b), velocidade (m/s) (c) e linhas de fluxo (d) para as geometrias ótimas sem ebulição, $(H_1/L_1)_{ooo} = 1.3$, $(H_2/L_2)_{oo} = 0.0625$ e $(L_3/L_{ocu})_o = 0.625$.

6. CONCLUSÕES

A presente tese é um estudo sobre a avaliação geométrica de canais com a presença de dois blocos aquecidos sujeitos a escoamentos no regime turbulento com e sem ebulição. O problema foi solucionado numericamente através do software Ansys Fluent 18.1. O processo de avaliação geométrica foi realizado através da aplicação do método Design Construtal associado ao método de Busca Exaustiva e ao método multi-critério TOPSIS. Em relação a geometria do canal, foram definidos dois graus de liberdade: a razão entre a altura e a largura de cada bloco, H_1/L_1 e H_2/L_2 . Além dos dois graus de liberdade também foi investigado o efeito da razão entre a distância entre os centros dos blocos e o comprimento de ocupação, L_3/L_{ocu} . Foram definidas três restrições a geometria: a área do canal (A), a área de ocupação de cada bloco (A_1 , igual para os dois blocos) e a área dos blocos (A_2 , igual para os dois blocos). Os objetivos da avaliação geométrica eram dois: maximizar a troca térmica entre os blocos aquecidos e o escoamento e minimizar a diferença de pressões entre a entrada e a saída do canal, ou seja, um objetivo térmico e um fluidodinâmico. O método TOPSIS possibilitou uma análise multi-objetiva do problema, levando em consideração os dois objetivos propostos através da definição de pesos para estes. A influência do processo de ebulição sobre os objetivos também foi investigada tendo sido simulados os mesmos casos com e sem o processo de ebulição habilitado.

O modelo numérico utilizado no presente estudo para a simulação dos casos de escoamentos convectivos internos com ebulição foi validado através da comparação com outros modelos numéricos, resultados experimentais e correlações empíricas disponíveis em trabalhos realizados por outros autores. Foram empregados o modelo multifásico Euleriano e o modelo de ebulição RPI (criado no *Rensselaer Polytechnic Institute*). Tendo sido comparados os resultados da fração de volume de vapor, da temperatura do escoamento e da temperatura nas paredes do canal, foi possível concluir que o modelo é adequado para simular estes tipos de escoamentos e que possui boa concordância com os resultados de outros autores.

Foi realizada a avaliação geométrica do canal com dois blocos aquecidos considerando três valores para L_3/L_{ocu} , 0.375, 0.5 e 0.625. A partir dos resultados obtidos, foi possível observar que as geometrias ótimas foram aquelas em que as razões dos dois blocos, H_1/L_1 e H_2/L_2 , apresentaram os maiores valores dentre os definidos neste trabalho (blocos possuem a maior intrusão possível no canal) considerando-se apenas o objetivo térmico e os menores valores (blocos possuem a menor intrusão possível no canal) considerando-se apenas o objetivo fluidodinâmico. Esses resultados estão dentro do esperado e de acordo com os conceitos básicos de transferência de calor e dinâmica dos

fluidos.

Em relação a análise térmica da avaliação geométrica, pode-se observar um aumento quase linear da taxa de transferência de calor com o aumento do valor dos dois graus de liberdade. A geometria ótima térmica foi a mesma tanto para os casos com ebulição como para os casos sem ebulição, sendo esta a geometria onde os dois blocos tem a maior altura possível e a menor distância entre eles dentro dos valores definidos no presente trabalho. Esta geometria gerou uma taxa de transferência de calor três vezes maximizada de $(q)_{3,\max} = 98,629.32 \text{ W}$ para os casos com ebulição e $(q)_{3,\max} = 85,619.89 \text{ W}$ para os casos sem ebulição. Isto significa que a geometria ótima tem um desempenho térmico da 15.19 % superior quando o processo de ebulição é habilitado. Foi possível observar que não há diferenças significativas do efeito da geometria sobre a taxa de transferência de calor ao comparar-se os casos com e sem ebulição.

A análise fluidodinâmica da avaliação geométrica foi realizada a partir dos resultados obtidos para a diferença de pressão entre a entrada e a saída do canal. Foi observado que o aumento das razões H_1/L_1 e H_2/L_2 provoca um aumento da queda de pressão. A geometria onde os blocos têm a menor intrusão no canal e a maior distância entre eles foi a que apresentou o melhor desempenho fluidodinâmico, sendo novamente igual para as duas situações, com e sem ebulição. Esta geometria é a que melhor facilita o escoamento do fluido e é completamente oposta a geometria ótima térmica. Também, foi observado que os casos com ebulição proporcionam um melhor desempenho fluidodinâmico, ou seja, uma menor queda de pressão ao longo do canal. Assim como observado na análise térmica, não há diferenças consideráveis dos efeitos de H_1/L_1 e H_2/L_2 sobre a queda de pressão ao longo do canal ao comparar-se os casos com e sem ebulição. A geometria ótima fluidodinâmica obtida para os casos com e sem ebulição foi $(H_1/L_1)_{\text{oo},F} = 0.0625$, $(H_2/L_2)_{\text{oo},F} = 0.0625$ e $(L_3/L_{\text{ocu}})_{\text{o},T} = 0.625$ e resultou numa queda de pressão três vezes minimizada igual à $(\Delta P)_{3,\min} = 1,993.80 \text{ Pa}$ no caso com ebulição e $(\Delta P)_{3,\min} = 2,047.48$ no caso sem ebulição. Portanto, a geometria ótima fluidodinâmica apresentou uma queda de pressão 2.62 % menor quando o processo de ebulição foi considerado em comparação quando este não foi considerado.

Uma conclusão importante possível de ser feita no presente trabalho é que os resultados obtidos nas análises térmica e fluidodinâmica demonstraram que, de uma forma geral, o regime de escoamento turbulento não teve influência significativa sobre o comportamento dos efeitos de H_1/L_1 e H_2/L_2 sobre os desempenhos térmico e fluidodinâmico quando comparado aos resultados obtidos em um estudo semelhante onde foi aplicado o regime de escoamento laminar (Feijó et al., 2018).

A análise multi-objetivo foi realizada a partir da aplicação do método multi-critério TOPSIS,

onde foram considerados pesos para os objetivos térmico (w_T) e fluidodinâmico (w_F). Para cada uma das combinações de pesos foi calculado o parâmetro C_b , a proximidade relativa da solução ideal, de cada uma das geometrias simuladas. No presente trabalho, a análise multi-objetivo considerou cinco combinações de pesos diferentes: $w_T = 0.9$ e $w_F = 0.1$ (C_1), $w_T = 0.7$ e $w_F = 0.3$ (C_2), $w_T = w_F = 0.5$ (C_3), $w_T = 0.3$ e $w_F = 0.7$ (C_4) e $w_T = 0.1$ e $w_F = 0.9$ (C_5).

Nos casos com o processo de ebulição habilitado, foi possível observar que quando se aumenta o valor de w_F e consequentemente se reduz w_T , a melhor geometria para o primeiro bloco (H_1/L_1) tem uma tendência de diminuição da sua altura, que deixa de ser a geometria com o maior valor, 1.55, como observado para $w_T = 0.9$ e $w_F = 0.1$, fica igual a 1.3 para $w_T = w_F = 0.5$ e cai para 0.0625 para $w_T = 0.1$ e $w_F = 0.9$. Também nos casos com ebulição, o efeito de H_2/L_2 sobre $(C_b)_{1,\max}$ mostrou um comportamento de forma que o desempenho multi-objetivo das geometrias com os menores valores de H_2/L_2 também aumentou à medida que se aumentou w_F . Porém, uma diferença significativa foi notada em relação ao que se observou com H_1/L_1 . Neste caso, a melhor geometria do segundo bloco (H_2/L_2) passa do maior valor, 1.55, obtido com a combinação $w_T = 0.9$ e $w_F = 0.1$, para o menor valor, 0.0625, em $w_T = 0.7$ e $w_F = 0.3$, ou seja, com apenas com um pequeno incremento de w_F houve uma grande diferença na geometria ótima do segundo bloco.

Em relação aos casos sem ebulição, o efeito da geometria dos dois blocos sobre a proximidade relativa da solução ideal, C_b , foi parecido ao observado nos casos com ebulição. No entanto, uma grande diferença foi notada na geometria ótima do segundo bloco quando $w_T = 0.7$ e $w_F = 0.3$. Nos casos sem ebulição foi observada uma geometria ótima igual a $H_2/L_2 = 1.2$, próxima do máximo valor definido ($H_2/L_2 = 1.55$), enquanto que nos casos com ebulição a geometria ótima do segundo bloco foi o oposto, a geometria com o menor valor de H_2/L_2 ($H_2/L_2 = 0.0625$). Foi possível verificar que os casos sem ebulição tiveram um desempenho multi-objetivo inferior aos casos com ebulição para todas as combinações de pesos, porém esta desvantagem tende a diminuir consideravelmente à medida que se aumenta o peso do objetivo fluidodinâmico. Quando $w_T = 0.9$ e $w_F = 0.1$, a geometria ótima dos casos com ebulição tem um desempenho multi-objetivo 22.87 % superior a geometria ótima dos casos sem ebulição. Esta diferença aumenta para 25.55 % quando $w_T = 0.7$ e $w_F = 0.3$ e, a partir deste ponto, o aumento de w_F faz com que esta superioridade dos casos com ebulição caia para 3.35 % quando $w_T = 0.1$ e $w_F = 0.9$. Além disso, foi possível observar que os casos com ebulição apresentaram não somente um melhor desempenho multi-objetivo para todas as combinações de pesos, mas também tiveram uma maior taxa de transferência de calor e uma menor queda de pressão ao longo do canal.

Em geral, é possível concluir que o processo de ebulição aumenta o desempenho multi-

objetivo das geometrias ótimas para qualquer combinação de pesos dos dois objetivos (térmico e fluidodinâmico) e que um maior peso do objetivo térmico (e consequentemente um menor peso do objetivo fluidodinâmico) tende a aumentar a vantagem proporcionada pelo processo de ebulição. Esta maior vantagem quando se considera um maior peso da transferência de calor possivelmente é resultado da presença do calor latente no processo de ebulição. Além disso, também é possível concluir que o processo de ebulição pode alterar consideravelmente a geometria ótima do canal dependendo da combinação de pesos dos objetivos definidas, como observado quando $w_T = 0.7$ e $w_F = 0.3$. Em relação a otimização geométrica da análise multi-objetivo, os resultados demonstraram que ao contrário das otimizações geométricas considerando apenas um objetivo, térmico ou fluidodinâmico, onde o canal possui geometrias ótimas em que os dois blocos possuem o mesmo aspecto, ao considerar-se um multi-objetivo é importante notar que a quantidade e também a posição de cada bloco dentro do canal podem influir, de forma que cada bloco poderá ter uma geometria ótima específica e diferente em relação aos outros, uma vez que tenha sido dada a devida liberdade para a sua evolução, como através da aplicação do método Design Construtal na presente tese. Uma análise da avaliação geométrica considerando o Coeficiente de Desempenho (*COP*) como objetivo apresentou resultados próximos aos obtidos na análise multi-objetivo quando $w_T = 0.5$ e $w_F = 0.5$, com ligeiras diferenças. Nesta análise, os casos com ebulição também demonstraram um desempenho melhor do que os casos sem ebulição, sendo que a geometria ótima dos casos com ebulição teve um desempenho 20.64 % maior do que a geometria ótima dos casos sem ebulição.

Além do problema proposto na seção 3.1 da presente tese, também foi realizado um estudo semelhante da avaliação geométrica de microcanais sujeitos a escoamentos convectivos com ebulição no regime laminar, onde os microcanais possuem dois blocos trapezoidais aquecidos. Foram empregados o modelo multifásico Mixture (Manninen et al., 1996) e o modelo de mudança de fase de Lee (Lee, 1980). Os resultados dos casos com ebulição também foram comparados com os resultados de casos sem ebulição para avaliar o efeito do processo de mudança de fase na geometria do microcanal. Foi verificado que, para as condições aplicadas no estudo, o processo de ebulição foi responsável por um aumento do desempenho multi-objetivo dos microcanais, porém não houve alterações significativas no processo de avaliação e otimização geométrica, tendo sido encontradas as mesmas geometrias ótimas tanto nos casos com ebulição como nos sem ebulição. Este trabalho é apresentado no Apêndice 1 da presente tese.

6.1. Proposta de Continuidade

Os resultados e conclusões obtidos nesta tese proporcionam uma reflexão e direcionam para

a proposição de novos estudos relacionados à pesquisa da otimização geométrica de canais com convecção e ebulição. São propostos os seguintes estudos a serem realizados no futuro:

- Avaliação geométrica considerando mais valores de L_3/L_{ocu} , sendo este considerado como um grau de liberdade;
- Investigação de diferentes formas geométricas dos blocos, como triangular e ondulada;
- Estudo da influência da turbulência na otimização geométrica de canais convectivos com ebulição, aplicando Números de Reynolds como 10,000, 20,000, 50,000 e 100,000;
- Aprimoramento do modelo numérico para possibilitar uma maior geração de vapor e geometrias mais complexas, assim como também para ser capaz de simular na microescala;
- Estudo da influência do arranjo geométrico de cilindros sujeitos a escoamentos convectivos externos com ebulição;
- Estudo dos efeitos do processo de condensação na otimização geométrica de canais com escoamentos convectivos.

7. REFERENCES

- ABDOUS, M.A.; SAFFARI, H.; AVVAL, H. B. The study of entropy generation during flow boiling in a micro-fin tube. **International Journal of Refrigeration**, v. 68, p. 76-93, 2016.
- AHMED, H. E.; SALMAN, B. H.; KHERBEET, A. S.; AHMED, M. I. Optimization of thermal design of heat sinks: A review. **International Journal of Heat and Mass Transfer**, v. 118, p. 129–153, 2018.
- ALAM, T., LI, W.; CHANG, W.; YANG, F.; KHAN, J.; LI, C. A comparative study of flow boiling HFE-7100 in silicon nanowire and plainwall microchannels. **International Journal of Heat and Mass Transfer**, v. 124, p. 829–840, 2018.
- AL-HAYES, R. A. M.; WINTERTON, R. H. S. Bubble diameter on detachment in flowing liquids. **International Journal of Heat and Mass Transfer**, v. 24, p. 223-230, 1981.
- ANSYS. **ANSYS® Fluent Theory Guide, version 18.1.0**. 2017a.
- ANSYS. **ANSYS® Fluent User's Guide, version 18.1.0**. 2017b.
- ARIYO, D. O.; BELLO-OCHEDE, T. Optimization of geometric and flow parameters of horizontal rectangular microchannels in subcooled two-phase flow. **Proceedings of 14th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics**, p. 1197-1202, 2019.
- BAI, P.; TANG, T.; TANG, B. Enhanced flow boiling in parallel microchannels with metallic porous coating. **Applied Thermal Engineering**, v. 58, n. 1-2, p. 291-297, 2013.
- BAO, Z. Y.; FLETCHER, D. F.; HAYNES, B. S. Flow boiling heat transfer of Freon R11 and HCFC123 in narrow passages. **International Journal of Heat and Mass Transfer**, v. 43, n. 18, 2000.
- BARTH, T.; JESPERSEN, D. The design and application of upwind schemes on unstructured meshes. **27th Aerospace Sciences Meeting**, 1989.
- BARTOLEMEI, G. G.; CHANTURIYA, V. M. Experimental study of true void fraction when boiling subcooled water in vertical tubes. **Teploenergetika**, v. 14, n. 2, p. 123-128, 1967.
- BEJAN, A. **Advanced Engineering Thermodynamics**. 2^a Ed., USA: Wiley, 1997.
- BEJAN, A. **Convection Heat Transfer**. 4th Edition, USA: Wiley, 2013.
- BEJAN, A. **Freedom and Evolution**. Cham, Switzerland: Springer, 2020.
- BEJAN, A.; KRAUS, A. D. **Heat Transfer Handbook**. Hoboken, New Jersey, USA: John Wiley and Sons, 2003.

- BEJAN, A.; LORENTE, S. Constructal law of design and evolution: Physics, biology, technology, and society. **Journal of Applied Physics**, v. 113, n. 151301, 2013.
- BEJAN, A.; LORENTE, S. **Design with Constructal Theory**, Hoboken Wiley, 2008.
- BEJAN, A.; ZANE, J. P. **Design in Nature**. New York, USA: Doubleday, 2012.
- BELLO-OCHEDE, T.; LIEBENBER, L.; MEYER, J. P. Constructal cooling channels for microchannel heat sinks. **International Journal of Heat and Mass Transfer**, v. 50, p. 4141-4150, 2007.
- BELLO-OCHEDE, T.; MEYER, J. P.; BEJAN, A. Constructal Ducts with wrinkled entrances. **International Journal of Heat and Mass Transfer**, v. 52, p. 3628-3633, 2009.
- BHATI, J.; PARUYA, S. Numerical simulation of bubble dynamics in subcooled flow boiling in a channel. **Nuclear Engineering and Design**, v. 371, 2021.
- BISERNI, C.; DALPIAZ, F. L.; FAGUNDES, T. M.; ROCHA, L. A. O. Constructal design of T-shaped morphing fins coupled with a trapezoidal basement: A numerical investigation by means of exhaustive search and genetic algorithm. **International Journal of Heat and Mass Transfer**, v. 109, p. 73-81, 2017.
- BISERNI, C.; ROCHA, L. A. O.; STANESCU, G.; LORENZINI, E. Constructal H-shaped cavities according to Bejan's theory. **International Journal of Heat and Mass Transfer**, v. 50, p. 2132-2138, 2007.
- BORBA MARCHETTO, D.; RIBATSKI, G. An experimental study on flow boiling heat transfer of HFO1336mzz(Z) in microchannels-based polymeric heat sinks. **Applied Thermal Engineering**, v. 180, p. 115815, 2020.
- BOWRING, R. W. A Simple but Accurate Round Tube, Uniform Heat Flux, Dryout Correlation Over the Pressure Range 0.7-17 mm/m². **UKAEA Report AEEWR789**, 1972.
- ÇEBİ, A.; CELEN, A.; DONMEZ, A. H.; KARAKOYUN., Y.; CELEN, P.; CELLEK, M. S.; DALKILIÇ., A. S.; TANER, T.; WONGWISES, S. A review of flow boiling in mini and microchannel for enhanced geometries. **Journal of Thermal Engineering**, v. 4, n. 3, p. 2037-2074, 2018.
- CHARNAY, R.; REVELLIN, R.; BONJOUR, J. Flow boiling characteristics of R-245fa in a minichannel at medium saturation temperatures. **Experimental Thermal and Fluid Science**, v. 59, p. 184-194, 2014.
- CHEN, J. C. Correlation for Boiling Heat Transfer to Saturated Fluids in Convective Flow. **Industrial & Engineering Chemistry, Process Design and Development**, v. 5, p. 322, 1966.
- CHEN, J.; ZHANG, S.; TANG, Y.; CHEN, H.; YUAN, W.; ZENG, J. Effect of operational parameters on flow boiling heat transfer performance for porous interconnected microchannel nets. **Applied**

- Thermal Engineering**, v. 121, p. 443–453, 2017.
- CHEN, T.; GARIMELLA, S. V. Measurements and high-speed visualization of flow boiling of a dielectric fluid in a silicon microchannel heat sink. **International Journal of Multiphase Flow**, v. 32, n. 8, p. 957–971, 2006.
- CHENG, L.; XIA, G. Fundamental issues, mechanisms and models of flow boiling heat transfer in microscale channels. **International Journal of Heat and Mass Transfer**. v. 108, Part A, p. 97–127, 2017.
- COLE, R. A photographic study of pool boiling in the region of the critical heat flux. **AIChE Journal**, v. 6, p. 533–542, 1960.
- COLLIER, J. G. **Convective Boiling and Condensation**. United Kingdom: McGraw-Hill, 1981.
- COLLIER, J. G.; THOME, J. R. **Convective Boiling and Condensation**. 3rd ed., Oxford, United Kingdom, Clarendon Press, 1994.
- DA SILVEIRA, T.; PINTO, V. T.; NEUFELD, J. P. S.; PAVLOVIC, A.; ROCHA, L. A. O.; DOS SANTOS, E. D.; ISOLDI, L. A. Applicability Evidence of Constructal Design in Structural Engineering: Case Study of Biaxial Elasto-Plastic Buckling of Square Steel Plates with Elliptical Cutout. **Journal of Applied and Computational Mechanics**, v. 7, p. 922–934, 2021.
- DADHICH, M.; PRAJAPATI, O. S.; SHARMA, V. A systematic review on the heat transfer investigation of the flow boiling process. **Chemical Engineering and Processing - Process Intensification**, v. 165, p. 108425, 2021.
- DAS, S.; PUNEKAR, H. On Development of a Semi-Mechanistic Wall Boiling Model. **Journal of Heat Transfer**, v. 138, 2016.
- DAVIS, E. J.; ANDERSON, G. H. The Incipience of Nucleate Boiling in Forced Convection Flow. **AIChE Journal**, v. 12, p. 774, 1966.
- DEL VALLE, V. H.; KENNING, D. B. R. Subcooled flow boiling at high heat flux. **International Journal of Heat and Mass Transfer**. v. 28, n. 10, p. 1907–1920, 1985.
- DENG, D.; ZENG, L.; SUN, W. A review on flow boiling enhancement and fabrication of enhanced microchannels of microchannel heat sinks. **International Journal of Heat and Mass Transfer**, v. 175, p. 121332, 2021.
- DITTUS, F. W.; BOELTER, M. K. **Publications on Engineering**. University of California, Berkeley, EUA, Vol. 2, p. 443, 1930.
- DREW, D. A. Mathematical Modeling of Two-Phase Flow. **Annual Review of Fluid Mechanics**, v. 15, p. 261–291, 1983.
- DREW, D. A.; LAHEY, R. T. **In particulate two-phase flow**. Boston, MA, USA: Butterworth-

- Heinemann, p. 509-566, 1993.
- DREW, T. B.; MUELLER, C. Boiling. **Transactions of AIChE**, v. 33, p. 449-471, 1937.
- ESTRADA, E. S. D.; BARRETO, E. X.; ISOLDI, L. A.; DOS SANTOS, E. D.; LORENTE, S.; L. A. O. ROCHA. Constructal design of tree shaped cavities inserted into a cylindrical body with heat generation. **International Journal of Thermal Sciences**, v. 152, 2020.
- FANG, X.; WANG, R.; CHEN, W.; ZHANG, H.; MA, C. A review of flow boiling heat transfer of nanofluids. **Applied Thermal Engineering**, v. 91, p. 1003–1017, 2015.
- FEIJÓ, B. C.; LORENZINI, G.; ISOLDI, L. A.; ROCHA, L. A. O.; GOULART, J. N. V.; DOS SANTOS, E. D. Constructal design of forced convective flows in channels with two alternated rectangular heated bodies. **International Journal of Heat and Mass Transfer**, v. 125, p. 710-721, 2018.
- FEIJÓ, B. C.; PAVLOVIC, A.; ROCHA, L. A.O.; ISOLDI, L. A.; LORENTE, S.; DOS SANTOS, E. D. Geometrical investigation of microchannel with two trapezoidal blocks subjected to laminar convective flows with and without boiling. **Reports in Mechanical Engineering**, v. 3, n. 1, p. 20-36, 2022.
- FERRARI, A.; MAGNINI, M.; THOME, J. R. Numerical analysis of slug flow boiling in square microchannels. **International Journal of Heat and Mass Transfer**, v. 123, p. 928-944, 2018.
- FERZIGER, J. H.; PÉRIC, M.; STREET, R. L. **Computational Methods for Fluid Dynamics**. 4th ed., USA: Springer, 2020.
- FREIRE, G. Emprego de Modelagem Computacional na Investigação da Influência Geométrica em Cavidade Aletada Sujeita a Convecção de Calor por Ebulição com Base no Design Construtal. Dissertation, Computational Modeling, FURG, 2021.
- GAO, L.; BHAVNANI, S. H. Enhanced boiling in microchannels due to recirculation induced by repeated saw-toothed cross-sectional geometry. **Applied Physics Letters**, v. 111, n. 18, 2017.
- GHANI, I. A.; SIDIK, N. A. C.; KAMARUZAMAN, N. Hydrothermal performance of microchannel heat sink: The effect of channel design. **International Journal of Heat and Mass Transfer**. v. 107, p. 21–44, 2017.
- GONZALES, GILL VELLEDA ; BISERNI, CESARE ; DA SILVA DIAZ ESTRADA, EMANUEL ; PLATT, GUSTAVO MENDES ; ISOLDI, LIÉRCIO ANDRÉ ; ROCHA, LUIZ ALBERTO OLIVEIRA ; DA SILVA NETO, ANTÔNIO JOSÉ ; DOS SANTOS, ELIZALDO DOMINGUES . Investigation on the Association of Differential Evolution and Constructal Design for Geometric Optimization of Double Y-Shaped Cooling Cavities Inserted into Walls with Heat Generation. **Applied Sciences-Basel**, v. 13, p. 1998-30, 2023.
- GONZALES, G. V. ; Lorenzini, G. ; ISOLDI, L. A. ; ROCHA, L. A. O. ; Dos Santos, E. D. ; SILVA NETO, A. J. . Constructal Design and Simulated Annealing applied to the geometric optimization

- of an isothermal Double T-shaped cavity. **International Journal of Heat and Mass Transfer**, v. 174, p. 121268-19, 2021.
- GONZALES, G. V. ; Estrada, E. da S. D. ; EMMENDORFER, L. R. ; ISOLDI, L. A. ; XIE, G. ; ROCHA, L. A. O. ; Dos Santos, E. D. . A comparison of simulated annealing schedules for constructal design of complex cavities intruded into conductive walls with internal heat generation. **Energy (Oxford)**, v. 93, p. 372-382, 2015.
- GONZÁLEZ, D.; AMIGO, J.; LORENTE, S.; BEJAN, A.; SUÁREZ, F. Constructal design of salt-gradient solar pond fields. **International Journal of Energy Research**, v. 40, n. 10, p. 1428–1446, 2016.
- GOOGLE EARTH. **Satellite view of the city of Rome, Italy**. Available in: <https://earth.google.com/web/@41.89483079,12.48316812,42.5206113a,7293.07083138d,35y,0h,0t,0r>. Accessed in: October 24, 2021.
- GUAN, P.; JIA, L.; YIN, L.; TAN, Z. Bubble departure size in flow boiling. **Heat and Mass Transfer**, v. 51, n. 7, p. 921–930, 2015.
- GUERRERO, E.; MUÑOZ, F.; RATKOVICH, N. Comparison between Eulerian and VOF models for two-phase flow assessment in vertical pipes. **CT&F - Ciencia, Tecnología y Futuro**, v. 7, n. 1, p. 73-84, 2017.
- GUNGOR, A. E.; WINTERTON, R. H. S. Simplified General Correlation for Saturated Flow Boiling and Comparisons of Correlations With Data. **Chemical Engineering Research and Design**, v. 65, p. 148-156, 1987.
- HE, H. P.; MA, H. G.; BAI, J. M. Influencing Factors on Heat Transfer Performance for Flow Boiling of R410A in Horizontal Micro-Fin Tubes. **Advanced Materials Research**, v. 472-475, p. 1676-1680, 2012.
- HEWITT, G. F. Critical Heat Flux in Flow Boiling. **Proceedings of the 6th International Heat Transfer Conference**, Toronto, Canada, v. 6, p. 143-171, 1978.
- HEWITT, G. H.; HALL-TAYLOR, N. S. **Annular Two-phase Flow**. NY, USA: Pergamon Press, 1970.
- HUIYING LI; VASQUEZ, S. A.; PUNEKAR, H.; MURALIKRISHNAN, R. Prediction of boiling and critical heat flux using an Eulerian multiphase boiling model. **Proceedings of the ASME**, 2011.
- HUO, X.; CHEN, L.; TIAN, Y. S.; KARAYIANNIS, T. G. Flow boiling and flow regimes in small diameter tubes. **Applied Thermal Engineering**. v. 24, n. 8-9, p. 1225-1239, 2004.
- HWANG, C. L.; YOON, K. **Multiple Attribute Decision Making: Methods and Applications**. New York, USA: Springer-Verlag, 1981.
- HWANG, C. L.; LAI, Y. J.; LIU, T. Y. A new approach for multiple objective decision making.

- Computers and Operational Research**, v. 20, n. 8, p. 889–899, 1993.
- INCROPERA, F. P.; DEWITT, D. P.; BERGMAN, T. L.; LAVINE, A. S. **Fundamentos de Transferência de Calor**. 6thed., Hoboken, New Jersey, USA: John Wiley and Sons, 2007.
- ISHII, M.; KIM, S. Micro Four-Sensor Probe Measurement of Interfacial Area Transport fo Bubbly Flow in Round Pipes. **Nuclear Engineering and Design**, v. 205, p. 123-131, 2001.
- ISHII, M.; MISHIMA, K. Two-fluid model and hydrodynamic constitutive relations. **Nuclear Engineering and Design**, v. 82, p. 107-126, 1984.
- IWANISZYN, M.; JODŁOWSKI, P. J.; SINDERA, K.; GANCARCZYK, A.; KORPYŚ, M.; JĘDRZEJCZYK, R. J.; KOŁODziej, A. Entrance effects on forced convective heat transfer in laminar flow through short hexagonal channels: Experimental and CFD study. **Chemical Engineering Journal**, v. 405, p. 126635, 2021.
- JURIC, D.; TRYGGVASON, G. Computations of Boiling Flows. **International Journal of Multiphase Flow**, v. 24, n. 3, p. 387-410, 1998.
- KANDLIKAR, S. G. A Parametric Study of Flow Boiling Heat Transfer Inside Horizontal and Vertical Tubes. **Experimental Heat Transfer, Fluid Mechanics and Thermodynamics**, p. 1618-1626, 1988.
- KANDLIKAR, S. G. Development of Flow Boiling Map for Subcooled and Saturated Flow Boiling of Different Fluids Inside Circular Tube. **Journal of Heat Transfer**, 1991.
- KANDLIKAR, S. G. Fundamental issues related to flow boiling in minichannels and microchannels. **Experimental Thermal and Fluid Science**, v. 26, n. 2-4, p. 389–407, 2002.
- KANDLIKAR, S. G.; SHOJI, M.; DHIR, V. K. **Handbook of Phase Change: Boiling and Condensation**. Philadelphia, USA: Taylor & Francis, 1999.
- KANDLIKAR, S.G.; GRANDE, W.J. Evolution of Microchannel Flow Passages – Thermohydraulic Performance and Fabrication Technology. **Heat Transfer Engineering**, v. 25, n. 1, 2003.
- KARAYIANNIS, T. G.; MAHMOUD, M. M. Flow boiling in microchannels: Fundamentals and applications. **Applied Thermal Engineering**, v. 115, 25, p. 1372-1397, 2017.
- KEW, P. A.; CORNWELL, K. Correlations for the prediction of boiling heat transfer in small-diameter channels. **Applied Thermal Engineering**, v. 17, p. 705–715, 1997.
- KHAN, M. G.; FARTAY, A. A review on microchannel heat exchangers and potential applications. **International Journal of Energy Research**, v. 35, p. 553–582, 2011.
- KONCAR, B.; KLJENAK, I.; MAVKO, B. Modelling of Local Two-Phase Flow Parameters in Upward Subcooled Flow Boiling at Low Pressure. **International Journal of Heat and Mass Transfer**, v. 47, p. 1499-1512, 2004.

- KONCAR, B.; KREPPER, E.; EGOROV, Y. CFD modeling of subcooled flow boiling for nuclear engineering applications. **Proceedings of the International Conference “Nuclear Energy for New Europe”**, p. 140/1-14, 2005.
- KONDEPUDI, D.; PRIGOGINE. **Modern Thermodynamics. From heat engines to dissipative structures**. 2nd ed., West Sussex, United Kingdom, John Wiley & Sons, 2015.
- KOUSALYA, A. S.; SINGH, K. P.; FISHER, T. S. Heterogeneous wetting surfaces with graphitic petal-decorated carbon nanotubes for enhanced flow boiling. **International Journal of Heat and Mass Transfer**, v. 87, p. 380–389, 2015.
- KUMAR, G. N. M.; CHAYAMPURATH, R.; PARAMBIL, S. M. Preserving of intercostobrachial nerve relieves breast carcinoma patients from postmastectomy pain syndrome. **International Journal of Research in Medical Sciences**, v. 5, n. 2, p. 509-514, 2017.
- KURUL, N.; PODOWSKI, M. Z. On the modeling of multidimensional effects in boiling channels. **Proceedings of the 27th National Heat Transfer Conference**, Minneapolis, Minnesota, USA, 1991.
- KUTATELADZE, S. S.; LEONT, A. I. Some applications of the asymptotic theory of the turbulent boundary layer. **Proceedings of Third International Heat Transfer Conf.**, American Institute of Chemical Engineers, New York, v. 3, p. 1–6, 1966.
- LAGE, J. L.; ANDERSON, R.; COSTA, V.; ERRERA, M. R.; FOWLER, A. J.; GOSSELIN, L.; KIMURA, S.; LEDEZMA, G. A.; LIM, J. S.; LORENTE, S.; MIGUEL, A.; MINKOWYCZ, W. J.; MOREGA, A. M.; BELLO-OCHEDE, T.; ORDÓÑEZ, J. C.; POULIKAKOS, D.; REIS, H.; ROCHA, L. A. O.; DA SILVA, A. K.; TREVISAN, O. V.; VARGAS, J. V.; WECHSATOL, W.; ZHANG, Z.; Professor Adrian Bejan on his 60th birthday. **International Journal of Heat and Mass Transfer**, v. 51, n. 25–26, 2008,
- LAHEY, R. T.; MOODY, F. J. The Thermal Hydraulics of Boiling Water Nuclear Reactor. **American Nuclear Society**, 1997.
- LARSEN, P. S.; TONG, L. S. Void Fractions in Subcooled Flow Boiling. **Journal of Heat Transfer**, v. 91, n. 4, p. 471, 1969.
- LAZAREK, G. M.; BLACK, S. H. Evaporative heat transfer, pressure drop and critical heat flux in a small vertical tube with R-113, **International Journal of Heat and Mass Transfer**, v. 25, n. 7, p. 945–960, 1982.
- LEE, C. H.; MUDAWAR, I. A mechanistic critical heat flux model for subcooled flow boiling based on local bulk flow conditions. **International Journal of Multiphase Flow**, v. 14, p. 711–728, 1988.
- LEE, J.; MUDAWAR, I. Two-phase flow in high-heat-flux micro-channel heat sink for refrigeration cooling applications: part II-heat transfer characteristics. **International Journal of Heat and Mass Transfer**, v. 48, p. 941–955, 2005.

- LEE, W. H. **A Pressure Iteration Scheme for Two-Phase Flow Modeling. Multiphase Transport Fundamentals, Reactor Safety, Applications.** Washington, DC, USA, 1980.
- LEMMERT, M.; CHAWLA, L. M. **Influence of flow velocity on surface boiling heat transfer coefficient in heat transfer in boiling.** New York, USA: Academic Press and Hemisphere, 1977.
- LEVY, S. Forced Convection Subcooled Boiling – Prediction of Vapour Volumetric Function. **General Electric Co. Report, GEAP**, p. 5157, 1966.
- LIMA, J. P. S.; CUNHA, M. L.; DOS SANTOS, E. D.; ROCHA, L. A. O.; REAL, M. V.; ISOLDI, L. A. Constructal Design for the ultimate buckling stress improvement of stiffened plates submitted to uniaxial compressive load. **Engineering Structures**, v. 203, p. 109883, 2020.
- LING, K.; SON, G.; SUN, D.; TAO, W. Three dimensional numerical simulation on bubble growth and merger in microchannel boiling flow. **International Journal of Thermal Sciences**, v. 98, p. 135-147, 2015.
- LIU, Q.; WANG, W.; PALM, B. A Numerical study of the transition from slug to annular flow in micro-channel convective boiling. **Applied Thermal Engineering**, v. 112, p. 73-81, 2017.
- LOPEZ DE BERTODANO, M. **Turbulent Bubbly Flow in a Triangular Duct**, 1991. Ph.D. Thesis. Rensselaer Polytechnic Institute, Troy, New York.
- LOPEZ DE BERTODANO, M.; LAHEY JR.; R. T., JONES, O. C. Development of a $\kappa - \epsilon$ model for bubbly two-phase flow. **Journal of Fluids Engineering**, v. 116, p. 128-134, 1994.
- LORENZINI, G.; BISERNI, C.; ESTRADA, E. D.; ISOLDI, L. A.; DOS SANTOS, E. D.; ROCHA, L. A. O. Constructal Design of convective Y-shaped cavities by means of genetic algorithm. **Journal of Heat Transfer**, v. 136, p. 71702-71710, 2014.
- LORENZINI, G.; ESTRADA, E. D.; ISOLDI, L. A.; DOS SANTOS, E. D.; ROCHA, L. A. O. Constructal Design of convective cavities inserted into cylindrical solid body for cooling. **International Journal of Heat and Mass Transfer**, v. 83, p. 75-83, 2015.
- LORENZINI, G.; GARCIA, F. L.; DOS SANTOS, E. D.; BISERNI, C.; ROCHA, L. A. O. Constructal Design Applied to the optimization of complex geometries: T-Y-shaped cavities with two additional lateral intrusions cooled by convection. **International Journal of Heat and Mass Transfer**, v. 55, p. 1505-1512, 2012.
- LORENZINI, G.; MACHADO, B. S.; ISOLDI, L. A.; DOS SANTOS, E. D.; ROCHA, L. A. O. Constructal Design of Rectangular Fin Intruded Into Mixed Convective Lid-Driven Cavity Flows. **Journal of Heat Transfer**, v. 138, n. 10, 2016.
- LORENZINI, G.; ROCHA, L. A. O. Geometric optimization of T-Y shaped cavity according to Constructal Design. **International Journal of Heat and Mass Transfer**, v. 52, p. 4683-4688, 2009.

- LU, C. T.; PAN, C. Stabilization of flow boiling in microchannel heat sinks with a diverging cross-section design. **Journal of Micromechanics and Microengineering**, v. 18, 2008.
- LUO, Y.; LI, W.; ZHOU, K.; SHENG, K.; SHAO, S.; ZHANG, Z.; DU, J., MINKOWYCZ, W. J. Three-dimensional numerical simulation of saturated annular flow boiling in a narrow rectangular microchannel. **International Journal of Heat and Mass Transfer**, v. 149, 2020.
- MAGNINI, M.; PULVIRENTI, B.; THOME, J. R. Numerical investigation of hydrodynamics and heat transfer of elongated bubbles during flow boiling in a microchannel. **International Journal of Heat and Mass Transfer**, v. 59, p. 451-471, 2013.
- MALISKA, C. R. **Transferência de Calor e Mecânica dos Fluidos Computacional**. Rio de Janeiro, Brazil: Livros Técnicos e Científicos Editora S.A., 2004.
- MANNINEN, M.; TAIVASSALO, V., Kallio, S. On the Mixture Model for Multiphase Flow. **VTT Publications**, 1996.
- MARDANPOUR, P.; IZADPANAHI, E.; POWELL, S.; RASTKAR, S.; BEJAN, A. Inflected wings in flight: uniform flow of stresses makes strong and light wings for stable flight. **Journal of Theoretical Biology**, v. 508, p. 110452, 2021.
- MARKAL, B.; AYDIN, O.; AVCI, M. An experimental investigation of saturated flow boiling heat transfer and pressure drop in square microchannels. **International Journal of Refrigeration**, v. 65, p. 1-11, 2016.
- MARTINS, J. C.; GOULART, M. M.; GOMES, M. DAS N.; SOUZA, J. A.; ROCHA, L. A. O.; ISOLDI, L. A.; DOS SANTOS, E. D. Geometric evaluation of the main operational principle of an overtopping wave energy converter by means of Constructal Design. **Renewable Energy**, v. 118, p. 727–741, 2018.
- MARTINS, J. C.; GOULART, M. M.; GOMES, M. N.; SOUZA, J. A.; ROCHA, L. A. O.; ISOLDI, L. A.; DOS SANTOS, E. D. Geometric Evaluation of the Main Operational Principle of an Overtopping Wave Energy Converter by Means of Constructal Design. **Renewable Energy**, v. 118, p. 727–741, 2018.
- MENTER, F. R. Review of the SST Turbulence Model Experience from an Industrial Perspective. **International Journal of Computational Fluid Dynamics**, v. 23, n. 4, 2009.
- MENTER, F. R. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. **AIAA Journal**, v. 38, n. 8, p. 1598-1605, 1994.
- MENTER, F. R.; KUNTZ, M.; BENDER, R. A scale-adaptive simulation model for turbulent flow predictions. **AIAA Journal**, 2003.
- MIGUEL, A. F. Constructal design of solar energy-based systems for buildings. **Energy and Buildings**, v. 40, n. 6, p. 1020–1030, 2008.

- MINER, M. J.; PHELA, P. E.; ODOM, B. A.; ORTIZ, C. A.; PRASHER, R. S.; SHERBECK, J. A. Optimized Expanding Microchannel Geometry for Flow Boiling. **Journal of Heat Transfer**, Vol. 135, 2013
- MOHAMMED, H. A.; GUNNASEGARAN, P.; SHUAIB, N. H. Influence of channel shape on the thermal and hydraulic performance of microchannel heat sink. **International Communication in Heat and Mass Transfer**. v. 38, p. 474–80, 2011a.
- MOHAMMED, H. A.; GUNNASEGARAN, P.; SHUAIB, N. H. Numerical simulation of heat transfer enhancement in wavy microchannel heat sink. **International Communication in Heat and Mass Transfer**. v. 38, p. 63–8, 2011b.
- MORAGA, F. J.; BONETTO, R. T.; LAHEY, R. T. Lateral forces on spheres in turbulent uniform shear flow. **International Journal of Multiphase Flow**, v. 25, p. 1321-1372, 1999.
- MORSHED, A. K. M. M.; PAUL, T. C.; KHAN, J. Effect of Cu–Al₂O₃ nanocomposite coating on flow boiling performance of a microchannel. **Applied Thermal Engineering**, v. 51, n. 1-2, p. 1135–1143, 2013.
- MOUKALLED, F.; DARWISH, M.; MANGANI, L. **The Finite Volume Method in Computational Fluid Dynamics: An Advanced Introduction with OpenFOAM® and Matlab**. Switzerland: Springer International Publishing, 2016.
- MUKHERJEE, S.; EBRAHIM, S.; MISHRA, P.C.; ALI, N.; CHAUDHURI, P. A Review on Pool and Flow Boiling Enhancement Using Nanofluids: Nuclear Reactor Application. **Processes** **2022**, v. 10, p. 177, 2022.
- NAQUIUDDIN, N. H.; SAW, L. H.; YEW, M. C.; YUSOF, F.; NG, T.C.; YEW, M. K. Overview of Micro-Channel Design for High Heat Flux Application. **Renewable and Sustainable Energy Reviews**, v. 82, p. 901–914, 2018.
- NAYAK, A. K.; LATHOUWERS, D.; VAN DER HAGEN, T. H. J. J.; SCHRAUWEN, F.; MOLENAAR, P.; ROGERS, A. A numerical study of boiling flow instability of a reactor thermosyphon system. **Applied Thermal Engineering**, v. 26, p. 644-653, 2006.
- NUKIYAMA, S. J. The Maximum and Minimum Values of the Heat Q Transmitted from Metal to Boiling Water under Atmospheric Pressure. **Journal of the Society of Mechanical Engineers**, Japan. v. 37, p. 367-374, 1934.
- NUNES, B. R.; RODRIGUES, M. K.; OLIVEIRA ROCHA, L. A.; LABAT, M.; LORENTE, S.; SANTOS, E. D.; BISERNI, C. Numerical-analytical study of earth-air heat exchangers with complex geometries guided by constructal design. **International Journal of Energy Research**, 2021.
- NUNES, B. R.; RODRIGUES, M. K.; ROCHA, L. A. O.; LABAT, M.; LORENTE, S.; DOS SANTOS, E. D.; ISOLDI, L. A.; BISERNI, C. Numerical-analytical study of Earth-air heat exchangers with complex geometries guided by constructal design. **International Journal of**

- Energy Research**, v. 45, p. 7157-18, 2021.
- OHADI, M.; CHOO, K.; DESSIATOUN, S.; CETEGEN, E. Emerging Applications of Microchannels. **Briefs in Applied Sciences and Technology**, p. 67–105, 2012.
- OJEDA, J.; MESSINA, S.; VÁZQUEZ, E.; MÉNDEZ, F. Constructal design of top metallic contacts on a disc-shaped solar cell. **Journal of Applied Research and Technology**, v. 19, n. 5, p. 492-507, 2021.
- OTOMO, Y.; SANTIAGO GALICIA, E.; ENOKI, K. Enhancement of Subcooled Flow Boiling Heat Transfer with High Porosity Sintered Fiber Metal. **Applied Sciences**, v. 11, n. 3, p. 1237, 2021.
- ÖZDEMİR, M. R.; MAHMOUD, M. M.; KARAYIANNIS, T. G. Flow Boiling Pressure Drop Characteristics in a Rectangular Metallic Microchannel. **Proceedings of the 5th World Congress on Mechanical, Chemical, and Material Engineering (MCM'19)**, Lisbon, Portugal, 2019.
- PERRET, C.; SCHAEFFER, CH.; BOUSSEY, J. Microchannel integrated heat sinks in silicon technology. **Proceedings of 1998 IEEE industry applications conference**, v. 2 USA, p. 1051–55, 1998.
- PRAJAPATI, Y. K. Influence of fin height on heat transfer and fluid flow characteristics of rectangular microchannel heat sink. **International Journal of Heat and Mass Transfer**, v. 137, p. 1041-1052, 2019.
- QASEM, N. A. A.; ZUBAIR, S. M. Compact and microchannel heat exchangers: A comprehensive review of air-side friction factor and heat transfer correlations. **Energy Conversion and Management**, v. 173, p. 555–601, 2018.
- RAI, S. K.; SHARMA, R.; SAIFI, M.; TYAGI, R.; SINGH, D.; GUPTA, H. Review of recent applications of micro channel in mems devices. **International Journal of Applied Engineering Research**, v. 13, n. 9, p. 64-69, 2018.
- RANZ, W. E.; MARSHALL, W. R. Vaporation form Droplets, Part I. **Chemical Engineering Progress**, v. 48, n. 3, p. 141-146, 1952.
- RATTNER, A. S.; GARIMELLA, S. Simple mechanistically consistent formulation for volume-of-fluid based computations of condensing flows. **Journal of Heat Transfer**, v. 136, n. 7, 2014.
- RAZERA, A. L.; DA FONSECA, R. J. C.; ISOLDI, L. A.; DOS SANTOS, E. D.; ROCHA, L. A. O.; BISERNI, C. Constructal design of a semi-elliptical fin inserted in a lid-driven square cavity with mixed convection. **International Journal of Heat and Mass Transfer**, v. 126, part B, p. 81-94, 2018.
- ROCHA, L. A. O.; LORENTE, S.; BEJAN, A. Constructal design for cooling a disc-shaped area by conduction. **International Journal of Heat and Mass Transfer**, v. 45, n. 8, p. 1643-1652, 2002.
- ROCHA, L. A. O.; LORENZINI, E.; BISERNI, C. Geometric optimization of shapes on basis of

- Bejan's Constructal Theory. **International Journal of Heat and Mass Transfer**, v. 32, p. 1281-1288, 2005.
- RODRIGUES, M. K.; BRUM, R. B.; VAZ, J.; ROCHA, L. A. O.; DOMINGUES, E. D.; ISOLDI, L. A. Numerical investigation about the improvement of the thermal potential of an Earth-Air Heat Exchanger (EAHE) employing the Constructal Design method. **Renewable Energy**, v. 80, p. 538-551, 2015.
- RODRIGUES, P. M.; BISERNI, C.; DE ESCOBAR, C. C.; ROCHA, L. A. O.; ISOLDI, L. A.; DOS SANTOS, E. D. Geometric optimization of a lid-driven cavity with two rectangular intrusions under mixed convection heat transfer: A numerical investigation motivated by constructal design. **International Communications in Heat and Mass Transfer**, v. 117, 2020.
- ROHSENOW, W. M. A Method of Correlating Heat Transfer Data for Surface Boiling of Liquids. **Trans ASME**, v. 74, p. 969, 1952.
- ROHSENOW, W. M.; HARTNETT, J. P.; CHO, Y. I. **Handbook of Heat Transfer**. 3rd ed., USA: McGraw-Hill, 1998.
- SADAGHIANI, A. K.; KOSAR, A. Numerical and experimental investigation on the effects of diameter and length on high mass flux subcooled flow boiling in horizontal microtubes. **International Journal of Heat and Mass Transfer**, v. 92, p. 824-837, 2016.
- SAHA, P.; ZUBER, N. Point of net vapor generation and vapor void fraction in subcooled boiling. **Proceedings of 5th International Heat Transfer Conference**, Tokyo, Japan, v. 4, p. 175-179, 1974.
- SEMPÉRTGUI-TAPIA, D. F.; RIBATSKI, G. The effect of the cross-sectional geometry on saturated flow boiling heat transfer in horizontal micro-scale channels. **Experimental Thermal and Fluid Science**, v. 89, p. 98-109, 2017.
- SIDDIQUI, O. K.; ZUBAIR, S. M. Efficient energy utilization through proper design of microchannel heat exchanger manifolds: A comprehensive review. **Renewable and Sustainable Energy Reviews**. v. 74, p. 969–1002, 2017.
- SONG, Y.; ASADI, M.; XIE, G.; ROCHA, L. A. O. Constructal wavy-fin channels of a compact heat exchanger with heat transfer rate maximization and pressure losses minimization. **Applied Thermal Engineering**, v. 75, p. 24-32, 2015.
- STEINER, H.; KOBOR, A.; GEBHARD, L. A wall heat transfer model for subcooled flow boiling. **International Journal of Heat and Mass Transfer**, v. 48, p. 4161-4173, 2005.
- SUJITH KUMAR, C. S.; SURESH, S.; PRAVEEN, A. S.; SANTHOSH KUMAR, M.C.; GOPI, V. Effect of surfactant addition on hydrophilicity of ZnO–Al₂O₃ composite and enhancement of flow boiling heat transfer. **Experimental Thermal and Fluid Science**, v. 70, p. 325–334, 2016.
- SUN, L.; MISHIMA, K. An evaluation of prediction methods for saturated methods for saturated flow

- boiling heat transfer. **International Journal of Heat and Mass Transfer**, v. 52, p. 5323–5329, 2009.
- TEIXEIRA, F. B.; LORENZINI, G.; ERRERA, M. R.; ROCHA, L. A. O.; ISOLDI, L. A.; DOS SANTOS, E. D. Constructal Design of triangular arrangements of square bluff bodies under forced convective turbulent flows. **International Journal of Heat and Mass Transfer**, v. 126, part A, p. 521-535, 2018.
- THOM, J. R. S.; WALKER, W. M.; FALLON, T. A.; REISING, G. F. S. Boiling in Subcooled Water During Flow Up Heated Tubes or Annuli. Symposium on Boiling Heat Transfer in Steam Generation Units and Heat Exchangers, Manchester, United Kingdom, **IMechE**, p. 6, 1965.
- THOME, J. R.; CIONCOLINI, A. Flow Boiling in Microchannels. **Advances in Heat Transfer**, v. 49, p. 157-224, 2017.
- TOLUBINSKI, V. I.; KOSTANCHUK, D. M. Vapor bubbles growth rate and heat transfer intensity at subcooled water boiling. **Proceedings of the 4th International Heat Transfer Conference**, Paris, France, 1970.
- TONG, L. S. Boundary-layer analysis of the flow boiling crisis. **International Journal of Heat and Mass Transfer**, v. 11, p. 1208–1211, 1968.
- TONG, L. S.; TANG, Y. S. **Boiling Heat Transfer and Two-Phase Flow**. 2nd ed., USA: Taylor and Francis, 1997.
- TONG, W.; BERGLES, A. E.; JENSEN, M. K. Pressure drop with highly subcooled flow boiling in small diameter tubes. **Experimental Thermal and Fluid Science**, v. 15, p. 202–212, 1997.
- TU, J. Y.; YEOH, G. H. On Numerical Modelling of Low-Pressure Subcooled Boiling Flows. **International Journal of Heat and Mass Transfer**, v. 45, p. 1197-1209, 2002.
- VERSTEEG, H. K.; MALALASEKERA, W. **An Introduction to Computational Fluid Dynamics – The Finite Volume Method**. 2nd ed., Essex, United Kingdom: Pearson Education Limited, 2007.
- VIEIRA, R. S.; PETRY, A. P.; ROCHA, L. A. O.; ISOLDI, L. A.; DOS SANTOS, E. D. Numerical evaluation of a solar chimney geometry for different ground temperatures by means of constructal design. **Renewable Energy**, v. 109, p. 222-234, 2017.
- VIJAYARANGAN, B. R.; JAYANTI, S.; BALAKRISHNAN, A. R. Studies on critical heat flux in flow boiling at near critical pressures. **International Journal of Heat and Mass Transfer**, v. 49, p. 259–268, 2006.
- WADEKAR, V. V. **An Alternative Model for Flow Boiling Heat Transfer - in Convective Flow Boiling**, J C. Chen, p. 187-192, Washington, DC, USA: Taylor e Francis, 1996.
- WANG, G.; ZHANG, Z.; WANG, R.; ZHU, Z. A Review on Heat Transfer of Nanofluids by Applied Electric Field or Magnetic Field. **Nanomaterials**, v. 10, n. 12, p. 2386, 2020.

- WARRIER, G. R.; DHIR, V. K.; MOMODA, L. A. Heat transfer and pressure drop in narrow rectangular channels. **Experimental Thermal and Fluid Science**, v. 26, p. 53–64, 2002.
- WEISMAN, J.; PEI, B. S. Prediction of critical heat flux in flow boiling at low qualities. **International Journal of Heat and Mass Transfer**, v. 26, p. 1463–1477, 1983.
- WHALLEY, P. B. **The Importance of External Flow Boiling. Pool and External Flow Boiling** (BERGLE, A. E.; DHIR, V. K.), New York, USA: ASME, p. 411-426, 1992.
- WILCOX, D. C. **Turbulence Modeling for CFD**. 3rd ed., USA: DCW Industries, 2006.
- WINTERTON, R. H. S. Flow boiling: Prediction of bubble departure. **International Journal of Heat and Mass Transfer**, v. 27, n. 8, p. 1422–1424, 1984.
- WU, Z.; FENG, H.; CHEN, L.; XIE, Z.; CAI, C. Pumping power minimization of an evaporator in ocean thermal energy conversion system based on constructal theory. **Energy, Elsevier**, v. 181(C), p. 974-984, 2019.
- WU, Z.; FENG, H.; CHEN, L.; XIE, Z.; CAI, C. Pumping power minimization of an evaporator in ocean thermal energy conversion system based on Constructal theory. **Energy**, v. 181, p. 974 – 984, 2019.
- XIA, G.; CHAI, L.; WANG, H.; ZHOU, M.; CUI, Z. Optimum thermal design of microchannel heat sink with triangular reentrant cavities. **Applied Thermal Engineering**, v. 31, p. 6–7:1208–19, 2011.
- YANG, Z.; PENG, X. F.; YE, P. Numerical and experimental investigation of two phase flow during boiling in a coiled tube. **International Journal of Heat and Mass Transfer**, v. 51, p. 1003-1016, 2008.
- ZHANG, R.; CONG, T.; TIAN, W.; QIU, S.; SU, G. Effects of turbulence models on forced convection subcooled boiling in vertical pipe. **Annals of Nuclear Energy**, v. 80, p. 293-302, 2015.
- ZHANG, S.; SUN, Y.; YUAN, W.; TANG, Y.; TANG, H.; TANG, K. Effects of heat flux, mass flux and channel width on flow boiling performance of porous interconnected microchannel nets. **Experimental Thermal and Fluid Science**, v. 90, p. 310–318, 2018.
- ZHANG, W.; HIBIKI, T.; MISHIMA, K. Correlation for flow boiling heat transfer in mini-channels. **International Journal of Heat and Mass Transfer**, v. 47, p. 5749-5763, 2004
- ZIKANOV, O. **Essential computational fluid dynamics**. Hoboken, New Jersey, USA: John Wiley and Sons, 2010.
- ZUBER, N.; FINDLAY, J. A. Average Volumetric Concentration in Two-Phase Flow Systems. **Journal of Heat Transfer, Transactions of the ASME**, p. 453-468, 1965.
- ZUBER, N.; TRIBUS, M.; WESTWATER, J. W. The Hydrodynamic Crisis in Pool Boiling of Saturated

Liquids. **International Developments in Heat Transfer**, p. 230-236, 1963.

APPENDIX 1 – Work published in Reports in Mechanical Engineering, Vol. 3, n° 1, 2022

GEOMETRICAL INVESTIGATION OF MICROCHANNEL WITH TWO TRAPEZOIDAL BLOCKS SUBJECTED TO LAMINAR CONVECTIVE FLOWS WITH AND WITHOUT BOILING

Bruno Costa Feijó¹, Cristiano Fragassa², Luiz Alberto Oliveira Rocha³, Liércio André Isoldi¹, Sylvie Lorente⁴, Elizaldo Domingues dos Santos¹

¹ School of Engineering, Universidade Federal do Rio Grande - FURG, Av Itália, km 8, s/n, 96203-900, Rio Grande, RS, Brazil, e-mail: bruno.feijo1989@gmail.com, liercioisoldi@furg.br, elizaldosantos@furg.br

² Department of Industrial Engineering, University of Bologna, Viale Risorgimento 2, 40136, Bologna, Italy, e-mail: cristiano.fragassa@unibo.it

³ Department of Mechanical Engineering, Federal University of Rio Grande do Sul (UFRGS), Sarmiento Leite St., 425, 90040-001, Porto Alegre-RS, Brazil, e-mail: luizrocha@mecanica.ufrgs.br

⁴ Department of Mechanical Engineering, Villanova University, PA 19085, USA, e-mail: sylvie.lorente@villanova.edu

Article Info

Article history:

Received October 12, 2021
Revised November 30, 2021
Accepted December 2, 2021

Keywords:

Heat transfer
Boiling flow
Microchannel
Constructal Design
Numerical study

ABSTRACT

Microchannels are important devices to improve the heat exchange in several engineering applications as heat, ventilation and air conditioning, microelectronic cooling, power generation systems and others. The present work performs a numerical study of a microchannel with two trapezoidal blocks subjected to laminar flows, aiming to analyze the influence of the boiling process on the geometric configuration of the microchannel. Constructal Design and Exhaustive Search are used for the geometrical evaluation of the blocks. The Mixture multi-phase model and the Lee phase change model were both employed for the numerical simulation of the boiling process. In this study, the influence of the height and higher width of the first block (H_{11}/L_{11}) over the heat transfer rate and pressure drop for different magnitudes of the ratio between the lower width and higher width (L_{12}/L_{11}) was investigated. It is considered water in monophasic cases and water/vapor mixture for multiphase flow. Two different Reynolds numbers ($Re_H = 0.1$ and 10.0) were investigated. Results indicated that, for the present thermal conditions, the consideration of boiling flows were not significant for prediction of optimal configurations. Results also showed that in the cases where the boiling process was enabled, the multi-objective performance was higher than in the cases without boiling, especially for $Re_H = 0.1$.

Copyright © 2020 Regional Association for Security and crisis management and European centre for operational research.
All rights reserved.

Corresponding Author:

Cristiano Fragassa,
University of Bologna.
Email: cristiano.fragassa@unibo.it

1. Introduction

Microchannels usually have diameters between 0.01 and 0.2 mm (Kandlikar & Grande, 2003) and differ significantly to conventional channels (those with a diameter larger than 3.0 mm). They are employed in small and compact equipment like heat exchangers, microchips, nuclear reactors, bioengineering devices, among others, and have an important role in dissipating the heat that is generated by those systems. There are many studies in the last few decades about microchannels and they have focused mainly on understanding the physics of fluid flow and heat transfer and the aspects of microchannels that affect its performance such as the geometry, fluid and material properties, flow and heat parameters (Khan & Fartay, 2011; Naqiuddin et al., 2018).

The evolution of processing power in computers and the improvements of computational fluid dynamics (CFD) softwares have enabled the development and research of more complex problems, such as phase change, especially that of convective boiling flows. The boiling process happens in a microchannel when its walls are at a temperature higher than the saturation temperature, causing the formation of vapor bubbles on the liquid. This is a process that can lead to an increase in the heat transfer coefficient (Collier & Thome, 1994; Rohsenow et al., 1998) and for that reason is a desired situation in many applications that have high rates of heat dissipation. Although, the boiling process still a considerably difficult problem compared to single phase flows because of the higher complexity of the physical phenomena involved (Kandlikar et al., 1999; Khan & Fartaj, 2010). Most of the studies dedicated to numerically simulate boiling flows have focused on techniques for accurate reproduction of the boiling phenomenon (Setoodeh et al., 2020; Bathi et al., 2020; Naghibzadeh et al., 2020). Therefore, little research has been done so far with the aim of exploring the geometric aspects of microchannels subjected to boiling flows. There is strong evidence from single phase studies that the microchannels geometry can have a very important effect over the overall performance of the system (Ghani et al., 2017).

Regarding the geometry of the microchannel, the present work employs the Constructal Design method for its geometrical evaluation. This method is based on the Constructal Law of design and evolution (Bejan & Lorente, 2008; Bejan & Zane, 2012; Bejan & Lorente, 2013; Bejan, 2020) which is a physical principle where the design patterns of flow systems, animate or inanimate, can be predicted deterministically by a physical principle of maximization of access of internal currents. Constructal Law states that “the design evolves in such a way as to facilitate access to the currents that flow into the finite size flow system” so these systems can continue to exist throughout of time. Constructal Law has encountered in engineering problems a good source of application of its principles, especially in the fields of fluid dynamics and heat transfer, where it has been proved very useful for the creation of efficient designs in flow systems. There is a wide number of works that have successfully applied the Constructal Design method in those areas in the last two decades (Rocha et al., 2002; Rocha et al., 2005; Biserni et al., 2007; Bello-Ochende et al., 2009; Lorenzini et al., 2012; Rodrigues et al., 2015; Gonzales et al., 2015; Hermany et al., 2018; Rodrigues et al., 2020; Magalhães et al., 2020; Moreira et al., 2021; Nunes et al., 2021). The Constructal Design method defines the search space, taking into account constraints and degrees of freedom, allowing the evaluation of the effect of system geometry variation over the performance indicators (which indicates the way to maximize the flow access to the system). It is important to not mistake the Constructal Design with a geometric optimization method. For this purpose, the Exhaustive Search was employed, which consists on the simulation of all the geometric possibilities of the search space, considering a step variation between the simulated geometries. An explanation about some methods of optimization associated with Constructal Design can be seen in the recent work of Gonzales et al. (2021).

This work performs a numerical study of convective boiling flows inside microchannels employing a CFD software, ANSYS FluentTM 14.5, based on the finite volume method (FVM). The Mixture multiphase model and the Lee phase change model were both employed to simulate the boiling process. The problem consists of a microchannel with 0.1 mm of height and 1.5 mm of length that has its walls with a constant temperature of 380 K. The microchannel has two trapezoidal blocks mounted on its walls in an alternated form, being the first on the lower surface and the second on the upper surface, both with the same temperature imposed at the walls of the microchannel. A fluid, water in its liquid state, enters the microchannel and due to the temperature difference is forced to boil along the microchannel length. Constructal Design was applied to the geometry evaluation of the microchannel by means of four degrees of freedom and two constraints; while the Exhaustive Search was also applied for the geometrical optimization. Two different situations were analyzed and compared: with and without boiling. In addition, the geometric evaluation of both situations was done through a multi-objective approach, considering simultaneously the fluid dynamics and the thermal purposes. Therefore, the aim of this study is to analyze the influence of boiling on the effect of design over the pressure drop and heat transfer rate of the microchannel flow, as well as, to recommend geometric configurations for the problem. At the best of authors' knowledge, this kind of investigation was not previously performed in the literature, mainly by means of application of Constructal Design.

2. Mathematical Modeling

2.1. Problem Description

The present study consists on the problem of a two-dimensional microchannel with two trapezoidal blocks mounted on its walls (Figure 1) that is subjected to a convective boiling flow. The walls of microchannel, including the two blocks, are at a constant temperature of $T_w = 380$ K, which is higher than the saturation point of the fluid ($T_{sat} = 373.15$ K), water. The fluid enters the microchannel with a temperature of $T_{in} = 372$ K and is considered incompressible. Due the temperature difference, the fluid is forced to boil along the microchannel and passes through the outlet as a mixture of liquid and vapor. The microchannel outlet is at atmospheric pressure ($P_{out} = 0$ atm). The microchannel has an operating pressure of $P = 1$ atm at its surfaces, the walls and blocks, are non-slip and impermeable ($u = v = 0$ m/s).

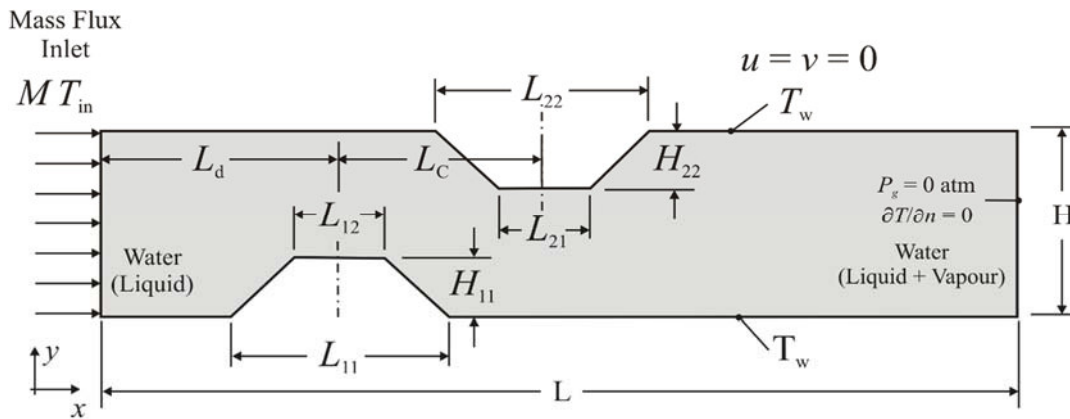


Figure 1. Computational domain of the microchannel with two heated trapezoidal blocks.

The microchannel has a length of $L = 1.5$ mm, height of $H = 0.1$ mm, $L_d = 0.4$ mm and $L_c = 0.2$ mm. To evaluate the effects of the boiling process on the geometric evaluation, two different scenarios were simulated: with and without boiling. The two situations have the same parameters and boundary conditions set to the problem and all the geometries considered in this study were simulated in both cases, the only difference is that for the cases without the boiling process the multiphase and the phase change models were deactivated. Also, for each of the two cases, two different values of mass flux were considered: $M = 1.0$ and 100.0 kg/(m².s), corresponding to the Reynolds numbers of $Re_H = 0.1$ and 10.0 , respectively. The Reynolds number was calculated by (Bejan, 2013):

$$Re_H = \frac{MH}{\mu} \quad (1)$$

where μ is the dynamic viscosity of the fluid (kg/(m.s)).

The Constructal Design method is employed for the geometrical evaluation process of the microchannel, with the definition of four degrees of freedom and two constraints. The four degrees of freedom are the ratios between the height and higher width (H_{11}/L_{11} , H_{22}/L_{22}) and between the lower and higher widths of both trapezoidal blocks (L_{12}/L_{11} , L_{21}/L_{22}). The two constraints are the areas of the two blocks. Both are set to a constant value of $2,000 \mu\text{m}^2$. The area of the two blocks can be calculated in the same way, as the area of block 1 (located on the lower surface) can be defined as:

$$A_{B1} = \frac{(L_{11} + L_{12})}{2} \cdot H_{11} \quad (2)$$

where H_{11} is the height of the first block, L_{11} is the higher width of the first block and L_{12} is the lower width of the first block.

The main purpose of the present study is to investigate the influence of the height and higher width of the first block (H_{11}/L_{11}) over the heat transfer rate (q) and over the pressure drop (ΔP) for different magnitudes of the ratio between the lower width and higher width (L_{12}/L_{11}). The objectives are the maximization of the heat transfer rate between the microchannel and the fluid and the minimization of the pressure drop in the microchannel. The heat transfer rate can be calculated as (Chen, 1966):

$$q = h_l A_s (T_w - T_{in}) + h_{NB} A_s (T_w - T_{sat}) \quad (3)$$

where A_s is the surface area of the microchannel (m^2), h_l is the heat transfer coefficient for the liquid phase flowing alone in the microchannel ($\text{W}/\text{m}^2\cdot\text{K}$) and h_{NB} is the nucleate boiling heat transfer coefficient ($\text{W}/\text{m}^2\cdot\text{K}$).

The pressure drop along the microchannel is calculated with:

$$\Delta P = P_{in} - P_{out} \quad (4)$$

where P_{in} and P_{out} are the averaged pressures in the inlet and outlet surfaces of the microchannel (Pa), respectively.

The geometric optimization of the microchannel was done with the Exhaustive Search method, according to the defined degrees of freedom. In the present work, only the first block have its dimensions varied, the second block have a constant geometry in all simulations, defined by $H_{21}/L_{21} = 0.50$ and $L_{22}/L_{21} = 0.50$. Regarding the geometry of the first block, it varies in the range of $0.1 \leq L_{12}/L_{11} \leq 1.0$ for two different magnitudes of the ratio H_{11}/L_{11} , $H_{11}/L_{11} = 0.1$ and 2.0 . For these conditions, 40 simulations were performed.

The geometric evaluation of the first trapezoidal block was performed in two steps: firstly, one of the ratios (L_{12}/L_{11}) was varied, keeping the other (H_{11}/L_{11}) fixed. This step found a maximum value for the heat transfer rate, which was called the once maximized rate, $q_{1,max}$, and, similarly, a minimum value was also found for the pressure drop between the inlet and outlet of the channel, $(\Delta P)_{1,min}$. The geometries correspondent to these values were called the once thermally optimized geometry, $(L_{12}/L_{11})_{o,T}$, and the once fluid dynamics optimized geometry, $(L_{12}/L_{11})_{o,F}$. In the second step, the same process is repeated, now for all values of H_{11}/L_{11} . Among all the simulations, a maximum heat transfer rate value is found and is called the twice maximized, $q_{2,max}$, and the same for the smallest pressure drop, called the twice minimized, $(\Delta P)_{2,min}$. These values correspond to the geometries once optimized ratio H_{11}/L_{11} for the thermal and fluid dynamics objectives, respectively, defined as: $(H_{11}/L_{11})_{o,T}$ and $(H_{11}/L_{11})_{o,F}$. Hence, it were obtained the twice optimized ratios $(L_{12}/L_{11})_{oo,T}$ and $(L_{12}/L_{11})_{oo,F}$. After, it was also performed a multi-objective analysis, were both performance indicators (thermal and fluid dynamics) were simultaneously considered with the aim of obtaining the optimum multi-objective geometry: $(L_{12}/L_{11})_{oo}$ and $(H_{11}/L_{11})_{oo}$. It is worth mentioning that the main purpose is the comparison between the cases with and without boiling and its influence on the geometrical analysis. Therefore, the investigation of only two values for the second degree of freedom is enough for the present findings.

2.2. Monophase Cases

For the cases without boiling, it was solved the conservation equation of mass and the balance of momentum equations for modeling of monophase, laminar, two-dimensional and forced convective flows. The mass conservation equation is defined by (Bejan, 2013):

$$\nabla \cdot \vec{V} = 0 \quad (5)$$

where $\vec{V}$ is the velocity vector (m/s).

The momentum conservation equations in the x and y directions, respectively, are written as follows (Bejan, 2013):

$$\nabla \cdot (\rho \vec{V} \vec{V}) = -\nabla P + \nabla \cdot (\mu \nabla \vec{V}) \quad (6)$$

where ρ is the density of the fluid (kg/m^3), P is the pressure (Pa), μ is the fluid dynamic viscosity ($\text{kg}/\text{m}\cdot\text{s}$).

The energy conservation equation is written as (Bejan, 2013):

$$\nabla \cdot (\vec{V} T) = \nabla \cdot (\lambda \nabla T) \quad (7)$$

where T is the temperature (K) and λ is the thermal diffusivity of the fluid (m^2/s).

2.3. Multiphase Cases

In the present study, the Mixture multiphase model was employed to simulate the interaction between the two phases (liquid and vapor) in the cases where boiling happens. The Mixture is a simplified version of the Eulerian model, which considers the phases as interpenetrating. It calculates the conservation equations for the mixture of the phases and the equation of the volume fraction for the secondary phases. It also has algebraic expressions for the relative and drift velocities. More details about the Mixture multiphase model can be seen in the work of Manninen et al. (1996).

The continuity conservation equation for the Mixture model is given by (Bejan, 2013):

$$\nabla \cdot (\rho_m \vec{V}_m) = 0 \quad (8)$$

where ρ_m is the mixture density (kg/m³) and $\vec{V}_m$ is the mass-averaged velocity (m/s), also known as the mixture velocity, which are given, respectively, by:

$$\rho_m = \sum_{k=1}^n \alpha_k \rho_k \quad (9)$$

$$\vec{V}_m = \frac{\sum_{k=1}^n \alpha_k \rho_k \vec{V}_k}{\rho_m} \quad (10)$$

where n is the number of phases, α_k , ρ_k and $\vec{V}_k$ are the volume fraction, density (kg/m³) and velocity (m/s) of phase k .

The balance of momentum equation is defined as (Manninen et al., 1996):

$$\nabla \cdot (\rho_m \vec{V}_m \vec{V}_m) = -\nabla p + \nabla \cdot (\mu_m \nabla \vec{V}_m) + \rho_m \vec{g} + \vec{F} - \nabla \cdot \left(\sum_{k=1}^n \alpha_k \rho_k \vec{V}_{dr,k} \vec{V}_{dr,k} \right) \quad (11)$$

where p is the pressure (Pa), $\vec{F}$ is a body force (N/m³), $\vec{g}$ is the gravity acceleration (m/s²) and μ_m is the dynamic viscosity of the mixture (N·s/m²). $\vec{V}_{dr,k}$ is the drift velocity for secondary phase k (m/s), representing the difference between the velocity of phase k and the mixture velocity:

$$\vec{V}_{dr,k} = \vec{V}_k - \vec{V}_m \quad (12)$$

The energy conservation equation for the mixture can be written as (Manninen et al., 1996):

$$\nabla \cdot \sum_{k=1}^n [\alpha_k \vec{V}_k (\rho_k E_k + p)] = \nabla \cdot (k_{eff} \nabla T) + S_E \quad (13)$$

where k_{eff} is the effective conductivity ($\sum \alpha_k k_k$), k_k is the thermal conductivity of phase k (W/m·K), S_E is a source term relative to any volumetric heat sources. For a compressible phase, the energy term E_k (J) is given by (Manninen et al., 1996):

$$E_k = h_k - \frac{p}{\rho_k} + \frac{V_k^2}{2} \quad (14)$$

and $E_k = h_k$ and for an incompressible phase, where h_k is the sensible enthalpy for phase k (J).

In the Mixture model, an additional transport equation is solved to take into account the volume fraction. This equation is expressed based on the continuity conservation equation for the secondary phase and for a primary phase named q and a secondary named p , it can be written as (Manninen et al., 1996):

$$\nabla \cdot (\alpha_p \rho_p \vec{V}_m) = -\nabla \cdot (\alpha_p \rho_p \vec{V}_{dr,p}) + \sum_{q=1}^n (\dot{m}_{qp} - \dot{m}_{pq}) \quad (15)$$

where $\dot{m}_{qp}$, $\dot{m}_{pq}$ are the rates of mass transfer between the phases (kg/m³·s).

One particularity of the Mixture model is the way it calculates the velocity of the phases. It takes into account the drift velocity and the relative velocity (also known as slip velocity), which is defined as the relative velocity between two phases (Manninen et al., 1996):

$$\vec{V}_{pq} = \vec{V}_p - \vec{V}_q \quad (16)$$

It is clear that the problem with Eq. (16) is that the velocities of the phases are unknown at principle and because of that, the relative velocity needs to be expressed in an algebraic expression, given by (Manninen *et al.*, 1996):

$$\vec{V}_{pq} = \frac{\tau_p}{f_{drag}} \frac{(\rho_p - \rho_m)}{\rho_p} \vec{a} \quad (17)$$

where τ_p is the particle relaxation time (s), d_p is the diameter of the particles (or droplets or bubbles) of secondary phase (m), $\vec{a}$ is the secondary phase acceleration of the particle (m/s²). The default drag function f_{drag} is defined by the correlation of Schiller & Naumann (1935):

$$f_{drag} = \begin{cases} 1 + 0.15 \text{Re}^{0.687} & \text{Re} \leq 1,000 \\ 0.0183 \text{Re} & \text{Re} > 1,000 \end{cases} \quad (18)$$

The Reynolds number in Eq. (18) is calculated based on the relative velocity, $\vec{V}_{pq}$ (Manninen *et al.*, 1996):

$$\text{Re} = \frac{d_p \rho_p |\vec{V}_{pq}|}{\mu_p} \quad (19)$$

Now, from Eq. (10), (12) and (16), it is possible to formulate a relation between the relative and the drift velocities:

$$\vec{V}_{dr,p} = \vec{V}_{pq} - \sum_{k=1}^n c_k \vec{V}_{kq} \quad (20)$$

where c_k is the mass fraction for any phase (k) and is defined as:

$$c_k = \frac{c_k \rho_k}{\rho_m} \quad (21)$$

In the case of a problem with only two phases, Eq. (20) can be written in a simpler manner:

$$\vec{V}_{dr,p} = \vec{V}_{pq} (1 - c_p) \quad (22)$$

Along with the Mixture model, the present study employed the Lee model (Lee, 1980) to simulate the boiling process. This model is governed by an equation that takes into account the heat transfer from both the liquid to vapor (evaporation) and vapor to liquid (condensation) phases:

$$\nabla \cdot (\alpha_v \rho_v \vec{V}_v) = \dot{m}_{lv} - \dot{m}_{vl} \quad (23)$$

where v and l represent, respectively, the vapor and liquid phases, $\dot{m}_l$ and $\dot{m}_{vl}$ are, respectively, the rates of mass transfer due to evaporation and condensation (kg/m³·s). They are given by (Lee, 1980):

$$\begin{cases} \dot{m}_{lv} = \text{coef} f * \alpha_l \rho_l \frac{(T_l - T_{sat})}{T_{sat}} & \text{If } T_l \geq T_{sat} \quad (\text{evaporation}) \\ \dot{m}_{lv} = 0 & \text{If } T_l < T_{sat} \end{cases} \quad (24)$$

$$\begin{cases} \dot{m}_{vl} = \text{coef} f * \alpha_v \rho_v \frac{(T_{sat} - T_v)}{T_{sat}} & \text{If } T_v \leq T_{sat} \quad (\text{condensation}) \\ \dot{m}_{vl} = 0 & \text{If } T_v > T_{sat} \end{cases} \quad (25)$$

where $\text{coef} f$ is the time relaxation parameter for the model of Lee (s⁻¹).

3. Numerical Modeling

The present study employed a FVM based software, ANSYS FluentTM 14.5, to resolve the numerical simulations. The SIMPLE algorithm was used for pressure-velocity coupling. A Green-Gauss Cell Based gradient and the QUICK method were used for the momentum, energy and volume fraction equations. For pressure, Body Force Weighted spatial discretization was used. All the simulations performed in this study were carried out in a steady state regime and the residuals were considered converged when reaching the value of 1.0×10^{-5} for continuity, 1.0×10^{-6} for velocities and volume fraction and 1.0×10^{-7} for energy.

To define the grid to be used in the simulations, a mesh independence study was carried out considering the most complex geometry in terms of the simulation ($H_{11}/L_{11} = 2.0$ and $L_{12}/L_{11} = 1.0$ with $Re_H = 10.0$ and phase change enabled), as shown in Table 1. Considering the mesh as independent when the relative deviation of the heat transfer rate between two successive meshes was less than 0.03% ($R = |(q^j - q^{j+1})/q^j| < 3.0 \times 10^{-4}$), it was observed that the mesh with 38,382 finite volumes achieved this criterion. It was also observed that a mesh with triangular volumes and extra refinement closer to the walls (Figure 2) was able to provide better convergence as it captured better the heat transfer phenomenon, especially in the cases with phase change.

Table 1. Mesh independence study.

Number of Volumes	q (W)	Relative Deviation
7,595	6,165.700	0.01629
12,275	6,065.214	0.00427
16,150	6,039.288	0.00997
21,046	5,979.053	0.00366
27,630	6,000.921	0.00144
38,382	6,009.571	0.00024
49,811	6,011.021	-

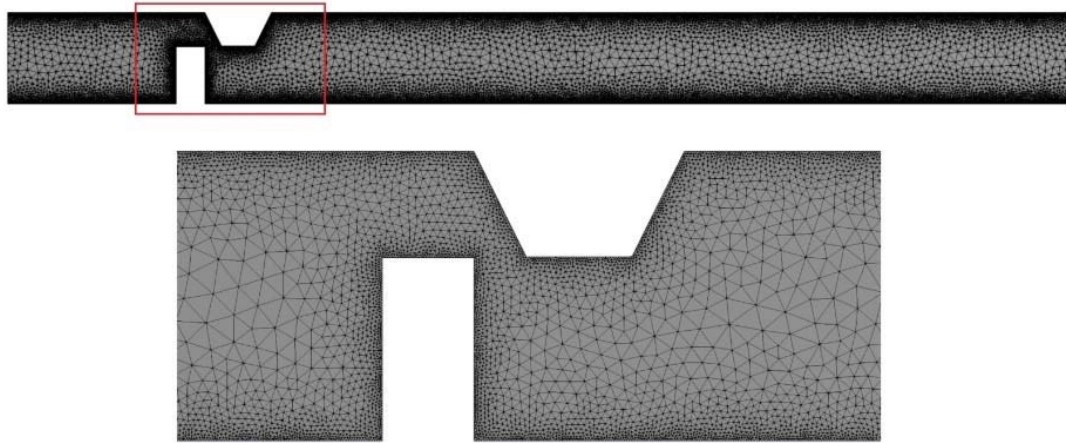


Figure 2. Mesh with triangular finite volumes employed in the present study.

To verify the numerical model employed in this study, a comparison was made with the results obtained in the numerical work of Vivekanand & Raju (2015), where a similar microchannel, with 0.1 and 1.0 mm of height and length, respectively, and without the trapezoidal blocks, was subjected to laminar boiling flows. Authors studied the microchannel in two different situations: with fixed temperature in the walls and with constant heat flux in the walls. The comparison of the vapor volume fraction generated along the microchannel can be seen in Figure 3 for the case with prescribed temperature in the walls and constant heat flux, Figure 3(a) and 3(b), respectively. It can be seen that the present study was able to achieve results very similar to those of Vivekanand & Raju (2015), with differences of less than 4 % being observed in the two cases. It is worth mentioning that the difference in the modeling of Vivekanand & Raju (2015) in comparison with the present model is the use of Volume of Fluid (VOF) method to tackle with the two phase flow

Therefore, the model employed in this study is considered adequate to simulate microchannels subjected to boiling flows in the laminar regime.

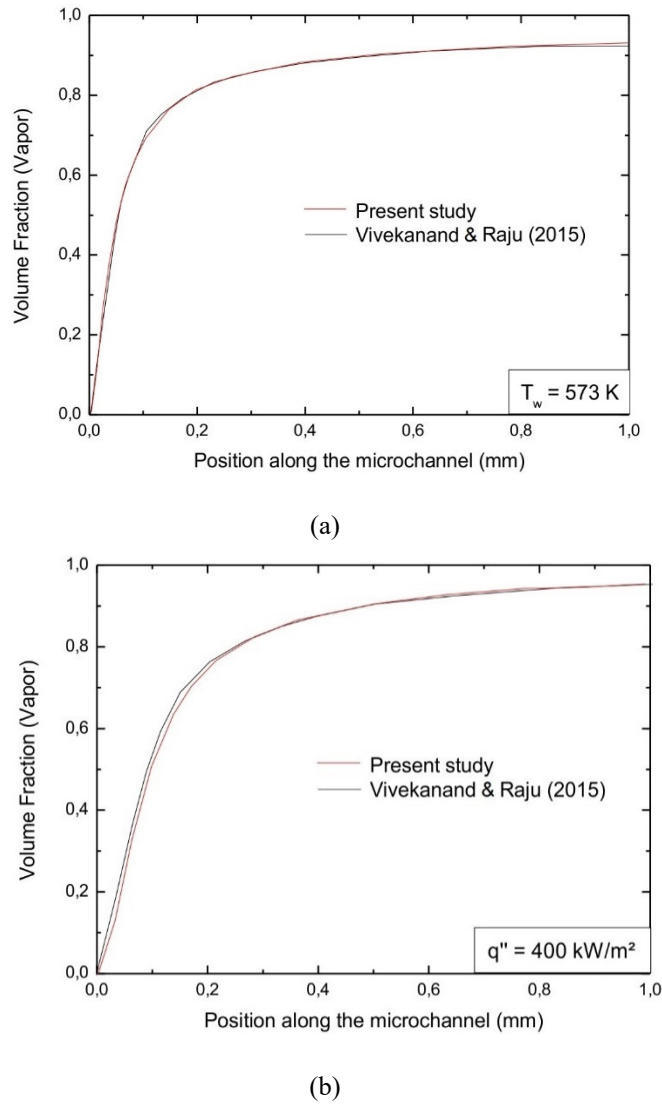


Figure 3. Verification of the model employed in this study with that of Vivekanand & Raju (2015): a) case study with prescribed temperature at the channel walls, b) case study with prescribed heat flux.

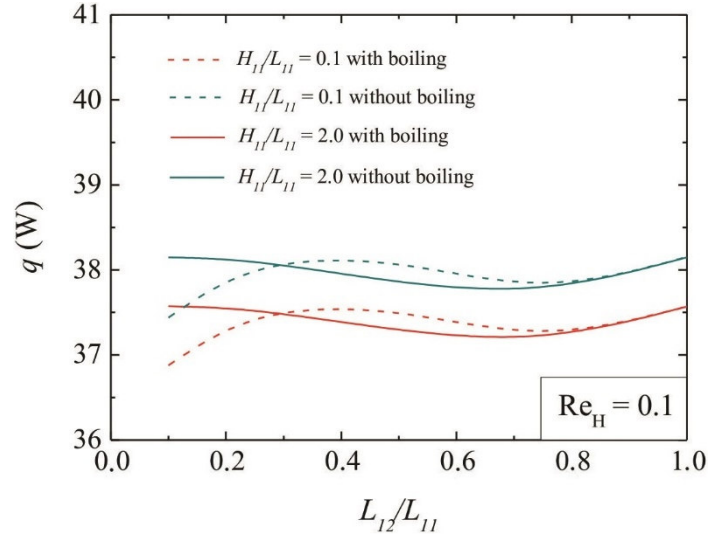
4. Results and Discussion

In this study, the results are presented in three steps: first, a thermal analysis is done, after that, a fluid dynamics analysis, and then, a multi-objective analysis. In each evaluation, it is shown the influence of the geometry (H_{11}/L_{11} and L_{12}/L_{11}) over the defined performance indicators, for $Re_H = 0.1$ and 10.0 , and it is also shown the comparison between the approaches with and without boiling.

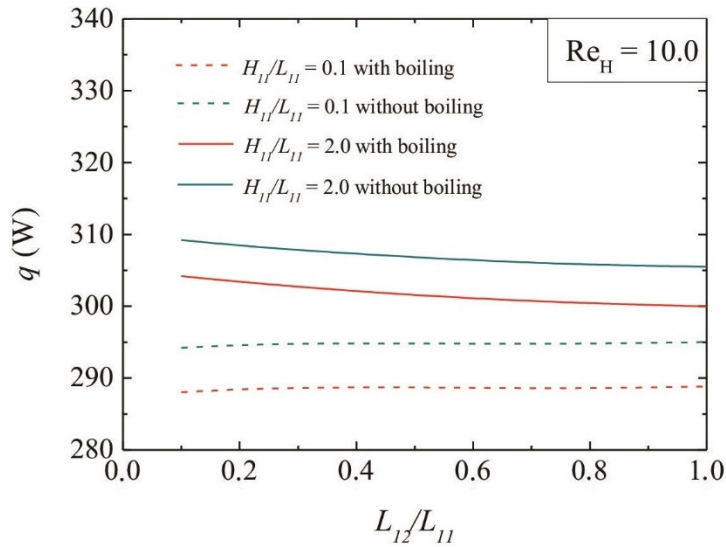
4.1. Thermal Analysis

The thermal analysis is done considering only the thermal purpose: the maximization of the heat transfer rate (q). Hence, the geometry that has the highest value of heat transfer rate is considered the optimal thermal geometry. The influence of L_{12}/L_{11} over the heat transfer rate, for different values of H_{11}/L_{11} , can be seen in Figure 4. It can be observed that the cases without boiling were able to generate slightly more heat transfer than the cases where the boiling process happened. It was noticed a difference in the heat transfer rate of around 1.5 % between the cases with and without boiling

for $Re_H = 0.1$ and between 1.6 and 2.1 % for $Re_H = 10.0$. The main reason for this behavior can be associated with the low intensity of the fluid flow for the present conditions, which does not carry the formed vapor from the microchannel walls, generating a small region where the vapor acts as a thermal insulator in the wall. For flows with higher intensity, it is expected that this tendency will change, with boiling flows leading to higher magnitudes of heat transfer rate.



(a)



(b)

Figure 4. Effect of L_{12}/L_{11} over the heat transfer rate (q) for different values of H_{11}/L_{11} and Reynolds number: (a) $Re_H = 0.1$; (b) $Re_H = 10.0$.

Concerning the geometric analysis, there is no difference in the behavior of the effect of the ratio L_{12}/L_{11} over the heat transfer rate (q) for the cases with and without boiling, i.e., for the present conditions the consideration of boiling does not led to different geometrical recommendations in comparison with the cases where boiling is disregarded.

Regarding the mass flux, it can be seen that in $Re_H = 10.0$, a more noticeable difference in the heat transfer rate is observed for the two values of H_{11}/L_{11} in comparison to the results obtained for $Re_H = 0.1$. Also, for $Re_H = 10.0$, L_{12}/L_{11} has almost no effect over the heat transfer for $H_{11}/L_{11} = 0.1$. The highest differences between the best and worst cases are

nearly 1.9 % and 5.1 % for $Re_H = 0.1$ and 10.0, respectively, for the cases without boiling, and of 1.9 % and 5.6 % for the cases with boiling.

From these results, it was possible to obtain the values of the optimal geometries concerning the thermal purpose. For the cases without boiling, they were $(H_{11}/L_{11})_{o,T} = 2.0$ and $(L_{12}/L_{11})_{oo,T} = 0.1$, the same for both values of Re_H , with a $q_{2,max} = 38.149$ W and $q_{2,max} = 309.199$ W for $Re_H = 0.1$ and 10.0, respectively. For the cases with boiling, the optimal thermal geometries were exactly the same as in the cases without boiling, with $q_{2,max} = 37.573$ W and $q_{2,max} = 304.206$ W for $Re_H = 0.1$ and 10.0, respectively.

Figure 5 shows the temperature fields for the optimal thermal geometries without boiling (a) and with boiling (b) and also the vapor volume fraction field for the case with boiling (c). It can be seen that there are no significant differences between the two situations. In both cases, the geometry with the highest value of H_{11}/L_{11} proved to be the best for increasing heat transfer. It is visible that the higher height of the first block forced the fluid to have a greater interaction with the heated walls of the microchannel. It also can be noted from Figure 5 (c) that in the cases with boiling, the vapor was being generated closer to the walls with an important contribution of the two trapezoidal blocks. The highest magnitudes of vapor volume fraction are reached in the regions of lower basis of the two trapezoidal blocks. Specifically for the present conditions, the vapor generation in stagnation regions of the trapezoidal blocks led to the insulation of the wall and meagered of heat transfer rate.

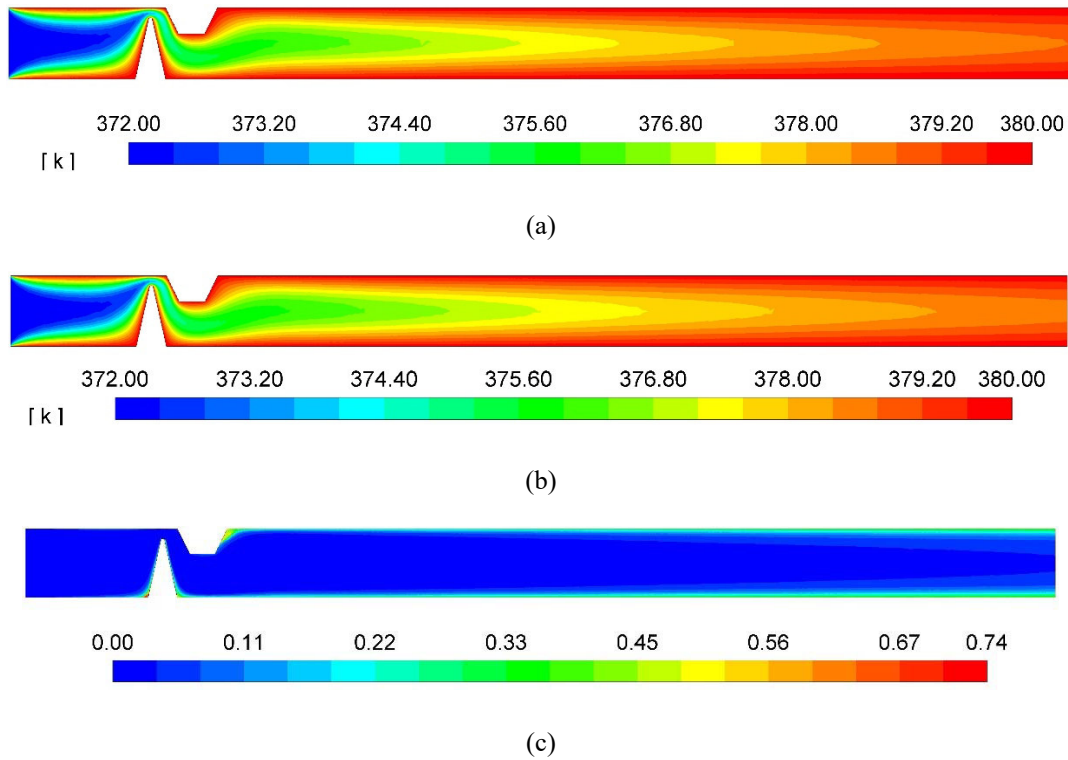
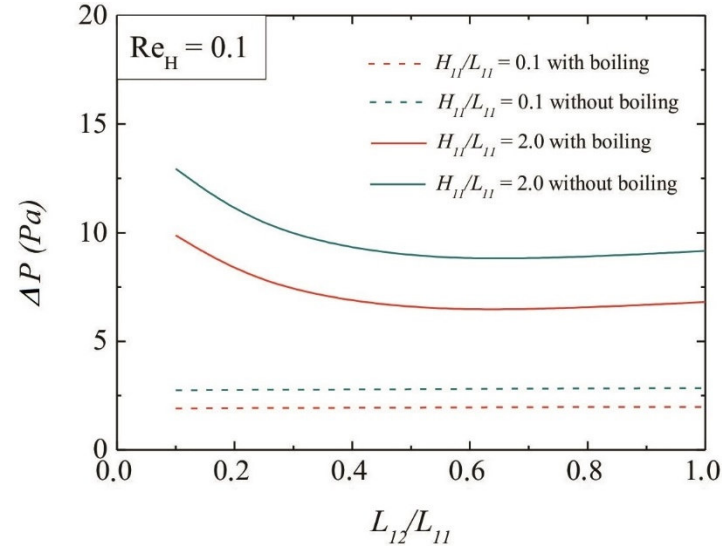


Figure 5. Temperature (K) fields for $Re_H = 10.0$, $L_{12}/L_{11} = 0.1$ and $H_{11}/L_{11} = 2.0$: (a) without boiling; (b) with boiling; (c) vapor volume fraction field for the case with boiling.

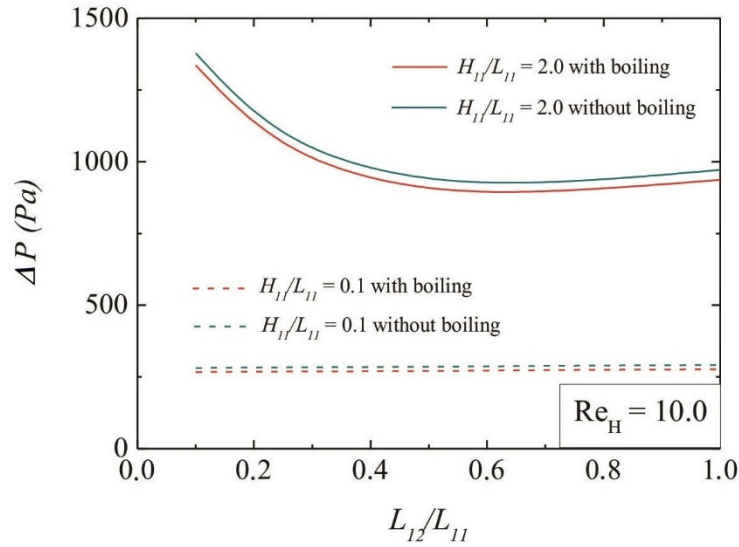
4.2. Fluid Dynamics Analysis

The fluid dynamics analysis is done considering the difference in the pressures between the inlet and the outlet, i.e., the pressure drop along the microchannel (ΔP), as the performance indicator. The purpose is to determine the optimal fluid dynamics geometry, the one with the lowest value of pressure drop. Figure 6 shows the results for $Re_H = 0.1$ (a) and 10.0 (b), with the influence of L_{12}/L_{11} over the pressure drop, for different values of H_{11}/L_{11} . Similar to what was observed in the thermal analysis, there was no difference in the behavior of the effect of the geometry over the pressure drop between the cases with and without boiling. Nevertheless, now the best results are from the cases with the boiling process enabled. Probably, this behavior can be associated with the generation of some portion of vapor, which is lighter than the monophasic water, being transported in more easy way. It can be seen that differences in the pressure drop between with and without boiling decreased significantly from $Re_H = 0.1$ to $Re_H = 10.0$, although the cases with boiling were still able

to perform better than those without boiling for any geometry simulated in the present study. Regarding the differences in the pressure drop, they stayed between 31 and 44 % comparing the cases with and without boiling for $Re_H = 0.1$ and between 3.1 and 5.5 % for $Re_H = 10.0$, considerably higher than those differences observed in the thermal analysis.



(a)



(b)

Figure 6. Effect of L_{12}/L_{11} over the pressure drop (ΔP) for different values of H_{11}/L_{11} and Reynolds number: (a) $Re_H = 0.1$; (b) $Re_H = 10.0$.

One can also observe in Figure 6 that L_{12}/L_{11} has almost no effect in the pressure drop for $H_{11}/L_{11} = 0.1$ but has a significant influence when $H_{11}/L_{11} = 2.0$, where the first block has much higher height. For $H_{11}/L_{11} = 2.0$, an optimal geometry is obtained with an intermediate value of $L_{12}/L_{11} = 0.5$ for both $Re_H = 0.1$ and 10.0 . The best fluid dynamics cases were 5.01 and 5.17 times better than the worst cases, for $Re_H = 0.1$ and 10.0 , respectively, for the cases without boiling, and 4.71 and 4.90 times better for the cases with boiling, notably higher than what was seen in the thermal analysis.

From these results, the values of the optimal fluid dynamics geometries were, for the cases without boiling, $(H_{11}/L_{11})_{o,F} = 0.1$ and $(L_{12}/L_{11})_{o,F} = 0.1$, the same for both values of Re_H , with a $(\Delta P)_{2,min} = 2.75$ Pa and $(\Delta P)_{2,min} = 281.30$ Pa for $Re_H = 0.1$ and 10.0 respectively. As occurred in the thermal analysis, for the cases with boiling, the optimal fluid dynamics geometries were the same as in the cases without boiling, with $(\Delta P)_{2,min} = 1.91$ Pa and $(\Delta P)_{2,min} = 266.63$ Pa for $Re_H = 0.1$ and 10.0, respectively.

Figure 7 shows the velocity and pressure fields for the optimal fluid dynamics geometries without (a) and with boiling (b). It can be seen that in the case without boiling, the highest value of the flow velocity is located between the two trapezoidal blocks and is about 0.0023 m/s. After passing through the two blocks, the fluid keeps an almost constant velocity profile. In the case with boiling, an acceleration is also observed around the two blocks, although with a higher velocity (about 0.0037 m/s). For the case with boiling, the flow has a further increase in its velocity along the whole microchannel that continues to accelerate up until the outlet. This fact reduces the pressure drop in the microchannel, increasing the fluid dynamics performance. This behavior can be credited to the much lower density of the vapor phase, which has an effect of increasing the fluid velocity, and is normally seen in boiling flows. Also, the gap in the results of the pressure drop observed between the two values of H_{11}/L_{11} , in both with and without boiling cases, happened due to the stricture effect occurred in the flow for $H_{11}/L_{11} = 2.0$. The constraint cause a significant increase in the velocity and in the pressure in that region, which decreased the fluid dynamics performance for those geometries compared to $H_{11}/L_{11} = 0.1$.

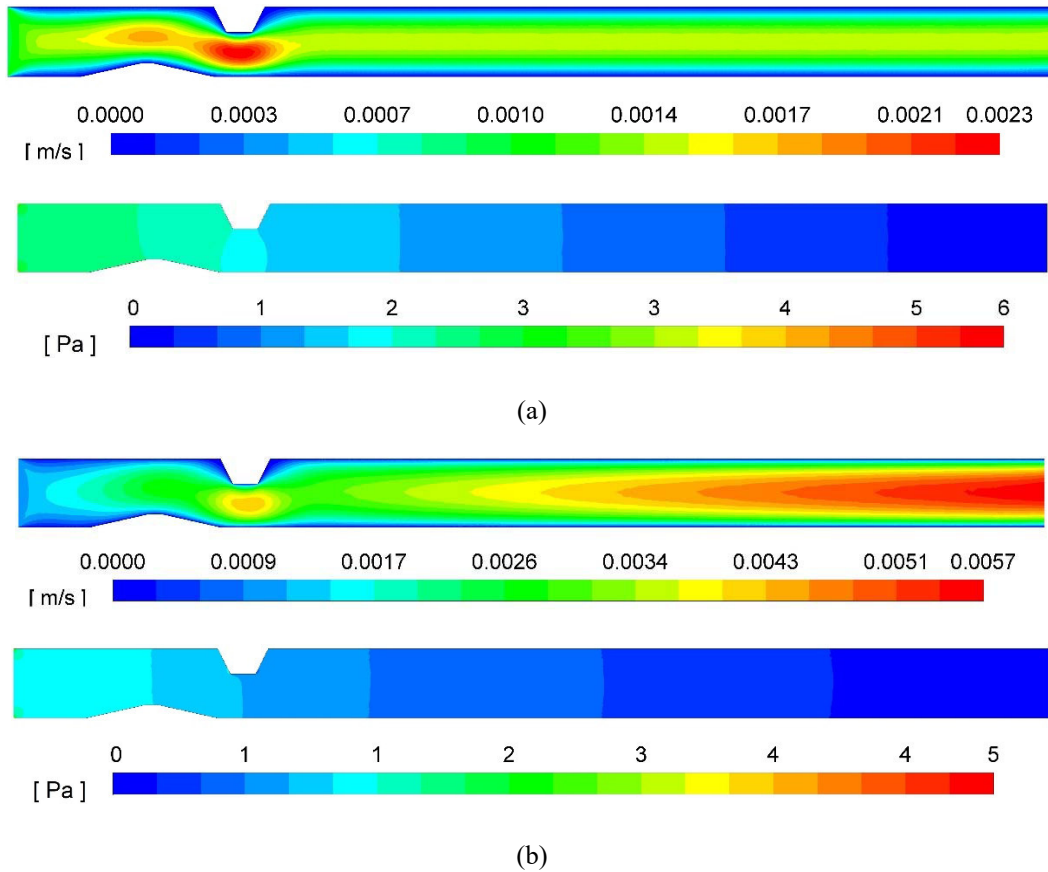


Figure 7. Velocity (m/s) and Pressure (Pa) fields for $Re_H = 0.1$, $L_{12}/L_{11} = 0.1$, $H_{11}/L_{11} = 0.1$ and: (a) without boiling; (b) with boiling.

4.3. Multi-objective Analysis

The final part of the geometric evaluation process is carried out considering simultaneously the two performance indicators in order to determine the optimum multi-objective geometry. To do so, the graphics presented in Figure 8 were plotted considering the results obtained for the pressure drop (ΔP) as a function of the inverse of the heat transfer rate (q) results. Therefore, the optimal multi-objective geometries are those which the point in the graphic is closer to the point where both axes are equal to zero, which represents an idealized problem with minimum pressure drop and maximum

heat transfer rate. As can be seen in Figure 8, the optimal geometries with $H_{11}/L_{11} = 0.1$ proved to have a considerable higher multi-objective performance compared to those with $H_{11}/L_{11} = 2.0$, for both mass fluxes and boiling conditions. It is also possible to note that the heat transfer rate is almost unaffected by H_{11}/L_{11} for $Re_H = 0.1$. For $Re_H = 10.0$, H_{11}/L_{11} seems to have some effect over heat transfer rate, though still much lower than that of the pressure drop. Overall, this multi-objective analysis showed that the defined degrees of freedom had a much greater influence over pressure drop than over the heat transfer rate.

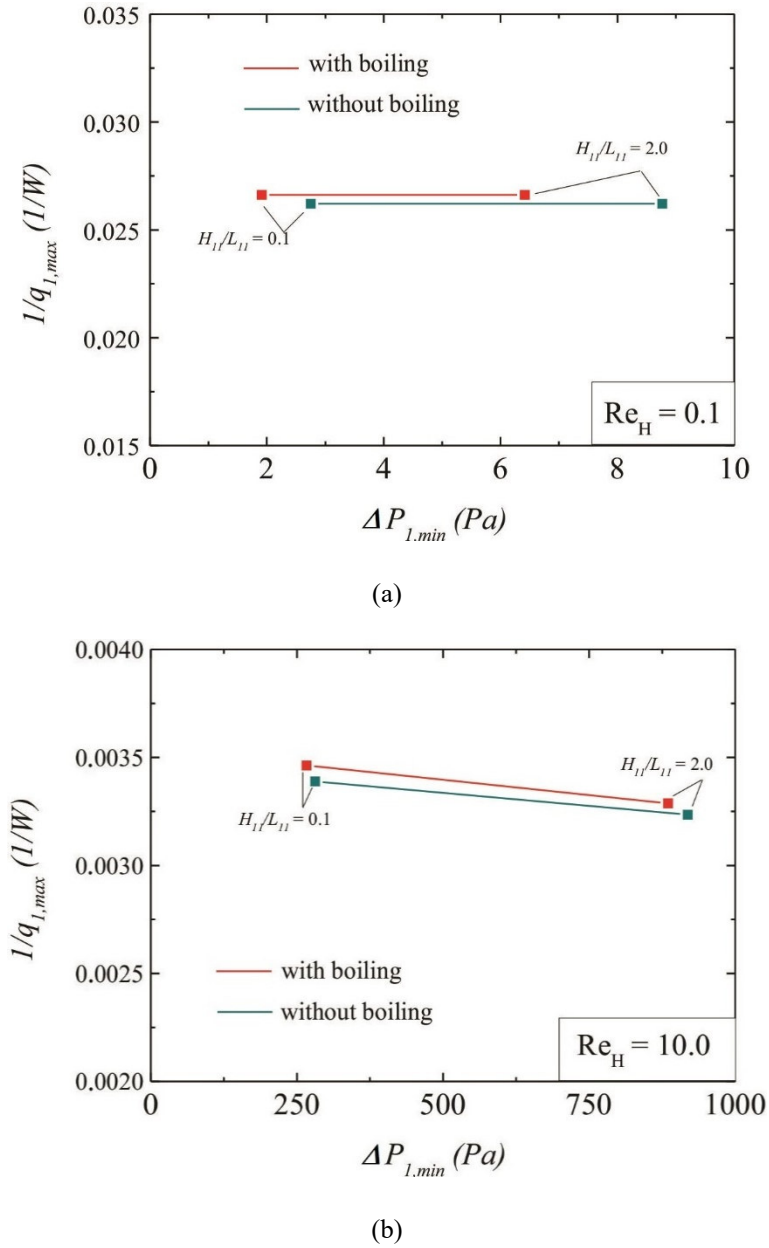


Figure 8. Effect of the once minimized pressure drop $(\Delta P)_{1,min}$ over the inverse of the once maximized heat transfer $(1/q)_{1,max}$ for different values of H_{11}/L_{11} and Reynolds number: (a) $Re_H = 0.1$; (b) $Re_H = 10.0$.

Table 2 shows a compilation of the overall optimal multi-objective geometries from the results obtained in this study, $(H_{11}/L_{11})_o$, and $(L_{12}/L_{11})_{oo}$. The optimal geometries found for both Re_H for the cases with and without boiling were the same, i.e., the one with the lowest values of H_{11}/L_{11} and L_{12}/L_{11} between all geometries evaluated in this study, and is also the same optimal geometry considering only the fluid dynamics objective.

Table 2. Optimal shapes for multi-objective problem

Re_H	Cases without boiling		Cases with boiling	
	0.1	10.0	0.1	10.0
$(H_{11}/L_{11})_o$	0.1	0.1	0.1	0.1
$(L_{12}/L_{11})_{oo}$	0.1	0.1	0.1	0.1
H_{11}	19.07 μm	19.07 μm	19.07 μm	19.07 μm
L_{11}	190.69 μm	190.69 μm	190.69 μm	190.69 μm
L_{12}	19.07 μm	19.07 μm	19.07 μm	19.07 μm

5. Conclusions

The present study performed numerical simulations of a microchannel with two trapezoidal blocks, subjected to convective boiling water flows, applying the Constructal Design method and Exhaustive Search to the geometrical evaluation of the microchannel. To investigate the influence of the boiling process over the geometrical evaluation, two different cases were evaluated: with and without boiling. Furthermore, each of those two situations were divided again into two different values of mass flux, 1.0 and 100.0 kg/(m²·s), corresponding to $Re_H = 0.1$ and 10.0, respectively. In this study, it was investigated the influence of the height and higher width of the first block (H_{11}/L_{11}) over the heat transfer rate (q) and the pressure drop (ΔP) for different magnitudes of the ratio between the lower width and higher width of the trapezoidal block (L_{12}/L_{11}). The geometry of the second block was kept constant throughout all simulations. The purposes of the geometric evaluation were to find geometries that would maximize the heat transfer rate (q) and minimize the pressure drop (ΔP). Thermal and fluid dynamics analyzes were carried out, as well as a multi-objective analysis.

The results showed that, for the conditions applied in the present study, the boiling process decreased the heat transfer rate approximately 1.6 % for $Re_H = 0.1$ and between 1.6 and 2.1 % for $Re_H = 10.0$, which was not intuitively expected in a first moment. This behavior can be related with the generation of vapor near the microchannel surfaces, mainly in the region of higher width of the trapezoidal blocks, where the fluid is stagnated due to low magnitude of Reynolds number. Despite of some differences in the magnitude of q , the presence of the boiling process did not generate any significant behavior change of the effect of the geometry (H_{11}/L_{11} and L_{12}/L_{11}) over the heat transfer rate. The best thermal performance was obtained for the highest intrusion of the trapezium in the channel, $(H_{11}/L_{11})_{o,T} = 2.0$ and lowest ratio $(L_{12}/L_{11})_{oo,T} = 0.1$ for the cases with and without boiling.

The fluid dynamics analysis showed a better performance for cases with the boiling process enabled, especially for $Re_H = 0.1$. In general, the differences in the results obtained for the pressure drop between the cases with and without boiling were between 31 and 44 % for $Re_H = 0.1$ and between 3.1 and 5.5 % for $Re_H = 10.0$, considerably higher than those differences observed in the thermal analysis. From the velocity fields, it was also possible to visualize that the fluid velocity profile along the microchannel accelerates in the cases with the boiling process, which does not happen in the cases without boiling. This was attributed to the much lower density of the vapor phase that tends to accelerate the whole fluid along microchannel, as it normally happens in boiling flows inside channels and microchannels. The optimal fluid dynamics geometry was the one where the first block causes the least obstruction to the flow, $(H_{11}/L_{11})_{o,F} = 0.1$ and $(L_{12}/L_{11})_{oo,F} = 0.1$, the same for both values of Re_H and for both with and without boiling.

The multi-objective analysis proved that the geometry of the first block had a considerable higher effect over the pressure drop than over the heat transfer rate and also that the cases with boiling had a better multi-objective performance than those without boiling, for the fluid dynamics and thermal conditions studied here. The optimal geometries found for both Re_H and for the cases with and without boiling were exactly the same, $(H_{11}/L_{11})_o = 0.1$ and $(L_{12}/L_{11})_{oo} = 0.1$, which is the same geometry as the optimal fluid dynamics geometric. Moreover, it is noticeable that the advantage in performance that the cases with the boiling process have for $Re_H = 0.1$ slightly decreases for $Re_H = 10.0$, although they still better than the cases without boiling. Another import observation is that for $Re_H = 10.0$ there was a slight increase in the share of heat transfer in the multi-objective, which leads to the conclusion that there is a possibility that an increase in the Reynolds number above this value can lead to an augmentation of this trend.

In general, the results demonstrated for lower magnitudes of Re_H that the consideration of boiling had not important influence over the design prediction of microchannel for the present thermal conditions and for the degrees of freedom investigated here. In spite of that, the design investigation proved to be important for improvement on the overall performance of microchannels heat exchangers. For future studies, a complete investigation with the four degrees of freedom and under other flow conditions will be performed.

Acknowledgement: E.D. dos Santos thanks FAPERGS - *Fundação de Amparo à Pesquisa do Estado do Rio Grande do Sul* for financial support by the Public Call N° 05/2019 – PqG (Grant number: 19/2551-0001847-9). The authors L.A.

Isoldi, L.A.O. Rocha and E.D. dos Santos thank CNPq – *Conselho Nacional de Desenvolvimento Científico e Tecnológico* for research grant (Processes: 306012/2017-0, 307791/2019-0, 306024/2017-9).

References

- Bhati, J., Paruya, S., & Naik L, J. (2020). Numerical Simulation of Bubble Dynamics in Subcooled Flow Boiling in a Channel. *Nuclear Engineering and Design*, 371, 110945.
- Bejan, A. (2013). *Convection Heat Transfer*. Hoboken, USA: John Wiley & Sons, Inc.
- Bejan, A. (2020). *Freedom and Evolution, Hierarchy in Nature, Society and Science*. Cham, Switzerland: Springer.
- Bejan, A., & Lorente, S. (2008). *Design with Constructal Theory*, Hoboken, New Jersey, USA: John Wiley & Sons.
- Bejan, A., & Lorente, S. (2013). Constructal law of design and evolution: Physics, biology, technology, and society. *Journal of Applied Physics*, 113, No. 151301.
- Bejan, A., & Zane, J. P. (2012). *Design in Nature*. New York, USA: Doubleday.
- Bello-Ochende, T., Meyer, J. P., & Bejan, A. (2009). Constructal Ducts with wrinkled entrances. *International Journal of Heat and Mass Transfer*, 52, 3628-3633.
- Biserni, C., Rocha, L. A. O., Stanescu, G., & Lorenzini, E. (2007). Constructal H-shaped cavities according to Bejan's theory. *International Journal of Heat and Mass Transfer*, 50, 2132-2138.
- Chen, J. C. (1966). Correlation for Boiling Heat Transfer to Saturated Fluids in Convective Flow. *Industrial and Engineering Chemistry Process Design and Development*, 5, 322.
- Collier, J.G., & Thome, J.R. (1994). *Convective Boiling and Condensation*. (3rd ed.) Oxford, United Kingdom: Clarendon Press.
- Feijó, B. C., Lorenzini, G., Isoldi, L. A., Rocha, L. A. O., Goulart, J. N. V., & Dos Santos, E. D. (2018). Constructal design of forced convective flows in channels with two alternated rectangular heated bodies. *International Journal of Heat and Mass Transfer*, 125, 710-721.
- FLUENT, Documentation Manual – FLUENT 14.5 (2013).
- Ghani, I. A., Sidik, N. A. C., & Kamaruzaman, N. (2017). Hydrothermal performance of microchannel heat sink: The effect of channel design. *International Journal of Heat and Mass Transfer*, 107, 21–44.
- Gonzales, G. V., Estrada, E. da S. D., Emmendorfer, L. R., Isoldi, L. A., Xie, G., Rocha, L. A. O., & Dos Santos, E. D. (2015). A Comparison of Simulated Annealing Schedules for Constructal Design of Complex Cavities Intruded into Conductive Walls with Internal Heat Generation. *Energy*, 93, 372 – 382.
- Gonzales, G. V., Lorenzini, G., Isoldi, L. A., Rocha, L. A. O., Dos Santos, E. D., & Silva Neto, A.J. (2021). Constructal Design and Simulated Annealing Applied to the Geometric Optimization of an Isothermal Double T-Shaped Cavity. *International Journal of Heat and Mass Transfer*, 174, 121268.
- Hermani, L., Lorenzini, G., Klein, R. J., Zinani, F. F., Dos Santos, E. D., Isoldi, L. A., & Rocha, L. A. O. (2018). Constructal Design Applied to Elliptic Tubes in Convective Heat Transfer Cross-Flow of Viscoplastic Fluids. *International Journal of Heat and Mass Transfer*, 116, 1054 – 1063.
- Kandlikar, S. G., & Grande, W. J. (2003). Evolution of Microchannel Flow Passages – Thermohydraulic Performance and Fabrication Technology. *Heat Transfer Engineering*, 25, 1.
- Kandlikar, S. G., Shoji, M., & Dhiri, V. K. (1999). *Handbook of Phase Change: Boiling and Condensation*. Philadelphia, USA: Taylor & Francis.
- Karayannis, T. G., & Mahmoud, M. M. (2017). Flow boiling in microchannels: Fundamentals and applications. *Applied Thermal Engineering*, 115, 25, 1372-1397.
- Khan, M. G., & Fartay, A. (2011). A review on microchannel heat exchangers and potential applications. *Int. J. Energy Res.*, 35, 553–582.

- Lee, W.H. (1980). A Pressure Iteration Scheme for Two-Phase Flow Modeling. *Multiphase Transport Fundamentals, Reactor Safety, Applications*. Washington, DC, USA.
- Lorenzini, G., Garcia, F. L., Dos Santos, E. D., Biserni, C., & Rocha, L. A. O. (2012). Constructal Design Applied to the optimization of complex geometries: T-Y-shaped cavities with two additional lateral intrusions cooled by convection. *International Journal of Heat and Mass Transfer*, 55, 1505-1512.
- Magalhães, G.C., Fragassa, C., Lemos, R.L., Isoldi, L.A., Amico, S.C., Rocha, L.A.O., Souza, J.A., & Dos Santos, E.D. (2020). Numerical Analysis of the Influence of Empty Channels Design on Performance of Resin Flow in a Porous Plate. *Applied Sciences*, 10, 4054.
- Manninen, M., Taivassalo, V., & Kallio, S. (1996). *On the Mixture Model for Multiphase Flow*. VTT Publications.
- Moreira, R.S.M., Escobar, C.C., Isoldi, L.A., Davesac, R.R., Rocha, L.A.O., & Dos Santos, E.D. (2021). Numerical Study and Geometric Investigation of Corrugated Channels Subjected to Forced Convective Flows. *Journal of Applied and Computational Mechanics*, 7, 727-738.
- Naghizadeh, S.M., Goharkhah, M., Sharifpur, M., & Meyer, J.P. (2020). Effects of Interphase Momentum Exchange Models on Simulation of Subcooled Flow Boiling. *International Communications in Heat and Mass Transfer*, 118, 104863.
- Naqiuddin, N.H., Saw, L.H., Yew, M.C., Yusof, F., Ng, T.C., & Yew, M.K. (2018). Overview of Micro-Channel Design for High Heat Flux Application. *Renewable and Sustainable Energy Reviews*, 82, 901–914.
- Nunes, B.R., Rodrigues, M.K., Rocha, L.A.O.; Labat, M., Lorente, S., Dos Santos, E.D., Isoldi, L.A., & Biserni, C. (2021). Numerical-Analytical Study of Earth-Air Heat Exchangers with Complex Geometries Guided by Constructal Design. *International Journal of Energy Research*, 45, 20970-20987.
- Ohadi, M., Choo, K., Dessiatoun, S., & Cetegen, E. (2012). Emerging Applications of Microchannels. *Briefs in Applied Sciences and Technology*, 67–105.
- Rai, S.K., Sharma, R., Saifi, M., Tyagi, R., Singh, D., & Gupta, H. (2018). Review of recent applications of micro channel in mems devices. *International Journal of Applied Engineering Research*, 13, 9, 64-69.
- Rocha, L. A. O., Lorente, S., & Bejan, A. (2002). Constructal design for cooling a disc-shaped area by conduction. *International Journal of Heat and Mass Transfer*, 45, 8, 1643-1652.
- Rocha, L. A. O., Lorenzini, E., & Biserni, C. (2005). Geometric optimization of shapes on basis of Bejan's Constructal Theory. *International Journal of Heat and Mass Transfer*, 32, 1281-1288.
- Rodrigues, M. K., Brum, R. B., Vaz, J., Rocha, L. A. O., Dos Santos, E.D., & Isoldi, L.A. (2015). Numerical investigation about the improvement of the thermal potential of an Earth-Air Heat Exchanger (EAHE) employing the Constructal Design method. *Renewable Energy*, 80, 538-551.
- Rodrigues, P.M., Biserni, C., Escobar, C.C., Rocha, L.A.O., Isoldi, L.A., & Dos Santos, E.D. (2020). Geometric Optimization of a Lid-Driven Cavity with Two Rectangular Intrusions under Mixed Convection Heat Transfer: A Numerical Investigation Motivated by Constructal Design. *International Communications in Heat and Mass Transfer*, 117, 104759.
- Rohsenow, W.M., Hartnett, J.P., & Cho, Y.I. (1998). *Handbook of Heat Transfer*. (3rd ed).USA: McGraw-Hill.
- Schiller, L., & Nauman, A. (1935). A drag coefficient correlation. *VDI Zeitung*, 77, 318–320.
- Setoodeh, H., Ding, W., Lucas, D., & Hampel, U. (2020). Modelling and Simulation of Flow Boiling with an Eulerian-Eulerian Approach and Integrated Models for Bubble Dynamics and Temperature-Dependent Heat Partitioning. *International Journal of Thermal Sciences*, 161, 106709.
- Siddiqui, O.K., & Zubair, S.M. (2017). Efficient energy utilization through proper design of microchannel heat exchanger manifolds: A comprehensive review. *Renewable and Sustainable Energy Reviews*, 74, 969–1002.
- Song, Y., Asadi, M., Xie, G., & Rocha, L.A.O. (2015). Constructal wavy-fin channels of a compact heat exchanger with heat transfer rate maximization and pressure losses minimization. *Applied Thermal Engineering*, 75, 24-32.

Vivekanand, S. V. B., & Raju, V. R. K. (2015). Simulation of Evaporation Heat Transfer in Rectangular Microchannel. *Procedia Engineering*, 127, 309-316.