



UNIVERSITÀ
DEGLI STUDI
DI PADOVA

Sede Amministrativa: Università degli Studi di Padova

Dipartimento di INGEGNERIA INDUSTRIALE

SCUOLA DI DOTTORATO DI RICERCA IN: INGEGNERIA INDUSTRIALE

INDIRIZZO: INGEGNERIA DELL'ENERGIA

CICLO XXVII

**ENHANCEMENT OF SINGLE- AND TWO- PHASE HEAT TRANSFER
INSIDE HEAT GENERATORS**

Direttore della Scuola: Prof. Paolo Colombo

Coordinatore d'Indirizzo: Prof. Luisa Rossetto

Supervisore: Prof. Davide Del Col

Dottorando: Alberto Bisetto

CONTENTS

CONTENTS	1
ABSTRACT	5
RIASSUNTO	7
LIST OF FIGURES	9
LIST OF TABLES	17
INTRODUCTION	19
1. EXPERIMENTAL AND THEORETICAL ANALYSIS OF HEAT TRANSFER INSIDE HEAT GENERATORS	23
1.1. Introduction	23
1.2. Experimental investigation	25
1.2.1. Description of the experimental apparatus	25
1.2.2. Experimental tests	29
1.2.3. Uncertainty analysis	31
1.2.4. Experimental results	32
1.3. Dynamic model of the heat generator	34
1.3.1. General description	35
1.3.2. Model variables	36
1.3.3. Flue gases subsystems	36
1.3.4. Walls subsystems	39
1.3.5. Water subsystem	40
1.3.6. Model flow chart	41
1.3.7. Modeling the presence of turbulators	42
1.4. Model validation	44
1.5. Model predictions with different geometries	48
1.6. Conclusions	54
Nomenclature	56
2. SINGLE-PHASE HEAT TRANSFER ENHANCEMENT BY MEANS OF TURBULATORS	59
2.1. Introduction	59
2.2. State of the art about turbulators analysis	61
2.3. Actual geometries	65
2.4. CFD modeling	67

2.5. Data reduction	69
2.6. Results	70
2.6.1.Theoretical analysis	71
2.6.2.Validation of the results through the heat generator model	85
2.7. Modeling of a new geometry	85
2.7.1. Description of the geometry	85
2.7.2. CFD model	86
2.7.3. Results	88
2.8. Conclusions	91
Nomenclature	93
3. TWO-PHASE HEAT TRANSFER ENHANCEMENT BY MEANS OF NANO-ENGINEERED SURFACES	95
3.1. Introduction	95
3.2. Theoretical background	97
3.2.1. Wetting regimes over smooth and rough surfaces	97
3.2.2. Contact angles theory	102
3.2.3. How to get superhydrophobicity	104
3.2.4. Dropwise condensation phenomena	106
3.3. State of the art about superhydrophobic aluminum surfaces	109
3.4. State of the art about dropwise condensation analysis	110
3.5. Experimental setup	113
3.5.1. Two-phase flow loop description	113
3.5.2. Description of the test section	118
3.5.3. Data reduction	122
3.5.4. Instruments calibration	124
3.5.5. Uncertainty analysis	125
3.5.6. Calibration tests	126
3.6. Condensation over standard aluminum surfaces	128
3.6.1. Experimental analysis	128
3.6.2. Theoretical analysis	130
3.7. Modification of wetting properties in aluminum surfaces	133
3.7.1. Materials	133
3.7.2. Superhydrophilic sample preparation	134
3.7.3. Hydrophobic sample preparation	134

3.7.4. Superhydrophobic sample preparation	134
3.7.5. Characterization of the surface	135
3.8. Condensation over engineered surfaces	141
3.8.1. FWC over superhydrophilic surfaces	141
3.8.2. DWC over superhydrophobic surfaces	143
3.8.3. Enhanced FWC over hydrophobic surfaces	148
3.8.4. Theoretical analysis of slip-driven FWC	151
3.9. Conclusions	155
Nomenclature	159
CONCLUSIONS	161
REFERENCES	165
PUBLICATIONS	171
ACKNOWLEDGEMENTS	173

ABSTRACT

Heat generators are actually the most used system for producing domestic heat and hot water. These systems are most commonly fire tube heat generators, which consist in a shell-and-tube heat exchanger with the flue gases produced by a stationary combustion process flowing inside the tubes, and the secondary fluid, i.e. water, located in the shell.

The first Chapter of this Thesis presents an experimental and theoretical analysis of the working conditions of a three pass fire tube heat generator, while operating both in stationary and transient regimes. In the literature, at the best of the author's knowledge, very limited research is published on such systems.

Experimental tests were performed varying heat generator working conditions as well as inserting or not turbulence generator inserts inside the tube composing the last flue gases pass.

A dynamic MatLab/Simulink model for the fire tube heat generator is presented. This has been validated using the new experimental dataset. The model is characterized by a subsystem structure which makes it easily adaptable to different geometries, and it can be used to predict the behavior of heat generators working in stationary and transient conditions.

Heat generators performances can be increased by enhancing heat transfer between the flue gases and the water inserting turbulence generators in the tube where the combustion products flow. In particular, inserting turbulators inside the last pass of fire tube heat generators leads to an increase of the system efficiency, due to reduced flue gases exit temperature which implies reduced thermal losses at the chimney. Thus, single phase heat transfer enhancement by means of turbulence generator inserts is an important field of research for the heat transfer industry.

However, it must be kept in mind that the convective heat transfer coefficient enhancement due to the turbulators presence is associated with an augmentation of the frictional losses in the system, thus both factors have to be taken into account when evaluating inserts performances.

In the second part of the present Thesis, after a review of the most common solutions presented in the literature, performance enhancement of turbulators geometries actually used by the heat generators manufacturers is evaluated by means of CFD simulations. Effects of several geometrical parameters, such as turbulator position inside the tube and pipe diameter, are analyzed with the simulations. Equations for predicting the inserts working conditions are presented. Finally, a modified geometry is presented with the aim of proposing a solution with enhanced thermal and frictional characteristics.

By recovering the latent heat in the exhaust products coming out from the heat generator it is possible to achieve extremely high system efficiencies. Thus, two-phase heat transfer is a field of interest in the heat generators industry.

During pure steam condensation the mayor resistance to the thermal transport is associated to the condensate layer which forms at the wall. Reducing, or eventually removing, the liquid film thickness at the wall will thus lead to an enhancement of the condensation heat transfer coefficient, increasing the heat transfer per unit area. This means the possibility to get higher performances of the system maintaining the same geometry, or to reduce the heat transfer area (thus the system cost) maintaining the same net effect.

The last Section of the present Thesis is focused on the enhancement of the condensation heat transfer coefficient by means of nano-engineered surfaces. Effect of surface wetting properties on the condensation process and performance is analyzed by studying the behavior of conventional, superhydrophilic, hydrophobic and superhydrophobic surfaces during pure steam flow condensation at different mass velocities. The aim of the research is to remove (dropwise condensation mode) or eventually reduce (filmwise condensation mode) the condensate layer over the wall during the two-phase process, by acting on surface superficial characteristics, as well as to evaluate the effect of surface roughness on the filmwise condensation heat transfer coefficient.

RIASSUNTO

I generatori di calore sono attualmente i sistemi più utilizzati nelle applicazioni di riscaldamento domestico. Questi sono di norma generatori di calore a tubi di fumo, i quali consistono in uno scambiatore a fascio tubiero in cui i fumi prodotti da un processo di combustione stazionaria fluiscono all'interno dei tubi, mentre il fluido secondario, comunemente acqua, si trova nel mantello.

Nel primo Capitolo di questa Tesi viene presentata un'analisi sperimentale e teorica del funzionamento di un generatore di calore a tre giri di fumo, operante sia in condizioni stazionarie che in condizioni dinamiche. In letteratura è estremamente difficile trovare dati teorici e sperimentali riguardanti questi sistemi.

Le prove sperimentali sono state svolte variando le condizioni di lavoro del generatore di calore e lavorando sia con che senza generatori di turbolenza all'interno dei tubi che compongono l'ultimo passaggio del sistema.

Un modello dinamico, sviluppato in ambiente MatLab/Simulink, del generatore di calore a tre giri di fumo è quindi presentato in questo elaborato. Il modello è caratterizzato da una struttura a sotto-sistemi che lo rende facilmente adattabile a geometrie diverse, e può essere utilizzato per predire il comportamento dei generatori di calore durante funzionamento in regime stazionario o dinamico.

Le prestazioni dei generatori di calore possono essere incrementate aumentando lo scambio di calore tra i gas combusti e l'acqua attraverso l'inserimento di generatori di turbolenza nei tubi dove fluiscono i gas. In particolare, l'inserimento di turbulatori all'interno dell'ultimo passaggio dei fumi nei generatori di calore comporta un aumento dell'efficienza globale del sistema, per via della minore temperatura di uscita dei gas che si riflette in minori perdite al camino. Per questo motivo, l'aumento dello scambio termico monofase attraverso l'inserimento di generatori di turbolenza è un importante ambito di ricerca per l'industria dei generatori di calore.

Ad ogni modo, è importante considerare che ad un aumento del coefficiente di scambio termico è associato un incremento delle perdite di carico per attrito nel sistema, ed entrambi gli elementi devono essere considerati nel valutare le prestazioni dei turbulatori.

Nella seconda parte di questo elaborato, dopo aver presentato una review delle soluzioni più comuni in letteratura, sono analizzate attraverso simulazioni CFD le prestazioni delle geometrie attualmente utilizzate nei generatori di calore. Gli effetti di diversi parametri geometrici, come la posizione del turbulatore all'interno del tubo e il diametro dello stesso, sono stati analizzati.

Inoltre, attraverso le simulazioni si sono ricavate delle equazioni predittive del comportamento degli inserti. In ultimo è proposta una modifica alle geometrie attuali al fine di proporre una soluzione più performante.

L'efficienza dei generatori di calore può essere incrementata attraverso la condensazione del vapore presente nei gas combusti allo scarico. Per questo, lo studio dello scambio termico bifase è di interesse per l'industria dei generatori di calore.

Durante il processo di condensazione del vapore la maggior parte della resistenza termica è localizzata nel condensato che si forma a contatto con la superficie fredda. Riducendo, o eventualmente annullando, lo spessore del film di liquido alla parete si possono quindi ottenere coefficienti di scambio termico bifase estremamente elevati, incrementando quindi il flusso termico specifico scambiato. Questo permetterebbe di incrementare le performance del sistema a parità di geometria, o di ridurre l'area di scambio (e quindi i costi del generatore di calore) a parità di effetto utile.

Per questo, nell'ultimo Paragrafo di questa Tesi si analizza l'incremento del coefficiente di scambio termico durante condensazione di vapore su superfici nano-ingegnerizzate. L'effetto delle proprietà di bagnabilità delle superfici sulla modalità e sulle prestazioni del processo di condensazione è studiato analizzando il comportamento di superfici convenzionali, superidrofiliiche, idrofobiche e superidrofobiche durante condensazione di vapore puro fluente a diverse velocità. Lo scopo della ricerca è di rimuovere (condensazione a gocce) o ridurre (condensazione a film con scivolamento del condensato) il film di liquido che si forma durante il processo bifase, agendo sulle proprietà della superficie, e di valutare l'effetto della rugosità superficiale sul coefficiente di scambio durante condensazione a film.

LIST OF FIGURES

Figure 1.1.1. Scheme of a three-pass fire tube heat generator. The position of the measuring sensors is also reported.	24
Figure 1.2.1.1. Picture of the heat generator used for the experimental analysis.	26
Figure 1.2.1.2. Photo of the heat generator (opened) used for the experimental analysis.	27
Figure 1.2.1.3. Picture of the shielded thermocouples located at the inlet of the third pass.	27
Figure 1.2.1.4. Sketch of the geometry simulated during CFD analysis of the second inversion.	28
Figure 1.2.1.5. Simulated temperature profiles at the entrance of the tubes for the three test cases. Green circles show the tube most representative of the mean conditions, while red and blue circles show the tubes with the maximum and minimum flue gases inlet temperatures, respectively.	29
Figure 1.2.2.1. Photo of the wave turbulators.	30
Figure 1.2.2.2. Experimental measurements during the 70 kW start-up test without turbulators (solid lines) and the 87 kW start-up test with turbulators (dashed lines): a) fuel mass flux, b) water mass flux and c) inlet water temperature.	31
Figure 1.2.4.1. Experimental data during: a) 70 kW start-up test without turbulators; b) 87 kW start-up test with turbulators. The graph reports the outlet water temperature and the flue gases temperatures at the exit of the first inversion, at the entrance and the exit of the tubes and at the entrance of the chimney.	33
Figure 1.2.4.2. Heat generator efficiency as a function of thermal power at the burner for the test with (TURB) and without (NO TURB) turbulators inside the last pass.	34
Figure 1.3.6.1. Flow chart of the model.	41
Figure 1.3.6.2. Simulink blocks composing the dynamic model.	42
Figure 1.3.7.1. Geometry of the CFD simulations referring to the turbulators used in the experimental campaign.	43

Figure 1.3.7.2. Ratio of the Nusselt number in the tube with turbulator to the one in the plain tube as a function of the Reynolds number of the flue gases. 44

Figure 1.4.1. Experimental versus calculated heat flow rates lost at the chimney without turbulators (blue diamonds) and with turbulators (yellow circles) inside the tubes. 46

Figure 1.4.2. Experimental versus calculated heat flow rates transferred to the water without turbulators (blue diamonds) and with turbulators (yellow circles) inside the tubes. 46

Figure 1.4.3. Experimental (solid lines) and calculated (dashed lines) trends during the start-up of the heat generator when running without turbulators inside the last flue gases pass at $q_{\text{burner}} = 70$ kW: outlet water temperature and flue gases temperatures at the exit of the first inversion, at the entrance and exit of the pipes and at the entrance of the chimney. 47

Figure 1.4.4. Experimental (solid lines) and calculated (dashed lines) trends during the start-up of the heat generator when running with turbulators inside the last flue gases pass at $q_{\text{burner}} = 87$ kW: outlet water temperature and flue gases temperatures at the exit of the first inversion, at the entrance and exit of the pipes and at the entrance of the chimney. 48

Figure 1.5.1. Useful heat flow rate and system efficiency as a function of Reynolds number of the flue gases inside the pipes, for two different tube geometries. 49

Figure 1.5.2. Experimental versus calculated heat flow rates lost at the chimney (a) and acquired by the water (b) for the 600 kW heat generator working with and without turbulators inside the pipes. 50

Figure 1.5.3. Heat flow rate lost at the chimney (a) and acquired by the water (b) as a function of the number of tubes in the last flue gases pass with and without turbulators for the 600 kW heat generator. 51

Figure 1.5.4. Heat generator thermal efficiency as a function of the number of tubes in the last flue gases pass with and without turbulators for the 600 kW heat generator. 52

Figure 1.5.5. Ratio of the friction factor in the tube with turbulator to the one in the plain tube as a function of the Reynolds number of the flue gases. 53

Figure 1.5.6. Flue gases pressure losses in the first inversion and in the pipes as a function of the number of tubes in the last pass, for the 600 kW heat generator. Trends for the cases with and without turbulators are reported. 54

Figure 2.2.1. Enhancement efficiency as a function of the fluid Reynolds number. a) comparison between single and dual full length twisted tape turbulators at different twist ratio y/w . b)

comparison between regularly spaced dual twisted tape turbulators ($y/w = 3$) at different space ratio s/d . Data from Eiamsa-ard et al. (2010a). 62

Figure 2.2.2. Thermal performance ratio of the serrated and smooth twisted tape turbulators as a function of the fluid Reynolds number, for different twist ratio y . Data from Chang et al. (2007a). 63

Figure 2.2.3. Thermal performance ratio of the broken twisted tape turbulators as a function of the fluid Reynolds number, for different twist ratio y , compared against values for the serrated and smooth geometries. Data from Chang et al. (2007b). 64

Figure 2.2.4. Thermal performance ratio of different types of twisted tape turbulators having twist ratio equal to 3. Data from Rahimi et al. (2009). 65

Figure 2.3.1. Sketch of the single-wave turbulator. Flue gases flow direction is showed by the arrow. 66

Figure 2.3.2. Sketch of the double-wave turbulator. Flue gases flow direction is showed by the arrow. 66

Figure 2.3.3. Geometry of simulation n.1. 67

Figure 2.4.1. Mesh of the tube fitted with the single-wave turbulator. 68

Figure 2.4.2. Comparison between experimental and CFD simulated Nusselt number (left) and friction factor (right) for different turbulence models. Graphs from Eiamsa-ard et al. (2009). 69

Figure 2.6.1. Results of CFD simulation n.1 in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). 71

Figure 2.6.2. Results of CFD simulation n.2 in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). 74

Figure 2.6.3. Comparison between the results of simulations n.1 ($z = 85$ mm) and n.2 ($z = 25$ mm) in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). 75

Figure 2.6.4. Results of CFD simulation n.3 in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). 77

Figure 2.6.5. Comparison between the results of simulations n.1 ($D = 36.4$ mm) and n.3 ($D = 42.5$ mm) in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal

performance ratio (c). In both cases turbulator is placed 85 mm inside the tube entrance.

78

Figure 2.6.6. Results of CFD simulation n.4 in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c).

80

Figure 2.6.7. Comparison between the results of simulations n.3 (single wave turbulator) and n.4 (double wave turbulator) in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). In both cases turbulator is placed 85 mm inside the tube (42.5 mm diameter) entrance.

81

Figure 2.6.8. Results of CFD simulation n.5 in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c).

83

Figure 2.6.9. Comparison between the results of simulations n.4 ($z = 85$ mm) and n.5 ($z = 25$ mm) in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). Simulations refer to the double wave turbulator inside the 42.5 mm tube.

84

Figure 2.7.1.1. Frontal view of the actual double wave turbulator. Zones not affected by the turbulator presence are highlighted.

85

Figure 2.7.1.2. Sketch of the modified double wave turbulator.

86

Figure 2.7.1.3. Frontal view of the modified double wave turbulator.

86

Figure 2.7.2.1. Modified double wave geometry used for periodical ANSYS/FLUENT simulations. Flue gases flow direction (in countercurrent) and velocity profile connection are shown by the red and yellow arrow, respectively.

87

Figure 2.7.3.1. Comparison between the results of the periodic simulations performed with the standard and the modified double wave insert in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c).

89

Figure 2.7.3.2. Modified double wave geometry used for periodical ANSYS/FLUENT simulations with opposite flue gases flow direction (co-current flow direction). Flue gases flow direction and velocity profile connection are shown by the red and yellow arrow, respectively.

89

Figure 2.7.3.3. Comparison between the results of the periodic simulations performed with the modified double wave insert in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). Results refer to co-current and countercurrent flow directions.

90

Figure 3.2.1.1. Liquid drop over a solid surfaces. θ represents the contact angle, while γ are the interfacial surfaces energies.	98
Figure 3.2.1.2. Schematic drawings of the actual and projected area of a rough surface.	100
Figure 3.2.1.3. Water drop on a rough substrate.	100
Figure 3.2.1.4. Solid–liquid–air interface (a) homogeneous (Wenzel), (b) composite (Cassie), and (c) with filled holes (Cassie with water penetration).	102
Figure 3.2.2.1. A water drop on a tilted solid surface. The contact angle at the front is the advancing contact angle, θ_{adv} , whereas that at the back is the receding contact angle, θ_{rec} .	103
Figure 3.2.3.1. Water drop over a superhydrophobic surface.	104
Figure 3.2.3.2. (Left) SEM image of the lotus leaf surface showing papillae. (Right) Water droplet on a lotus leaf.	105
Figure 3.4.1. Cylindrical sample used for DWC analysis, sketch from Torresin et al. (2013).	112
Figure 3.4.2. Condensation outside tube, photo from Vemuri et al. (2006).	113
Figure 3.5.1.1. Schematic of the experimental thermosyphon flow loop for condensation tests. P = Pressure transducer, T = Thermocouple, dT = Thermopile, CFM = Coriolis mass Flow Meter, MFM = Magnetic mass Flow Meter, MF = Mechanical Filter.	114
Figure 3.5.1.2. Photo of the whole apparatus. Main components are reported.	118
Figure 3.5.2.1. Photos of the metallic specimen from two opposite sides.	119
Figure 3.5.2.2. Scheme of the position of the metallic sample inside the test section. Red and blue arrows refer to vapor and cooling water flow directions, respectively.	119
Figure 3.5.2.3. Sketch of the whole test section.	120
Figure 3.5.2.4. Photo of the components of the test section before (left) and after (right) assembling.	121
Figure 3.5.2.5. Simulated geometry.	121
Figure 3.5.2.6. Results of one of the CFD simulations. Temperature profile along the specimen (left) and water velocity profile inside the cooling system channel (right).	122
Figure 3.5.3.1. Temperature profiles at the inlet and outlet of the metallic specimen for a case test ($T_{sat} = 106.6^{\circ}\text{C}$).	123

Figure 3.5.4.1. Photo of the temperature calibration system (Superthermometer + Thermistors).

125

Figure 3.5.6.1. Thermal balance of the experimental apparatus. Comparison between the heat given to the system (at the boiling chamber) and the one extracted from it (at the test section and the post condenser).

127

Figure 3.5.6.2. Thermal balance in the test section. Comparison between the heat measured at the cooling side and the one evaluated by looking at the temperatures profiles inside the specimen.

128

Figure 3.6.1.1. Heat flux (a) and heat transfer coefficient (b) versus mean logarithmic temperature difference between the saturated vapor and the surface during condensation over the conventional sample. Data refer to different steam mass velocities G [$\text{kg m}^{-2} \text{s}^{-1}$].

129

Figure 3.6.1.2. Heat flux (a) and heat transfer coefficient (b) data at different steam specific mass flow rate, maintaining $T_{IN-coolant} = 15^\circ\text{C}$ and $m_{coolant} = 320 \text{ kg h}^{-1}$, during condensation over the untreated sample.

130

Figure 3.6.2.1. Comparison between experimental and calculated heat transfer coefficient.

132

Figure 3.6.2.2. Experimental and calculated heat transfer coefficient as a function of the steam specific mass velocity (a) and comparison between experimental and calculated data (b). Tests at $T_{IN-coolant} = 15^\circ\text{C}$ and $m_{coolant} = 320 \text{ kg h}^{-1}$.

133

Figure 3.7.5.1. Example of contact angle characterization. Firstly, a frame of the moving drop is extracted from the video (a). Then the circle fitting the droplet profile near the contact point is created (b). Subsequently, the angle between the circle radius and the contact line is measured via software (c). Finally, the contact angle is geometrically evaluated (d).

136

Figure 3.7.5.2. Examples of advancing and receding contact angles for the four specimens.

137

Figure 3.7.5.3. SEM images of the as-received sample, at different magnifications (1000x, 10000x, 20000x and 80000x).

138

Figure 3.7.5.4. SEM images of the roughened sample, at different magnifications (1000x, 10000x, 20000x and 80000x).

139

Figure 3.7.5.5. SEM images of the etched sample before the re-oxidation step, at different magnifications (1000x, 10000x, 20000x and 80000x).

140

Figure 3.7.5.6. AFM images of the as-received (left) and the roughened (right) specimens. 141

Figure 3.7.5.7. Profile lines for the as-received (left) and the roughened (right) specimens. 141

Figure 3.8.1.1. Heat flux (a) and heat transfer coefficient (b) versus mean logarithmic temperature difference between the saturated vapor and the surface during condensation over the superhydrophilic sample. Data refer to different steam mass velocities G [$\text{kg m}^{-2} \text{s}^{-1}$]. 142

Figure 3.8.1.2. Ratio between the heat transfer coefficient on the superhydrophilic sample (HTC_{SH}) and the one on the untreated sample (HTC_{UN}) at $G_{steam} = 1.5 \text{ kg m}^{-2} \text{s}^{-1}$ and at $G_{steam} = 6.5 \text{ kg m}^{-2} \text{s}^{-1}$. 143

Figure 3.8.2.1. Temperature profiles inside the superhydrophobic specimen during the transition between dropwise and filmwise condensation mode. Saturation temperature is about 106°C , steam flows at about $6.5 \text{ kg m}^{-2} \text{s}^{-1}$. 144

Figure 3.8.2.2. Superficial temperature difference between top and bottom as a function of the removed heat flux for the filmwise condensation tests over the untreated sample. 145

Figure 3.8.2.3. Mean logarithmic temperature difference between the steam and the wall (a) and heat transfer coefficient (b) as a function of the acquisition time during the transition between dropwise and filmwise condensation mode. Saturation temperature is about 106°C , steam flows at about $6.5 \text{ kg m}^{-2} \text{s}^{-1}$. 145

Figure 3.8.2.4. Droplets cycle on a superhydrophobic aluminum sample at about 10 K of surface subcooling and about $6.5 \text{ kg m}^{-2} \text{s}^{-1}$ of vapor mass velocity. The arrow indicates the vapor flow direction. 145

Figure 3.8.3.1. Heat flux (a) and heat transfer coefficient (b) versus mean logarithmic temperature difference between the saturated vapor and the surface during condensation over the hydrophobic sample. Data refer to different steam mass velocities G [$\text{kg m}^{-2} \text{s}^{-1}$]. 148

Figure 3.8.3.2. Comparison between the heat transfer coefficient on the hydrophobic (HY) sample and the one on the untreated (UN) specimen (a), and ratio between the two (b), as a function of the mean logarithmic temperature difference between the steam and the wall. Data refer to three steam mass velocities G ($1.5 \text{ kg m}^{-2} \text{s}^{-1}$, $3.1 \text{ kg m}^{-2} \text{s}^{-1}$ and $5.1 \text{ kg m}^{-2} \text{s}^{-1}$). 149

Figure 3.8.3.3. Variation of the local Nusselt number with the longitudinal distance for different values of the slip length β . Graph from Pati et al. (2013). 151

Figure 3.8.4.1. Slip length β and velocity profile over a surface.

152

Figure 3.8.4.2. Comparison between the experimental data and the values calculated imposing nonzero slip velocity at the solid-liquid interface. Data refer to different steam mass velocities G .

155

LIST OF TABLES

Table 1.2.2.1. Working conditions during steady state test runs.	30
Table 1.2.4.1. Measured temperatures of water and gases during steady state test runs.	32
Table 1.4.1. Simulated values of gas and water temperatures in the fire tube heat generator operating at steady-state conditions.	45
Table 1.5.1. Working conditions for the 600 kW heat generator.	50
Table 1.5.2. Simulated values of gas and water exit temperatures in the 600 kW fire tube heat generator operating at steady-state conditions.	50
Table 2.3.1. Simulated geometries.	67
Table 2.4.1. Boundary conditions of the CFD simulations.	68
Table 3.5.1.1. Pressure losses in the vapor line.	117
Table 3.5.5.1. Type B experimental uncertainty of the measured parameters.	125

INTRODUCTION

Heat generators are actually the most used system for producing domestic heat and hot water. These systems are apparatuses that realize the purpose of heating a fluid (most commonly water) without achieving its boiling point, through the heat given by the exhaust products of a stationary combustion. Usually these systems consist in shell-and-tube heat exchangers, characterized either by the secondary fluid flowing inside the tube bank, while the flue gases flow in the shell, or by the opposite configuration, with the exhaust products inside the tubes and the secondary fluid in the external side. According to these two flow configurations the systems are called water tube heat generators or fire tube (i.e. smoke tube) heat generators, respectively. Moreover, fire tube heat generators can be classified according to the number of passes of the flue gases inside the system. Given their large diffusion, in this Thesis attention will be paid on fire tube systems.

Heat generators inefficiencies are mainly related to heat losses at the chimney, due both to flue gases air index and exit temperature. A heat generator which works with low efficiency implies a bigger fuel consumption for achieving the same performances, meaning higher costs and environmental impacts. Growth of fuel prices as well as a greater attention on environmental problems (combined with more restrictive laws), have thus pushed the manufactures to focus on enhancing heat generators efficiency and reliability. Therefore, it is important to perform both theoretical and experimental studies on such systems, with the aim of developing computational models for optimization and prediction of heat generators performances when varying working conditions or geometries, as well as of arranging experimental database that can be used for analysis and comparison purposes. After that, strategies for enhancing heat transfer performances have to be analyzed. These should concern both single-phase and two-phase heat transfer. In fact, in modern heat generators latent heat recovery through condensation of steam in the flue gases at the exit of the system is used for enhancing heat transfer to the water, thus system efficiency. Because of this, enhanced performances can be achieved both increasing single-phase convective heat transfer coefficient of the flue gases flowing inside the tube bank as well as by augmenting two-phase heat transfer coefficient during vapor condensation process.

To the best of the author's knowledge, literature data and models about fire tube heat generators are extremely rare. Thus, the primary aim of the present work is to experimentally investigate heat transfer inside such systems, analyzing the working conditions of a three pass fire tube heat generator, both during stationary and transient regimes. The new experimental database is then used for validation of a MatLab/Simulink dynamic model developed for prediction of heat

generators behavior, as shown later on. After validation the model was used for predicting performances of fire tube heat generators working at various conditions and with various internal geometries, since its blocks structure makes it easily adaptable to different configurations. Results of this analysis will be presented at the end of Chapter 1 of the present Thesis.

As previously introduced, heat generators efficiency can be enhanced by reducing flue gases temperature at the chimney, this way reducing associated thermal losses. This can be achieved by enhancing flue gases convective heat transfer coefficient inside the tube bank. The method used so far for reaching this goal is to insert in the pipes elements, named turbulators, which enhance flue gases turbulence, this way enhancing convective heat transfer coefficient. Because of this, single phase heat transfer enhancement by means of turbulence generator inserts is an important field of research for the heat transfer industry.

The presence of such elements allows to reach significantly lower exhaust products exit temperature, as shown in this Thesis, but it also increases frictional pressure losses along the flue gases line. Thus, turbulators efficiency should be studied by analyzing both thermal and frictional augmentation.

Different turbulators geometries can be found in the literature but, for practical industrial application, manufacture simplicity plays a crucial role. Because of this, heat generator manufactures currently focus on geometries that can be realized through pressing methods, such as the “wave” turbulators presented later on.

In the second part of this work, CFD simulations will be carried out for analysis of the performances of two turbulators shapes actually used in heat generator industry. Effects of several geometrical parameters, such as turbulator position inside the tube and pipe diameter, will be analyzed. Discussion about optimal working conditions, criticalities and benefits of the two solutions will be undertaken. Moreover, an alternative solution will be proposed, simulated and compared against the previous, with the aim of increase thermal and frictional performances.

As discussed above, by recovering the latent heat in the exhaust products coming out from the heat generator it is possible to achieve higher system efficiencies. Thus, two-phase heat transfer is a field of interest in the heat generators industry.

When steam condensation occurs the mayor resistance to the thermal transport is associated to the condensate layer forming near the solid wall. Reducing, or eventually eliminating, the liquid film thickness at the wall will thus significantly augment the condensation heat transfer coefficient, increasing the heat transfer per unit area. Thus, it will be possible to achieve higher system performances maintaining the same geometry, or eventually to reduce the heat transfer area (thus the system cost) maintaining the same net effect. Because of this, the last part of the

present Thesis is focused on the analysis of condensation heat transfer coefficient enhancement by means of nano-engineered surfaces.

Owing to their water repellency properties, hydrophobic and superhydrophobic surfaces have recently been studied as promising solution to several challenges, including enhancement of two-phase heat transfer performances.

Surface wettability is defined by the contact angles of a water drop sitting over it. For a static drop equilibrium contact angle is taken into account while for moving drops the advancing and receding contact angles are taken as a reference. The difference among the last two gives the contact angle hysteresis. Surfaces presenting contact angles lower than 90° are called hydrophilic, while those having contact angle bigger than 90° are called hydrophobic. Moreover, if the surface presents extremely low contact angles ($< 30^\circ$) it's named superhydrophilic, while if it presents extremely high contact angles, i.e. greater than 150° , and low contact angle hysteresis, i.e. lower than 10° , it is named superhydrophobic.

Hydrophilic and superhydrophilic surfaces are produced by properly modifying superficial morphology, in order to obtain a micro-/ nano- scale superficial roughness, while hydrophobic surfaces are produced by lowering surface free energy. Proper surface roughness can be obtained through different techniques, as micromachining, micro-contact-printing, chemical etching in aqueous solutions and deep radiative ion etching. Low surface energy can be obtained by coating the substrate with a thin layer of a material having small surface energy, such as organic substances, polymers, and noble metals. By combining adequate surface roughness and low surface free energy it is finally possible to obtain superhydrophobic properties over the substrate, allowing water drops to sit over the surface with a quasi-spherical shape and to easily roll-off from it.

In this Thesis, techniques for modifying wetting properties of aluminum substrates are presented, together with surface wetting properties characterization. Theoretical and experimental analysis of superhydrophobicity over aluminum substrates has been undertaken during a six months research period at the Laboratory of Thermodynamics in Emerging Technologies of ETH Zurich. Then, effect of surface wetting characteristics on the condensation mode and performance is analyzed by testing the behavior of conventional, superhydrophilic, hydrophobic and superhydrophobic surfaces during pure steam flow condensation at different mass velocities, with the aim of removing (dropwise condensation mode) or eventually reducing (slip-driven filmwise condensation mode) the condensate layer over the wall during the two-phase process, as well as to evaluate the effect of surface roughness on the filmwise condensation heat transfer coefficient.

1 EXPERIMENTAL AND THEORETICAL ANALYSIS OF HEAT TRANSFER INSIDE HEAT GENERATORS

In the first Chapter of this Thesis an experimental analysis and a modeling study of a three-pass fire tube heat generator, fed with natural gas, is presented. Very limited research is published in the literature on such systems. In the present work a heat generator was tested while operating both in stationary and in dynamic regime. Tests were performed varying fuel flow rate, air index, water inlet temperature and flow rate, with and without turbulators inside the last pass.

The experimental data were used to validate a dynamic model of fire tube heat generators. The present model was developed with reference to a three-pass smoke tube heat generator, dividing it into subsystems, which are representative of its main components. The subsystems structure of the model makes it easily adaptable to heat generators having different geometrical characteristics and it can be used to predict the behavior of heat generators working in stationary and transient conditions.

1.1 Introduction

A heat generator is an apparatus that realizes the purpose of heating a fluid (i.e. water), without achieving its boiling point, through the heat given by the products of a stationary combustion. A heat generator which causes the vaporization of the secondary fluid is called boiler. Heat generators usually have a shell-and-tube geometry, and can be divided in two macro categories: water tube heat generators and fire tube (or smoke tube) heat generators. The first group is characterized by the secondary fluid flowing inside the tube bank, while the combustion products flow outside in the shell. The second group displays a different configuration, with gas inside tubes and water in the external side. Moreover, fire tube heat generators can be classified according to the number of passes of the flue gases inside the system.

Figure 1.1.1 shows the classical scheme of a three-pass fire tube heat generator, which is the one of interest in this work. The Figure reports also the position of the measuring sensors for the present experimental analysis, which will be presented in the next Paragraph.

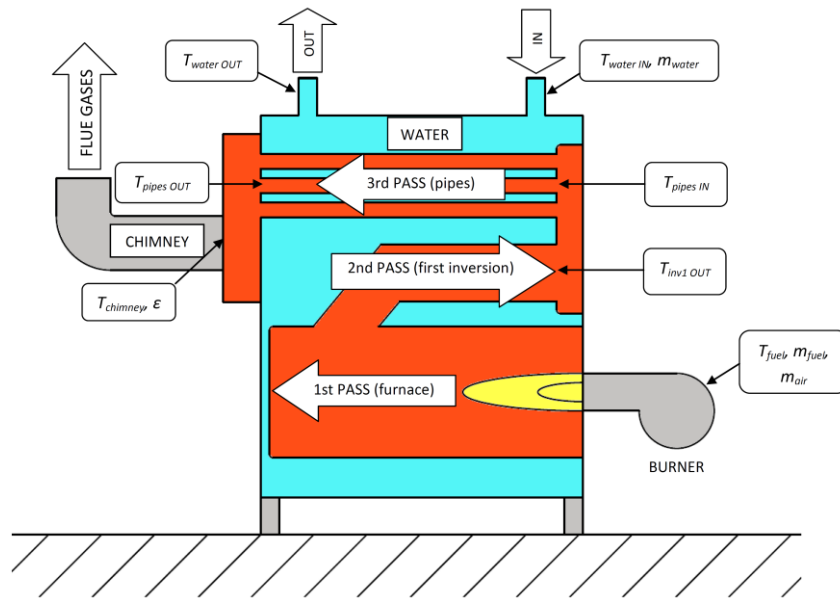


Figure 1.1.1. Scheme of a three-pass fire tube heat generator. The position of the measuring sensors is also reported.

The major causes of inefficiency in a heat generator are the losses at the chimney, related both to the flue gas exit temperature and the air index. A heat generator working with low efficiency implies a bigger consumption of fuel to obtain the same performance, which means higher costs and environmental impacts. The growth of the fuels prices and a greater attention to the environmental problems, combined with more restrictive laws, have pushed the manufacturers to focus on increasing the efficiency and the reliability of the heat generator systems. Therefore it is important to study experimentally and theoretically heat generators operating at different working configurations, in order to optimize them and to increase their efficiency.

Literature data about fire tube heat generators or boilers are rare. Furthermore, in the literature there are several models for simulating water tube boilers (Astrom *et al.*, 2000; Kim *et al.*, 2005; Emara-Shabaik *et al.*, 2009), while comprehensive models for fire tube systems are difficult to find. There are some fire tube boilers models developed using CFD codes (Gómez *et al.*, 2008; Pezo *et al.*, 2006; Bordbar and Hyppanem, 2007), but, as stated in Gutiérrez Ortiz (2011), these may be highly dependent on the initial and boundary conditions as well as on grid nodes and turbulence model. Gutiérrez Ortiz (2011) developed a dynamic model for the analysis of fire tube boilers performance, and implemented it using MatLab. The model is based on mass, energy and momentum conservation, and was verified against results from literature. Sørensen (2004) and Sørensen *et al.* (2003) modeled and simulated fire tube boilers, covering both the flue gas and the water side of the system. The dynamic model was formulated as a number of submodels merged

together using MatLab/Simulink. The model was experimentally verified against a full scale boiler plant.

Thus, the first objective of the present Thesis is to investigate experimentally the heat transfer inside fire tube heat generators, both during stationary and transient working regimes. The experimental data will then be used to develop and validate a computational model of fire tube heat generators, which can be useful to predict their working conditions, both in stationary and transient regimes.

Differently from the models by Gutiérrez Ortiz (2011) and Sørensen *et al.* (2003) the present model was developed and validated for prediction of single phase heat generator performance, thus it can be applied directly for analysis of systems where no phase change of the heat transfer fluid (normally water vaporization) takes place. Although such heat generators are very common, they are almost not treated in the open literature.

1.2 Experimental investigation

In this Section experimental data acquired on a three-pass fire tube heat generator, installed in Reillo's Laboratory in Piombino Dese (PD), are presented.

As a beginning, experimental apparatus and test conditions are introduced. Then, description of the uncertainty analysis is reported for the measured values. Finally, experimental data acquired using the present setup are shown.

Tests presented hereinafter were performed both introducing and not introducing turbulators inside the setup. These are elements used to enhance convective heat transfer coefficient of the flue gases flowing inside the tubes of the heat generators. A detailed analysis about the enhancement of the single-phase convective heat transfer coefficient will be presented in Chapter 2 of the present Thesis.

1.2.1 Description of the experimental apparatus

The heat generator used for the present experimental study is shown in Figure 1.2.1.1.



Figure 1.2.1.1. Picture of the heat generator used for the experimental analysis.

It is a 90 kW (nominal power) three-pass fire tube heat generator, with a scheme similar to the one proposed in Figure 1.1.1, fed with natural gas (treated here as pure methane). The secondary fluid is water.

The furnace has an internal diameter equal to 334 mm and a length equal to 836 mm. The first inversion, corresponding to the second pass of the flue gases inside the generator, has an internal diameter equal to 146.4 mm and a length equal to 652 mm. The third pass is characterized by twenty-two parallel tubes, each one having an internal diameter equal to 36.4 mm and a length equal to 826 mm. Between the first inversion and the tubes there is the second inversion, which is a cylindrical element with an internal diameter equal to 444 mm. At the exit of the tubes there is a cylindrical collecting element, which leads the flue gases to the chimney. The volume of the water inside the tank is equal to 182.4 liters.

Turbulators can be placed inside the tubes to enhance flue gases convective heat transfer coefficients.

A photo of the interior of the generator, showing the combustion chamber, the outlet of the second pass and the entrance of the tubes, is reported in Figure 1.2.1.2.



Figure 1.2.1.2. Photo of the heat generator (opened) used for the experimental analysis.

Experimental tests were performed in order to acquire flue gases temperatures inside the system and to evaluate the heat flow rate transferred by the combustion products to the water.

Flue gases temperatures were acquired at the exit of the second pass, at the entrance and at the exit of the tubes and at the entrance of the chimney. These were measured by means of K-type thermocouples. Shields were fixed on the thermocouples to prevent irradiation exchange between the temperature sensors and the surrounding walls. A photo of the shielded thermocouples used to acquire flue gases temperatures at the entrance of the third pass is presented in Figure 1.2.1.3.



Figure 1.2.1.3. Picture of the shielded thermocouples located at the inlet of the third pass.

Each shield consists in a polished stainless steel duct, fixed around the temperature sensor through a spring system. It has a 15 millimeters square cross section shape and a length equal to 80 mm. The temperature sensors are located in the middle of the shields, which have five holes (each having a diameter equal to 3 mm) on each side, in order to assist the passage of the flue gases inside the duct.

Through preliminary CFD simulations of the second inversion the authors evaluated which one of the twenty-two pipes was the most representative of the average conditions of the flue gases flowing in the third passage, choosing this specific one as the reference for the acquisition of flue gases temperature at the entrance and at the exit of the tubes. The flue gases temperature was also measured in the tubes that, from the simulations, resulted to have the lower and the higher flue gases inlet temperature.

A sketch of the simulated geometry for the second inversion is reported in the following figure.

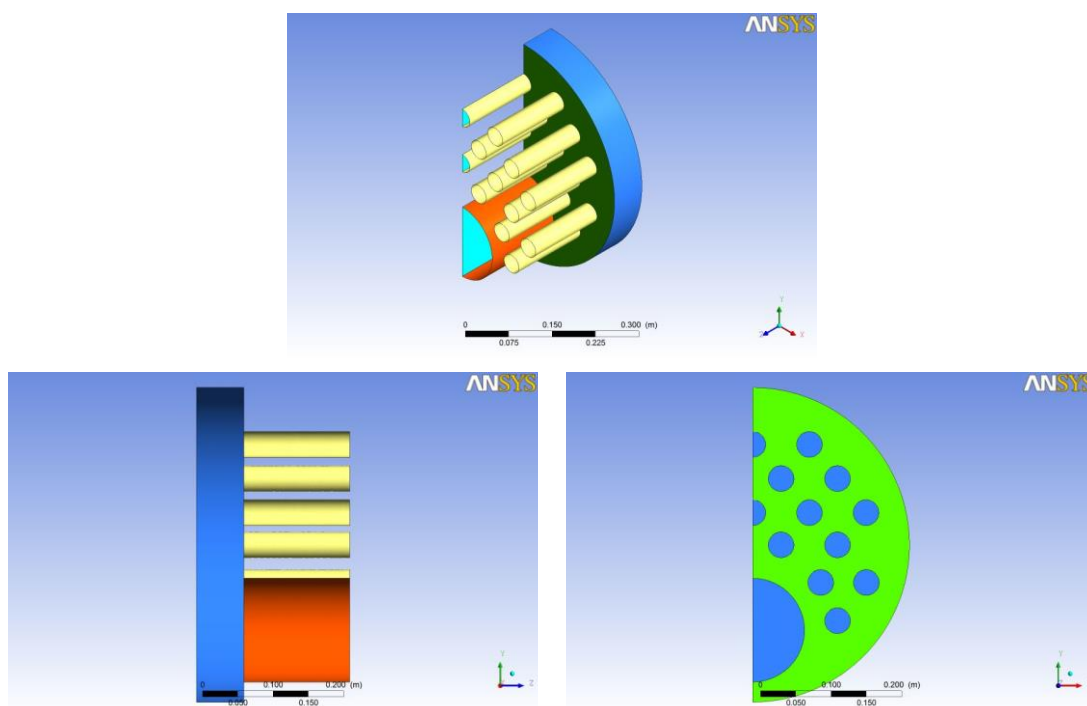


Figure 1.2.1.4. Sketch of the geometry simulated during CFD analysis of the second inversion.

Simulations were performed imposing the temperature and the convective heat transfer coefficient of the water inside the tank, neglecting heat losses through the insulation, and fixing flue gases inlet temperature and mass flow rate. Three different fuel mass fluxes were tested, representative of the heat generator working conditions.

Figure 1.2.1.5 shows the results of the CFD analysis in terms of simulated temperature profile of the flue gases at the entrance of the tubes, for the three simulations. The tube which resulted to be the most representative of the medium conditions is pointed out, as well as those which resulted to have the lower and the higher flue gases inlet temperatures.

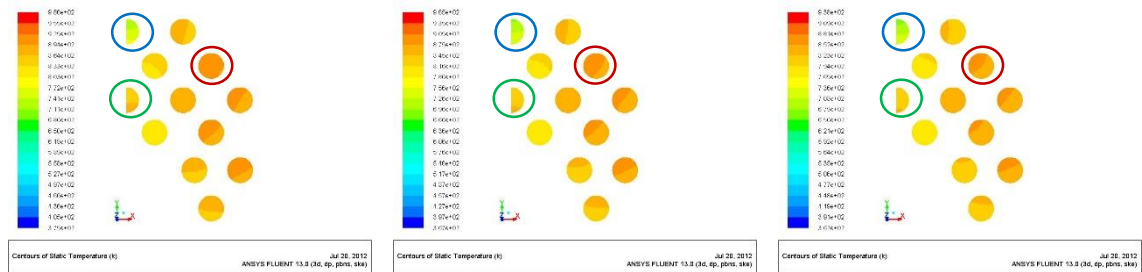


Figure 1.2.1.5. Simulated temperature profiles at the entrance of the tubes for the three test cases. Green circles show the tube most representative of the mean conditions, while red and blue circles show the tubes with the maximum and minimum flue gases inlet temperatures, respectively.

The heat flow rate absorbed by the water in the shell was evaluated by measuring the water inlet and outlet temperatures, by means of platinum resistance temperature detectors (Pt 100), and the water mass flow rate, by means of an electromagnetic flow meter.

Moreover, the following parameters were measured: fuel and air temperatures using Pt 100 and K-type thermocouple respectively; fuel flow rate using a volumetric flow meter and CO₂ volumetric percentage in dry flue gases using IR-active continuous gas analyzer. The location of the measuring sensors is reported in the heat generator sketch of Figure 1.1.1.

1.2.2 Experimental tests

A first group of tests was performed to evaluate the performance of the heat generator when working without turbulators inside the tubes.

The system was analyzed trying to cover a wide range of operating conditions, varying the heat generator thermal power at the burner from 70 kW to 87 kW. Data relative to the 70 kW test were acquired from the start-up of the system until stationary conditions were reached. Data relative to the other tests were acquired only in stationary regime. The main characteristics of the four tests are reported in Table 1.2.2.1, referring to the stationary conditions. The air index ε is evaluated through the CO₂ volumetric percentage in dry flue gases as

$$\varepsilon = 100(|CO_2|_{st} / |CO_2| - 1) \quad (1.1)$$

with $|CO_2|_{st} = 11.7\%$ stoichiometric value for methane.

	<i>WITHOUT TURBULATORS</i>				<i>WITH TURBULATORS</i>			
q_{burner} [kW]	<i>70</i>	<i>77</i>	<i>81</i>	<i>87</i>	<i>70</i>	<i>77</i>	<i>81</i>	<i>87</i>
m_{fuel} [kg h ⁻¹]	4.73	5.27	5.55	5.93	4.88	5.30	5.58	5.91
m_{water} [kg s ⁻¹]	0.86	0.89	0.94	0.95	0.91	0.92	0.97	0.97
ε [%]	21.6	19.9	19.3	20	20.7	20.7	19.6	19.5
T_{fuel} [°C]	19.4	19.6	19.9	20.1	21.7	22.7	21.1	22.1
T_{env} [°C]	23	23.4	23.6	23.6	25.3	26.7	25.2	26.5
$T_{water,IN}$ [°C]	60.6	59.4	59.5	59.3	58.7	60.2	58.6	62.1

Table 1.2.2.1. Working conditions during steady state test runs.

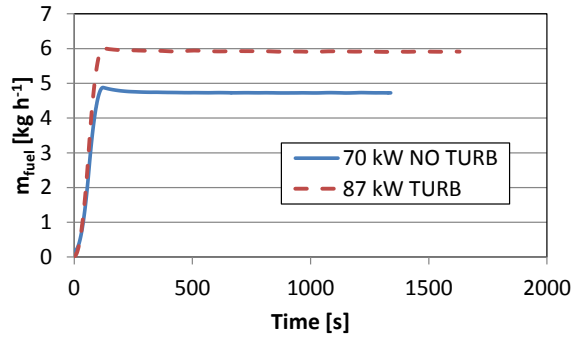
A second group of tests were performed inserting twenty-two wave turbulators inside the heat generator (one in each tube). Tests were performed decreasing the thermal power at the burner from 87 kW to 70 kW. Data relative to the 87 kW test were acquired since the start-up of the system, while the others were acquired only in stationary conditions. Table 1.2.2.1 reports also the main characteristics of this group of tests (in stationary regime). Figure 1.2.2.1 shows a photo of the wave turbulators. A specific analysis of enhancement of convective heat transfer coefficient through turbulators will be presented in Sec. 2 of the present work.



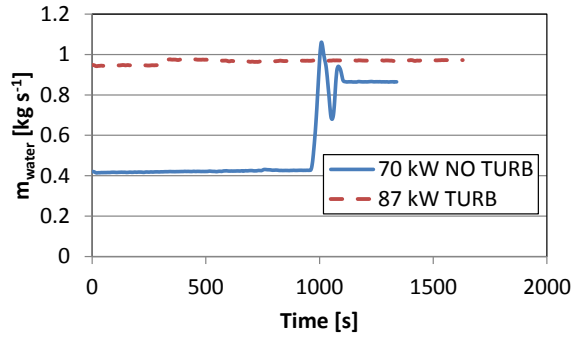
Figure 1.2.2.1. Photo of the wave turbulators.

Figure 1.2.2.2 shows the experimental values of fuel mass flux, water mass flux and inlet water temperature for two start-up tests. Stationary conditions are achieved for the two runs after about 1200 s.

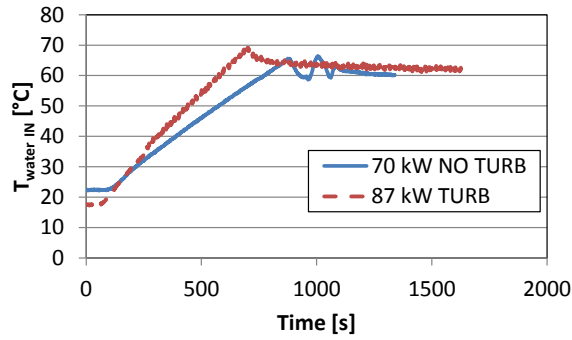
The present tests were performed at different thermal power at the burner (from 70 kW to 90 kW), maintaining $\varepsilon \approx 20\%$ and $T_{WATER,IN} \approx 60^\circ\text{C}$ (in stationary conditions).



a)



b)



c)

Figure 1.2.2.2. Experimental measurements during the 70 kW start-up test without turbulators (solid lines) and the 87 kW start-up test with turbulators (dashed lines): a) fuel mass flux, b) water mass flux and c) inlet water temperature.

1.2.3 Uncertainty analysis

The combined standard uncertainty of the measured parameter $y = f(x_1, x_2, \dots, x_N)$ is evaluated as

$$u_c(y) = \sqrt{\sum_{i=1}^N \left(\frac{df}{dx_i} \right)^2 [u_A^2(x_i) + u_B^2(x_i)]} \quad (1.2)$$

where $u_A(x_i)$ and $u_B(x_i)$ are respectively the type A and type B uncertainties of the i -th parameter x . The expanded uncertainty $u_M(y)$ is then obtained by multiplying $u_c(y)$ by a coverage factor $k = 2$. In this analysis the uncertainties of the thermodynamic properties of the flue gas and the water are neglected.

The fuel flow meter has a B-type uncertainty equal to ± 0.4 % of the reading, while the uncertainty of the water flow meter is equal to ± 1 %. All the temperature detectors are calibrated within the ± 0.4 K error band. Finally, the IR gas analyzer has a type B uncertainty equal to ± 0.5 % of the measured value.

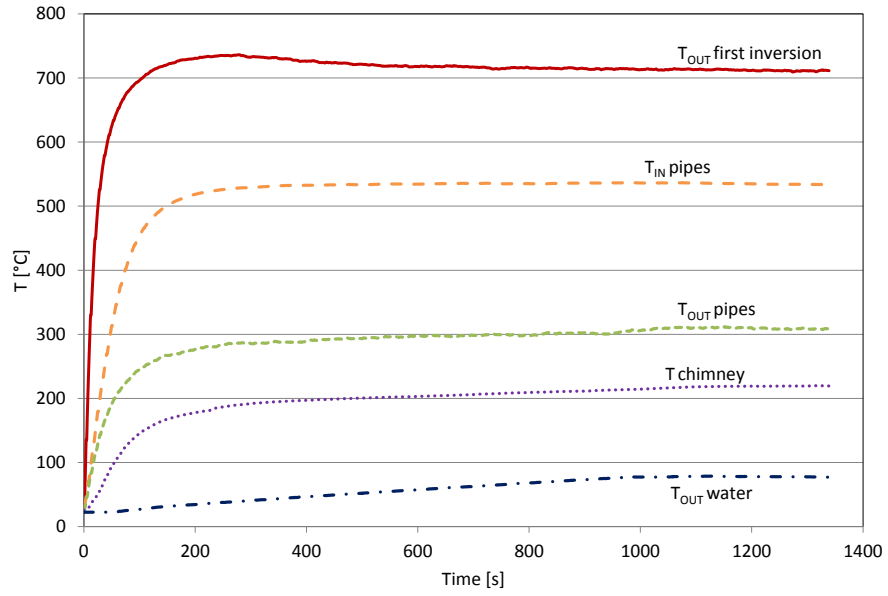
Maximum values of the expanded uncertainty of the measured thermal power at the burner, of the measured heat flow rate lost at the chimney and of the measured heat flow rate acquired by the water resulted to be 1%, 2.5% and 9% respectively.

1.2.4 Experimental results

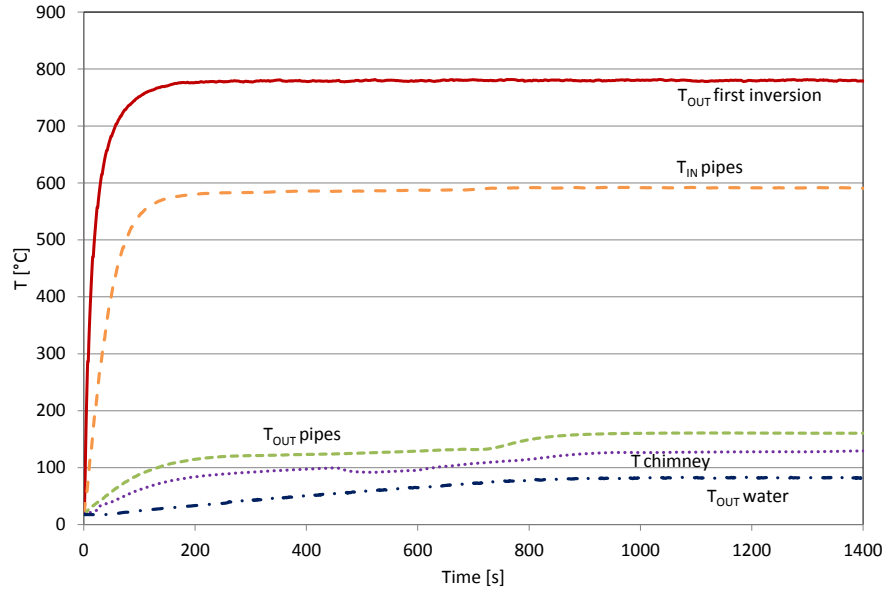
Experimental results of the stationary tests are reported in Table 1.2.4.1, while Figure 1.2.4.1 shows the trends of the flue gases temperatures and of the outlet water temperature during the two dynamic tests. Stationary conditions refer to tests during which stable output conditions are reached and maintained for more than 300 s.

	<i>WITHOUT TURBULATORS</i>				<i>WITH TURBULATORS</i>			
q_{burner} [kW]	70	77	81	87	70	77	81	87
$T_{inv1\ OUT}$ [°C]	712	733	743	755	737	754	766	779
$T_{pipes\ IN}$ [°C]	535	554	561	571	550	568	579	591
$T_{pipes\ OUT}$ [°C]	309	333	343	355	141	149	153	160
$T_{chimney}$ [°C]	219	233	241	250	111	119	120	130
$T_{water\ OUT}$ [°C]	77.7	76.9	77.3	77.8	76.2	78.6	77.5	82.3

Table 1.2.4.1. Measured temperatures of water and gases during steady state test runs.



a)



b)

Figure 1.2.4.1. Experimental data during: a) 70 kW start-up test without turbulators; b) 87 kW start-up test with turbulators. The graph reports the outlet water temperature and the flue gases temperatures at the exit of the first inversion, at the entrance and the exit of the tubes and at the entrance of the chimney.

The thermal balance of the system, as reported in Eq. (1.3), was positively checked for the present data within $\pm 1.5\%$.

$$\begin{aligned}
 m_{fuel} [H_U + c_{p,fuel} (T_{fuel} - T_{ref})] + m_{air} c_{p,air} (T_{air} - T_{ref}) = \\
 = m_{water} c_{p,water} (T_{water,OUT} - T_{water,IN}) + m_{fg} c_{p,fg} (T_{chimney} - T_{ref})
 \end{aligned}
 \tag{1.3}$$

By looking at results in Table 1.2.4.1 and in Figure 1.2.4.1 it is possible to see the significant effect of the turbulators on the flue gases exit temperature. While the temperatures of the flue gases at the exit of the second pass and at the entrance of the last one are similar among tests having the same thermal power at the burner, flue gases temperatures at the exit of the tubes, and similarly at the chimney, are 170 K –200 K higher when working without turbulators as compared to the case with the turbulators inserted in the system.

Higher exit temperature at the chimney means higher thermal losses, given by the last term in the right hand side of Eq. (1.3). Higher thermal losses at the chimney lead to lower heat generator efficiencies, given by the ratio of the heat flow rate acquired by the water (first term in the right hand side of Eq. (1.3)) to the total heat flow rate entering the system (left hand side of Eq. (1.3)). Tests with turbulators present global efficiencies 3% to 10% higher (the larger increase the bigger the thermal power at the burner) than those without elements inside the tubes.

Figure 1.2.4.2 shows heat generator efficiency as a function of the thermal power at the burner for the tests with and without turbulators inside the tubes.

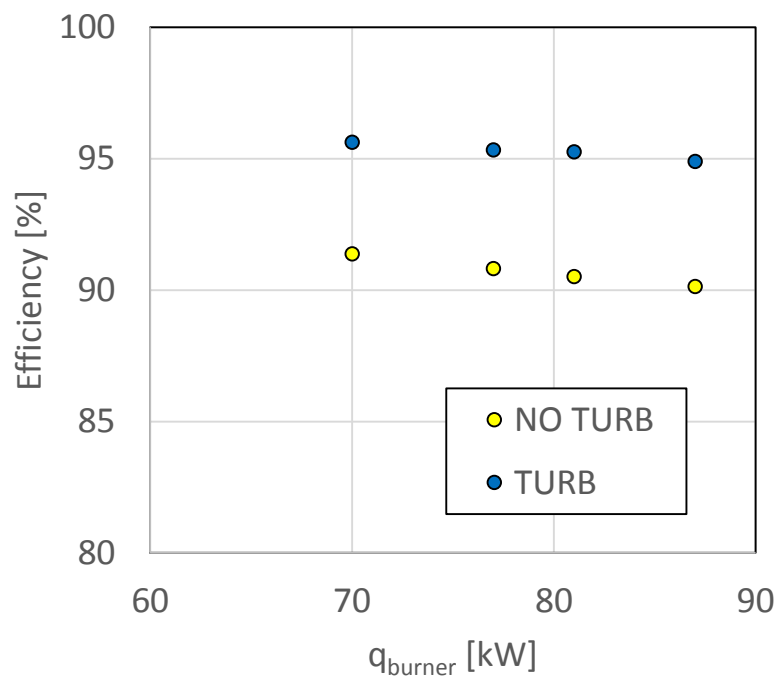


Figure 1.2.4.2. Heat generator efficiency as a function of thermal power at the burner for the test with (TURB) and without (NO TURB) turbulators inside the last pass.

1.3 Dynamic model of the heat generator

In this Section a model for predicting both stationary and dynamic working conditions of the three-pass fire tube heat generator previously studied is presented. The model is implemented using Matlab/Simulink tools.

The present dynamic model is essentially based on the analysis proposed by Sørensen (2004) and by Sørensen *et al.* (2003) for fire tube boilers. Each portion of the heat generator was analyzed through energy and mass balances, in order to obtain a number of Differential Algebraic Equations (DAEs), solved by means of MatLab/Simulink.

The DAEs are solved using ode15s Simulink solver. This is a variable step solver that computes the model's state at the next time step using variable-order numerical differentiation formulas. This solver decreases the simulation step size to increase accuracy when the system's continuous states are changing rapidly, while it increases the step size to save simulation time when the system's states are changing slowly. Ode15s is a multi-step solver, so generally it needs the solutions at several preceding time points to compute the current one.

The model is composed of several blocks referring to the different components of the heat generator, this way it is easily modifiable and adaptable to different geometrical configurations.

1.3.1 General description

The heat generator was divided into three main subsystems: the first one relative to the flue gas side, the second one relative to the walls and the third one relative to the water side. Moreover, the flue gas side subsystem was divided into a number of blocks equal to the number of components of the heat generator.

Thus, referring to the three-pass heat generator shown in Figure 1.1.1, the flue gas subsystem will be characterized by five blocks, corresponding to the furnace, the first and the second inversion, the pipes and the exit collector. A similar division is used also for the walls subsystem. Furthermore, the second and the third pass of the flue gases were discretized into ten elements. Referring to the global energy balance of the system, the heat produced by the combustion of the fuel is partially transferred to the water, partially lost with the flue gases at the chimney and partially lost through the generator case to the environment. In the present analysis the losses through the generator case are neglected. The thermal balance of the simulated system was checked (as shown in Sec. 1.2.4) to verify that this approximation is absolutely acceptable. Low discrepancies ($< 1.5\%$) between input and output heat flow rates computed as in Eq. (1.3) show that losses through the generator case can be neglected without affecting the results.

The flue gases produced by the combustion of the fuel in the burner move from the combustion chamber to the chimney, exchanging heat with the walls by radiation and convection, thus

progressively reducing their temperature. A portion of the heat received by the walls passes through them and is transferred to the water by convection, while the remaining stands in the walls increasing their temperature. Finally the heat received from the walls increases the outlet water temperature.

1.3.2 Model variables

The dynamic model requires as input data the mass flow rate and inlet temperatures of the fuel, air and water, the system geometry and the walls characteristics. The output of the model are the flue gas conditions at the chimney (mass flow rate and temperature), the outlet water temperature, the flue gases temperatures inside the heat generator and the thermal fluxes exchanged between the combustion products and the water.

The model needs a series of algebraic loops to be solved, so it is necessary to define some initial conditions. Initial flue gases temperatures and walls temperatures are defined by the model itself, respectively equal to the environmental and water temperatures. On the contrary, the initial temperature of the water inside the tank needs to be set by the user.

In the following Sections the equation used for solving the flue gases, walls and water subsystems are discussed.

1.3.3 Flue gases subsystem

Flue gases thermodynamic properties and heat transfer coefficients are evaluated as proposed by Annaratone (2008).

Each flue gas block is characterized by an entering heat flow rate, an outgoing heat flow rate and a heat flow rate transferred between the exhaust products and the walls. For the furnace block the entering heat flow rate is the one related to the combustion of the fuel (given by the left-hand side of Eq. (1.3)), the outgoing heat flow rate is the one related to the flue gases coming out from the combustion chamber and the transferred heat flow rate is the one related to the radiative heat exchanged between the flame and the walls and to the radiative and convective heat exchanged between the flue gases and the walls.

In the following blocks, the entering heat flow rate is the one associated to the flue gases at the inlet, the outgoing is associated to the flue gases at the outlet and the transferred one is the heat flow rate exchanged by radiation and convection between the flue gases and the walls.

The mass balance of the exhaust products blocks is

$$m_{fg,OUT} = m_{fg,IN} \quad (1.4)$$

with $m_{fg,IN} = m_{fuel} + m_{air}$ for the furnace block.

The energy balance of each flue gases block is

$$dU_{fg}/d\tau = q_{IN} - q_{OUT} - q_{fg \rightarrow walls} \quad (1.5)$$

where q_{IN} and q_{OUT} are respectively the heat fluxes entering and exiting the system, $q_{fg \rightarrow wall}$ is the heat flux exchanged with the walls and U_{fg} is the internal energy of the combustion products:

$$U_{fg} = V_{fg} \rho_{fg} c_{p,fg} (\bar{T}_{fg} - T_{ref}) \quad (1.6)$$

In Eq. (1.6), T_{ref} is the reference temperature equal to 273.15 K, V_{fg} is the block internal volume, ρ_{fg} and $c_{p,fg}$ are the density and the isobaric specific heat of the flue gases at $\bar{T}_{fg}$, which is the mean temperature of the exhaust products inside the system:

$$\bar{T}_{fg} = 0.5 \cdot (T_{fg,IN} + T_{fg,OUT}) \quad (1.7)$$

where $T_{fg,IN}$ and $T_{fg,OUT}$ are the temperatures of the flue gases at the inlet and the outlet of the block, respectively.

The heat flow rate coming out from each combustion products block, which is equal to the one entering the following element, is evaluated as

$$q_{fg,OUT} = m_{fg,OUT} c_{p,fg,OUT} (T_{fg,OUT} - T_{ref}) \quad (1.8)$$

with the isobaric specific heat referred to the mean temperature between outlet and reference temperatures.

The heat flux acquired by the walls in the furnace is evaluated as the sum of the radiative heat exchanged between the flame and the walls and the radiative and convective heat exchanged between the flue gases and the walls

$$q_{fg \rightarrow walls} = q_{R,flame} + q_{R,fg} + q_{C,fg} \quad (1.9)$$

The three terms in Eq. (1.9) are evaluated as follows, considering the furnace as a cylindrical system

$$q_{R,flame} = K \alpha_{R,flame} A_{flame} (T_{flame} - \bar{T}_{wall}) \quad (1.10)$$

$$q_{R,fg} = K \alpha_{R,fg} A_{fur} (\bar{T}_{fur} - \bar{T}_{wall}) \quad (1.11)$$

$$q_{C,fg} = K \alpha_{C,fg} A_{fur} (\bar{T}_{fur} - \bar{T}_{wall}) \quad (1.12)$$

with $\bar{T}_{fur}$ mean temperature of the flue gases inside the furnace and A_{fur} internal area of the combustion chamber. The corrective factor

$$K = \left\{ 1 + \frac{\ln(D_m / D_{int})}{2\pi\lambda L} [\alpha_{R, flame} A_{flame} + (\alpha_{R, fg} + \alpha_{C, fg}) A_{fur}] \right\}^{-1} \quad (1.13)$$

is needed because the mean temperature of the walls $\bar{T}_{wall}$ is defined at the mean walls diameter

$D_m = 0.5 \cdot (D_{int} + D_{ext})$. The flame temperature and the heat transfer coefficients used in Eqs. (1.10) – (1.13) are evaluated as proposed in Annaratone (2008). Flame temperature is calculated as

$$T_{flame} = 1048.15 + 0.18 \frac{q_{IN, fur}}{m_{fg}} + (0.048 \frac{q_{IN, fur}}{m_{fg}} - 20) \frac{q_{IN, fur}}{1000 S_{fur}} + \\ - (0.0017 \frac{q_{IN, fur}}{m_{fg}} + 0.2) \left(\frac{q_{IN, fur}}{1000 S_{fur}} \right)^2 \quad (1.14)$$

where $q_{IN, fur}$ is the heat flow rate entering the furnace with the fuel and the air in [kW], S_{fur} is the cross section area of the furnace and T_{flame} is the flame temperature. The following equations are solved to calculate the heat transfer coefficients in Eqs. (1.10) – (1.13):

$$\alpha_{R, flame} = \frac{\sigma \varepsilon [T_{flame}^4 - \bar{T}_{wall}^4]}{[T_{flame} - \bar{T}_{wall}]} \quad (1.15)$$

$$\alpha_{R, fg} = \frac{\bar{q}_{R, CO_2} + \bar{q}_{R, H_2O}}{[\bar{T}_{fur} - \bar{T}_{wall}]} \quad (1.16)$$

$$\alpha_{C, fg} = \begin{cases} 0.023 Re^{0.8} Pr^{0.4} \frac{\lambda_{fg}}{D} k_{TURB} ; \text{ if } Re > 2100 \\ 1.86 (Re Pr \frac{D}{L})^{1/3} (\mu_{fg} / \mu_{walls})^{0.14} \frac{\lambda_{fg}}{D} k_{TURB} ; \text{ if } Re \leq 2100 \end{cases} \quad (1.17)$$

where σ is the Stefan-Boltzmann constant, ε is the flame emissivity, $\bar{q}_{R, CO_2}$ and $\bar{q}_{R, H_2O}$ are the specific heat fluxes transferred by radiation from the carbon dioxide and the steam to the walls (evaluated as in Annaratone, 2008), Re and Pr are Reynolds and Prandtl flue gas numbers, μ_{fg} and λ_{fg} are the dynamic viscosity and the thermal conductivity of the combustion products at the proper temperature and μ_{walls} is the exhaust products dynamic viscosity at T_{walls} . In Eq. (1.16), $\alpha_{R, fg}$ is evaluated considering only the heat radiated by the carbon dioxide and the steam, neglecting

the other components of the flue gases. However, Trinks *et al.* (2004) confirm that the heat emitted by the diatomic gases (such as O₂, N₂ and H₂) is negligible in comparison to the one radiated by the triatomic ones (such as CO₂ and H₂O), so this assumption does not lead to substantial errors. Finally, k_{TURB} is a parameter that accounts for presence of turbulators, as it will be shown later.

The flame area was evaluated as follows:

$$A_{flame} = \pi D_{int}^{1.5} (m_{fg} / m_{fg,ref})^{0.85} \quad (1.18)$$

The flame area is directly dependent on the mass flow rate of the flue gas, thus on the mass flow rates of fuel and air. The reference flue gas mass flow rate is $m_{fg,ref} = 0.0375 \text{ kg s}^{-1}$.

Eqs. (1.9) – (1.13) can be used also for the others flue gases blocks, fixing $A_{flame} = 0$. For the pipes block it must be considered that the flue gases mass flux was equally divided among the twenty-two pipes. The convective heat transfer coefficient calculated by Eq. (1.17) is obtained using a corrective factor k_{TURB} to account for the presence of turbulators inside the tubes. Referring to the second inversion and to the exit collector, convective heat transfer coefficients of flue gases were evaluated through CFD analysis. For these elements the heat given by the exhaust products is transferred both to cylindrical and plane walls, thus the previous equation were properly adapted. The heat dissipation between the flue gases and the environment through the insulation of the second inversion was neglected in this analysis, while the losses through the external walls of the exit collector were considered.

Finally, it is possible to evaluate the variation of the mean temperature of flue gases over time:

$$d\bar{T}_{fg} / d\tau = \frac{q_{IN} - q_{OUT} - q_{fg \rightarrow walls}}{V_{fg} \rho_{fg} c_{p,fg}} \quad (1.19)$$

1.3.4 Walls subsystems

The walls subsystem is divided into a number of blocks equal to the number of components of the heat generator (five), as already described for the flue gas.

On the hot side, in each block the walls receive the heat evaluated with Eq. (1.9); a part of this is transferred to the water, the remaining is absorbed by the walls increasing their internal energy:

$$U_{walls} = V_{walls} \rho_{walls} c_{p,walls} (\bar{T}_{walls} - T_{ref}) \quad (1.20)$$

The energy balance of each walls side block is

$$dU_{walls}/d\tau = q_{fg \rightarrow walls} - q_{walls \rightarrow water} \quad (1.21)$$

Eqs. (1.20) – (1.21) lead to the variation of the mean walls temperature (referring to the mean thickness) over time:

$$d\bar{T}_{walls}/d\tau = \frac{q_{fg \rightarrow walls} - q_{walls \rightarrow water}}{V_{walls} \rho_{walls} c_{p,walls}} \quad (1.22)$$

The heat flow rate acquired by the water inside the tank is calculated, referring to a cylindrical element, as

$$q_{walls \rightarrow water} = \left[\frac{1}{\alpha_{C,water}} + \frac{D_{ext} \ln(D_{ext}/D_m)}{2\lambda} \right]^{-1} A_{walls} \left(\frac{D_{ext}}{D_{int}} \right) \Delta T_{ml} \quad (1.23)$$

The same equation can be used for the heat transfer between the water and the vertical walls, considering the conductive thermal resistance of the vertical plate $s/2\lambda$ (at the mean thickness). By estimating the velocity profile of the water in the shell the present authors verified that the forced convective contribution to the overall heat transfer is negligible as compared to the natural convection one. Thus, the convective heat transfer coefficient of the water is evaluated, referring to Incropera and DeWitt (1996), as follows when referring to a horizontal cylindrical element,

$$\alpha_{C,water} = \left\{ 0.6 + 0.387 Ra^{1/6} \left[1 + \left(\frac{0.559}{Pr} \right)^{9/16} \right]^{-8/27} \right\}^2 \lambda_{water} / D_{ext} \quad (1.24)$$

and when referring to a vertical wall as

$$\alpha_{C,water} = \left\{ 0.825 + 0.387 Ra^{1/6} \left[1 + \left(\frac{0.492}{Pr} \right)^{9/16} \right]^{-8/27} \right\}^2 \lambda_{water} / D_{ext} \quad (1.25)$$

In Eqs. (1.24) – (1.25) Ra and Pr are Rayleigh and Prandtl numbers of the water.

1.3.5 Water subsystem

The water subsystem is characterized by the heat flow rates acquired from the walls, by the heat flow rate entering with the inlet water and by the one outgoing with the outlet water. Thus, the energy balance of the system is

$$dU_{water}/d\tau = q_{water,IN} + \Sigma q_{walls \rightarrow water} - q_{water,OUT} = \Sigma q_{walls \rightarrow water} - q_{water} \quad (1.26)$$

where

$$U_{water} = V_{water} \rho_{water} c_{p,water} (\bar{T}_{water} - T_{ref}) \quad (1.27)$$

$$q_{water} = m_{water} c_{p,water} (T_{water,OUT} - T_{water,IN}) \quad (1.28)$$

$$T_{water,OUT} = \bar{T}_{water} \quad (1.29)$$

Through Eqs. (1.26) – (1.27) it is possible to evaluate the variation of the mean water temperature inside the tank over time:

$$d\bar{T}_{water} / d\tau = \frac{\Sigma q_{walls \rightarrow water} - q_{water}}{V_{water} \rho_{water} c_{p,water}} \quad (1.30)$$

The water thermodynamic properties used in Eqs. (1.24) – (1.30) are evaluated as proposed in IAPWS (1997).

1.3.6 Model flow chart

A sketch of the flow chart of the model is presented in Figure 1.3.6.1.

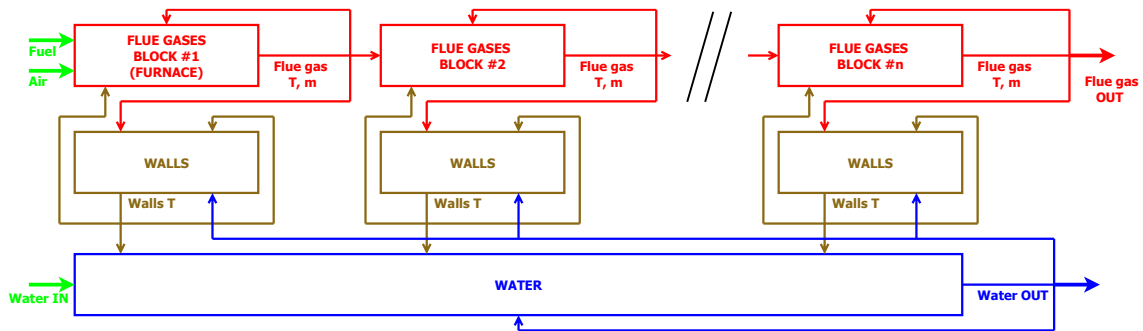


Figure 1.3.6.1. Flow chart of the model.

As it can be seen from the Figure, the model needs a series of algebraic loops to be solved. The inlet conditions of the fuel and the air are given as input to the first block relative to the flue gases which is the one pertaining to the furnace. The next blocks receive as input the conditions of the flue gases coming from the previous hot side block. To evaluate the outlet conditions each block needs also tentative values of the mean temperature of the flue gases and the walls, which are variables of the model themselves. This way, two algebraic loops are created for each flue gases block, one originating from the block itself, the other one from the corresponding walls subsystem. Similarly, each walls subsystem (which receives the flue gases conditions from the corresponding hot side block) leads to two loops, one originating from the block itself (because tentative values of $\bar{T}_{walls}$ are necessary for the solution) and one originating from the water side subsystem (because tentative values of $\bar{T}_{water}$ are also needed).

Finally, the water subsystem needs as input the conditions of the inlet water, the temperature of the walls (given by the solid side subsystems) and the tentative value of the mean water temperature inside the tank, originating one more algebraic loop for the evaluation of the outlet water conditions.

Figure 1.3.6.2 reports a snapshot of the model developed in Matlab/Simulink.

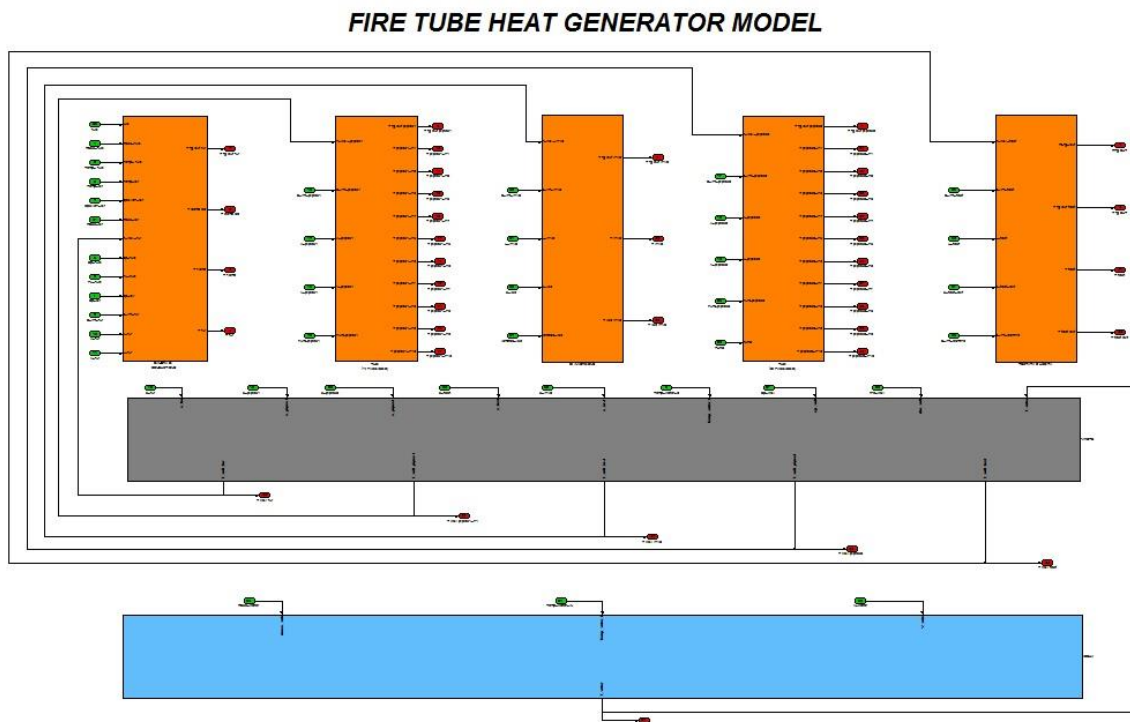


Figure 1.3.6.2. Simulink blocks composing the dynamic model.

1.3.7 Modeling the presence of turbulators

A deep analysis of the enhancement of single-phase convective heat transfer by means of turbulators inside the last-pass tubes will be presented in Chapter 2 of this Thesis.

Herein, equations that have been used to account for the presence of the turbulence generators during experimental validations of the dynamic model are briefly presented.

Inserts used during the experimental campaign consist in wave turbulators having a global length equal to 598.5 mm and a height equal to 20 mm, placed 85 mm apart from the tube entrance.

The effects of turbulators on the flue gases convective heat transfer coefficients were analyzed through CFD simulations. Simulations were performed in a tube fitted with swirl generators inserts, fixing flue gases inlet temperature and walls temperature. The mass flow rate of flue gases was varied in order to cover a range of gas Reynolds numbers between 250 and 50000.

The reference geometry for these CFD simulations is shown in Figure 1.3.7.1.

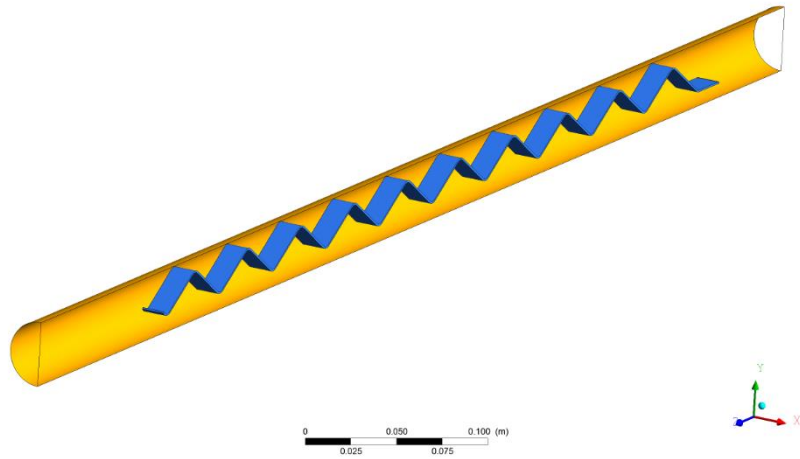


Figure 1.3.7.1. Geometry of the CFD simulations referring to the turbulators used in the experimental campaign.

Eiamsa-ard *et al.* (2009) experimentally and theoretically studied twisted tape swirl generators comparing CFD simulations results obtained with different turbulence models against experimental data. According to their study, the one which gives better predictions is the Shear Stress Transport (SST) $k-\omega$ turbulence model. Thus, the present simulations were performed using the SST $k-\omega$ model, applying low-Re corrections.

From the simulations it is possible to evaluate the convective heat transfer coefficient inside the tube as

$$\alpha = q / (A \cdot \Delta T_{ml}) \quad (1.31)$$

Where q is the transferred heat flow rate, ΔT_{ml} is the mean logarithmic temperature difference between the flue gases and the walls and A is the heat transfer area of the walls. When α is known, it is possible to calculate the Nusselt number of the flue gas for the tube fitted with the turbulator

$$Nu = \alpha D / \lambda \quad (1.32)$$

and compare it against the value obtained with a plain tube (Eq. (1.17) with $k_{TURB} = 1$), named Nu_0 . This comparison is presented in Figure 1.3.7.2 as a function of flue gases Reynolds number.

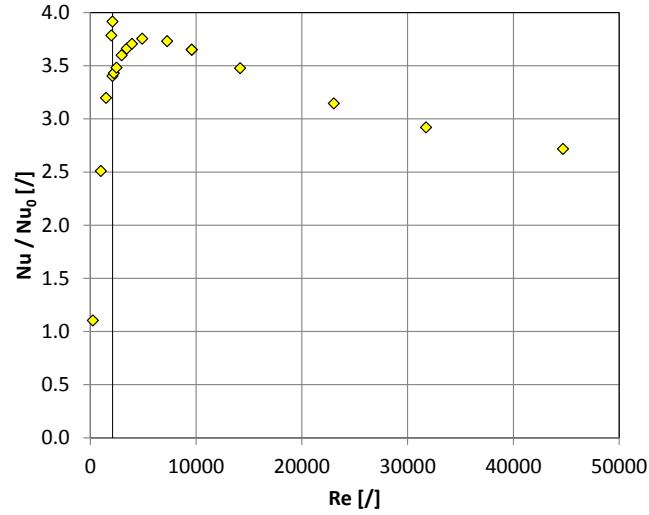


Figure 1.3.7.2. Ratio of the Nusselt number in the tube with turbulator to the one in the plain tube as a function of the Reynolds number of the flue gases.

The Figure shows that the heat transfer enhancement due to the wave turbulators increases with Re when the exhaust products flow in laminar regime, until a maximum Nusselt numbers ratio Nu/Nu_0 equal to 4 when $Re = 2100$. When $Re > 2100$ Nu/Nu_0 increases until it reaches a maximum value around $Re = 7000$, then it decreases.

From the results in Figure 1.3.7.2 a correlation to evaluate the corrective factor

$k_{TURB} = Nu / Nu_0$ of the wave turbulator, as a function of the flue gases Reynolds number, can be obtained.

$$k_{TURB} = \begin{cases} -3.0503 \cdot 10^{-7} Re^2 + 2.2301 \cdot 10^{-3} Re + 0.5694; & \text{if } Re \leq 2100 \\ -3.9833 \cdot 10^{-16} Re^4 + 1.2336 \cdot 10^{-11} Re^3 - 1.4587 \cdot 10^{-7} Re^2 + 7.5078 \cdot 10^{-4} Re + 2.3561; & \text{if } 2100 < Re \leq 10000 \\ 5.0468 \cdot 10^{-10} Re^2 - 5.4213 \cdot 10^{-5} Re + 4.1321; & \text{if } Re > 10000 \end{cases} \quad (1.33)$$

This expression of k_{TURB} is used in Eq. (1.17). When there are no turbulators in the pipes $k_{TURB} = 1$.

1.4 Model validation

The data presented in Section 1.2 were used to validate the present model in stationary and dynamic working conditions. The data reported in Table 1.2.2.1 are given as input to the model for eight stationary simulations, over a simulation time equal to 2000 s. The main output values

of the model, in terms of water and gas temperatures, are listed in Table 1.4.1 and can be directly compared to the experimental corresponding values reported in Table 1.2.4.1.

	WITHOUT TURBULATORS				WITH TURBULATORS			
q_{burner} [kW]	70	77	81	87	70	77	81	87
$T_{inv1\ OUT}$ [°C]	711.2	735.5	748.4	758	738.3	753.8	767.8	791.5
$T_{pipes\ IN}$ [°C]	529.9	550	560.8	570.3	553.9	568.3	579.3	592
$T_{pipes\ OUT}$ [°C]	307.8	325.7	335.3	347.3	150.2	154.9	155.5	161.7
$T_{chimney}$ [°C]	220.2	233.4	240.6	250	117.4	121.8	121.9	127.7
$T_{water\ OUT}$ [°C]	77.5	77.6	77.6	78.4	76.4	79.1	77.4	81.9

Table 1.4.1. Simulated values of gas and water temperatures in the fire tube heat generator operating at steady-state conditions.

These data refer to stationary working conditions. Moreover, the model was verified using the experimental values provided by the generator manufacturer and obtained in the same experimental apparatus. This database refers to stationary total heat flow rate data, which means that only water temperatures and flue gases temperatures at the chimney were measured. The manufacturer database refers to conditions similar to those presented in Sec. 1.2.

Figures 1.4.1 and 1.4.2 present the validation of the model using test runs in stationary conditions, comparing the experimental and the calculated heat flow rates lost at the chimney (Figure 1.4.1) and the experimental and calculated heat flow rates acquired by the water (Figure 1.4.2). These flow rates are evaluated as follows:

$$q_{chimney} = m_{fg} c_{p,fg} (T_{chimney} - T_{env}) \quad (1.34)$$

$$q_{water} = m_{water} c_{p,water} (T_{water,OUT} - T_{water,IN}) \quad (1.35)$$

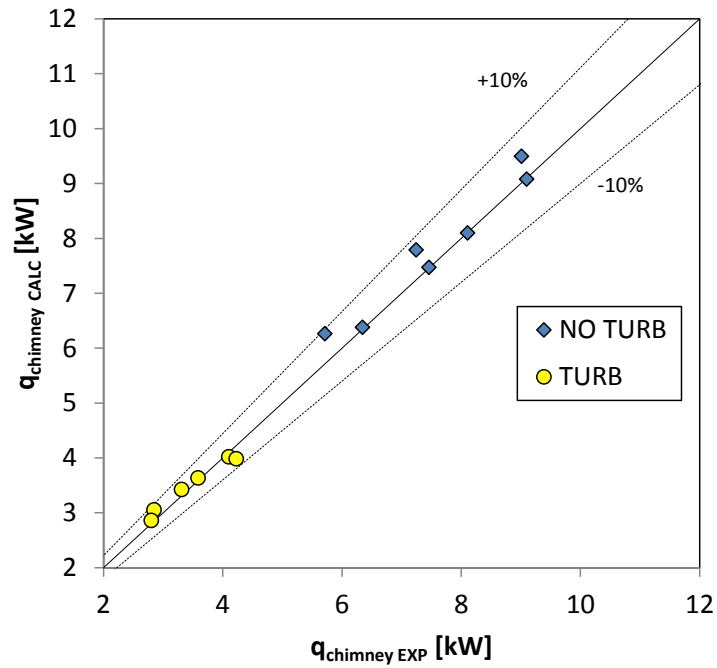


Figure 1.4.1. Experimental versus calculated heat flow rates lost at the chimney without turbulators (blue diamonds) and with turbulators (yellow circles) inside the tubes.

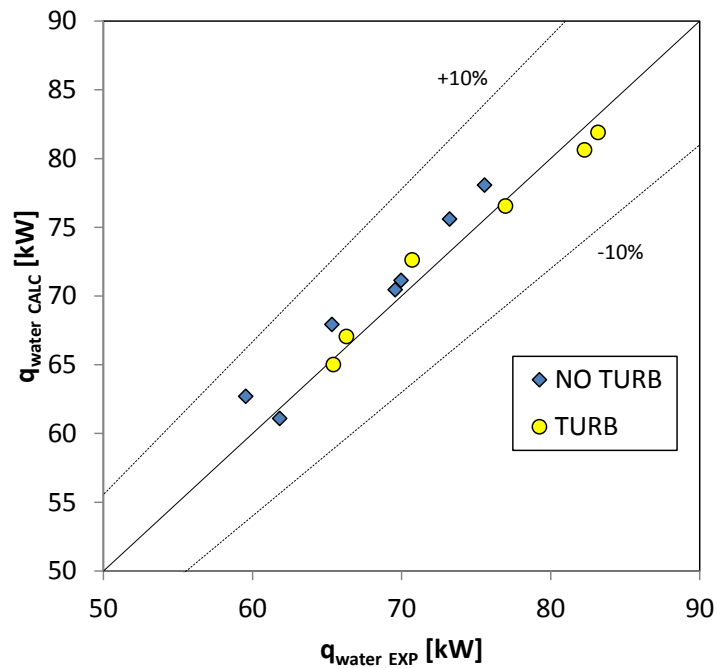


Figure 1.4.2. Experimental versus calculated heat flow rates transferred to the water without turbulators (blue diamonds) and with turbulators (yellow circles) inside the tubes.

As it can be seen from Figures 1.4.1 and 1.4.2, the validation of the model using the data in Table 1.2.2.1 provides a very good result.

The calculated values of flue gas temperature at the chimney and water outlet temperature allow a prediction of the total heat flow rate data within an absolute uncertainty lower than $\pm 10\%$. Figure 1.4.1 also displays the effects of inserting turbulators inside the pipes: heat flow rates lost at the chimney are much lower for tests with turbulators as compared to the case without turbulators, which means higher heat generator efficiency.

Good predictive accuracy was also found in terms of internal temperatures of the flue gases, as it can be seen in the dynamic validation of the model. The dynamic validation was performed simulating the system start-up, giving as model input the values presented in Figure 1.2.2.2. Comparison between calculated data and experimental ones (presented in Figure 1.2.4.1) is reported in Figures 1.4.3 and 1.4.4.

Figure 1.4.3 shows the experimental (solid lines) and calculated (dashed lines) trends of the flue gases temperatures at the exit of the first inversion, at the entrance and exit of the pipes, at the entrance of the chimney and finally the outlet water temperature, during the start-up of the heat generator (without turbulators inside the last flue gases pass). Figure 1.4.4 shows the same trends, referring to the test with turbulators inside the pipes.

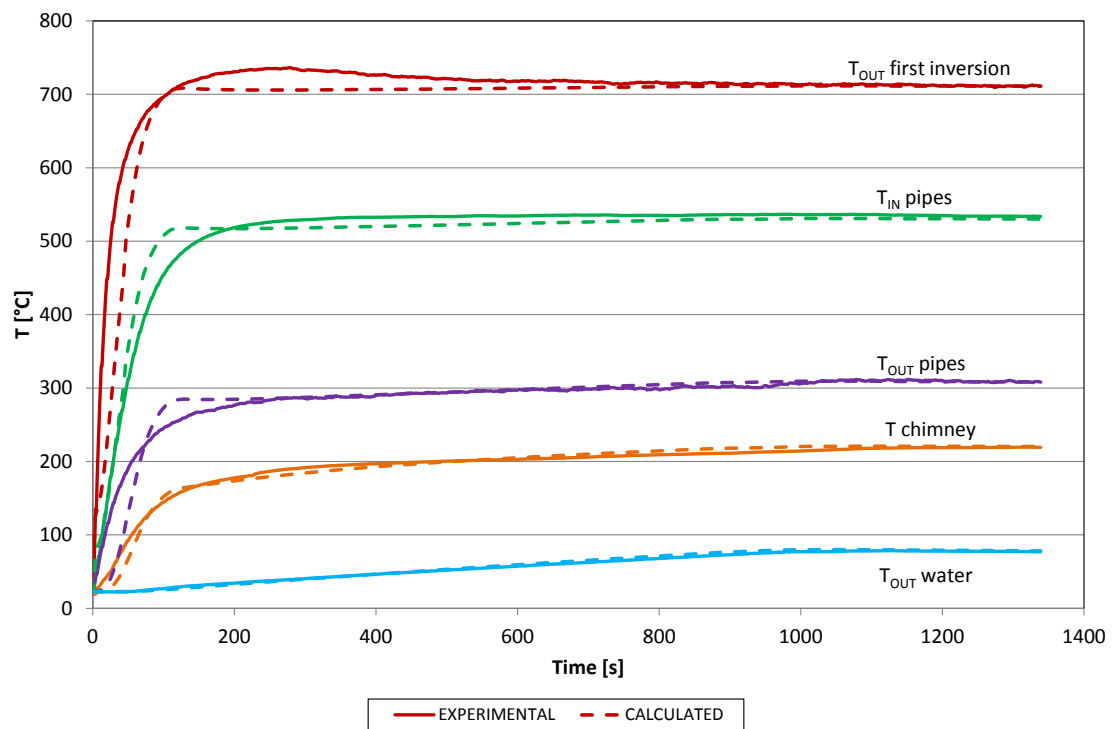


Figure 1.4.3. Experimental (solid lines) and calculated (dashed lines) trends during the start-up of the heat generator when running without turbulators inside the last flue gases pass at $q_{\text{burner}} = 70 \text{ kW}$: outlet water temperature and flue gases temperatures at the exit of the first inversion, at the entrance and exit of the pipes and at the entrance of the chimney.

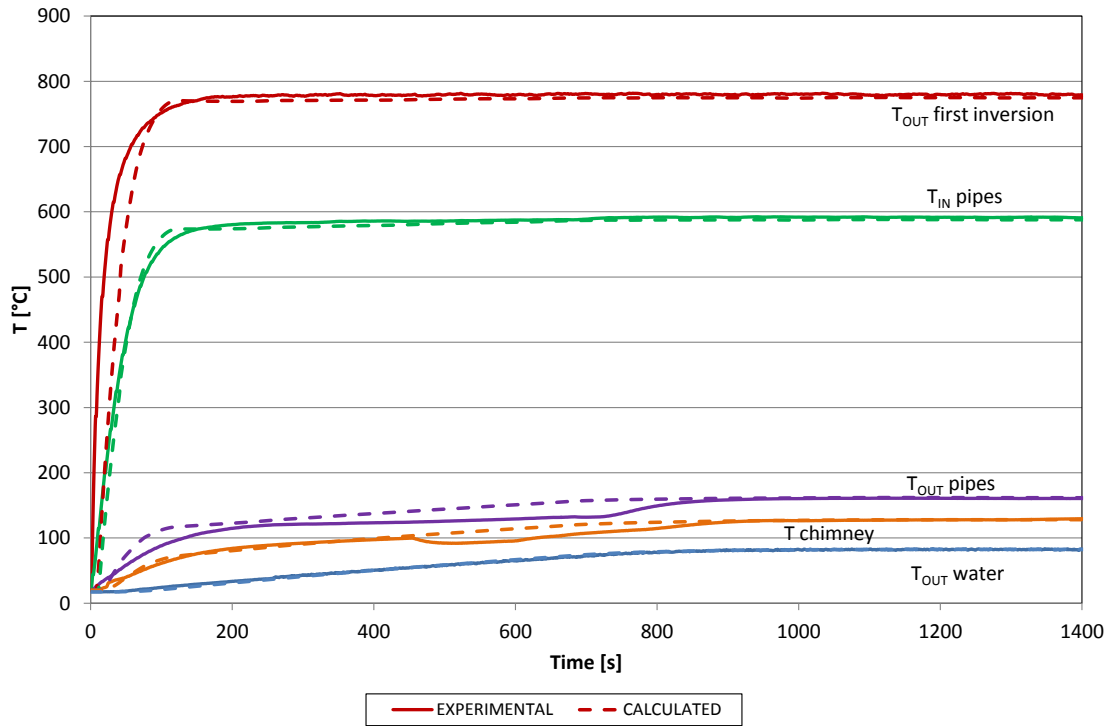


Figure 1.4.4. Experimental (solid lines) and calculated (dashed lines) trends during the start-up of the heat generator when running with turbulators inside the last flue gases pass at $q_{burner} = 87$ kW: outlet water temperature and flue gases temperatures at the exit of the first inversion, at the entrance and exit of the pipes and at the entrance of the chimney.

The validation of the model using experimental data in dynamic conditions gave a very good agreement. Low discrepancies between experimental and calculated trends of the flue gas temperatures during the initial transient regime were found, while the water temperature profile is very well predicted by the model. The differences in the transient conditions could be related to an approximate evaluation of flue gases and wall thermal inertia. However, stationary results (i.e. the values achieved at the end of the simulation time, around 1200 s) are not affected by such discrepancies.

1.5 Model predictions with different geometries

The model presented in this Thesis can be implemented for prediction of working conditions of fire tube heat generators operating with different geometries. As an example, Figure 1.5.1 shows the results obtained by running the model with different thermal power at the burner (thus different flue gases mass flow rate and consequentially different Reynolds number of the flue gases flowing inside the pipes), maintaining constant the remaining input parameters, which are

equal to those presented in Table 1.2.2.1 for the 70 kW test without turbulators. Two different geometries are compared in the figure, the one previously presented and a similar one with a bigger inside diameter (43.7 mm) of the pipes composing the third pass. The different configurations are compared both in terms of useful heat to the water (left axis) and system thermal efficiency (right axis), defined as the ratio between the heat acquired by the water and the thermal power at the burner.

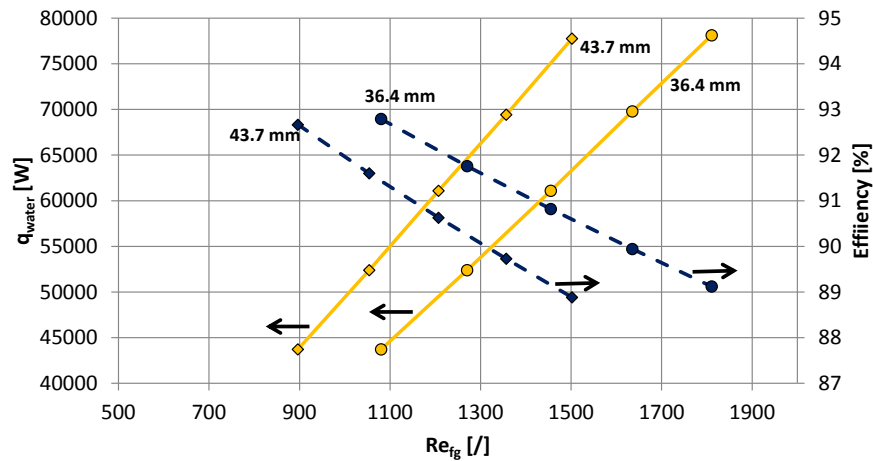


Figure 1.5.1. Useful heat flow rate and system efficiency as a function of Reynolds number of the flue gases inside the pipes, for two different tube geometries.

The figure shows that, for a fixed pipes diameter, the larger the thermal power at the burner (thus the larger the Reynolds number of the flue gases inside the last pass) the larger is the useful heat flow rate, but the lower is the system thermal efficiency. It also shows that, for the same thermal power, the thermal efficiency increases with lower diameter pipes.

Moreover, the model can be used for analysis of heat generators having different scale and geometry. As an example, it has been used for simulating performances of a 600 kW nominal power heat generator.

This has the same three-pass configuration showed in Figure 1.1.1, but with the combustion chamber located in the upper part of the water tank and the pipes located in the lower one. Again, the burner is feed with natural gas (still treated as pure methane) and the secondary fluid is water. The furnace of the large scale system has an internal diameter equal to 594 mm and a length equal to 1798 mm; while dimensions of the first inversion are equal to 248 mm and to 1791 mm, respectively. The third pass is composed by ninety parallel tubes, each one having an internal diameter equal to 39.4 mm and a length equal to 1920 mm. The cylindrical element which forms

the second inversion has an internal diameter equal to 694 mm and a thickness equal to 85 mm.

Finally, the volume of the water inside the tank is equal to 770 liters.

The 600 kW heat generator was tested while working with and without turbulators inside the ninety tubes at the conditions reported in Table 1.5.1.

	<i>WITHOUT TURBULATORS</i>	<i>WITH TURBULATORS</i>
q_{burner} [kW]	600	600
m_{fuel} [kg h ⁻¹]	41.64	41.48
m_{water} [kg s ⁻¹]	6.7	6.95
ε [%]	20.37	20.62
T_{fuel} [°C]	27	25
T_{env} [°C]	28	25
$T_{water,IN}$ [°C]	61	60

Table 1.5.1. Working conditions for the 600 kW heat generator.

Results of the two simulations are reported in the following Table and are compared in Figure 1.5.2 against experimental global heat transfer data acquired by the manufacturer.

	<i>WITHOUT TURBULATORS</i>	<i>WITH TURBULATORS</i>
$T_{chimney}$ [°C]	241.4	80.9
$T_{water OUT}$ [°C]	80.9	79.9

Table 1.5.2. Simulated values of gas and water exit temperatures in the 600 kW fire tube heat generator operating at steady-state conditions.

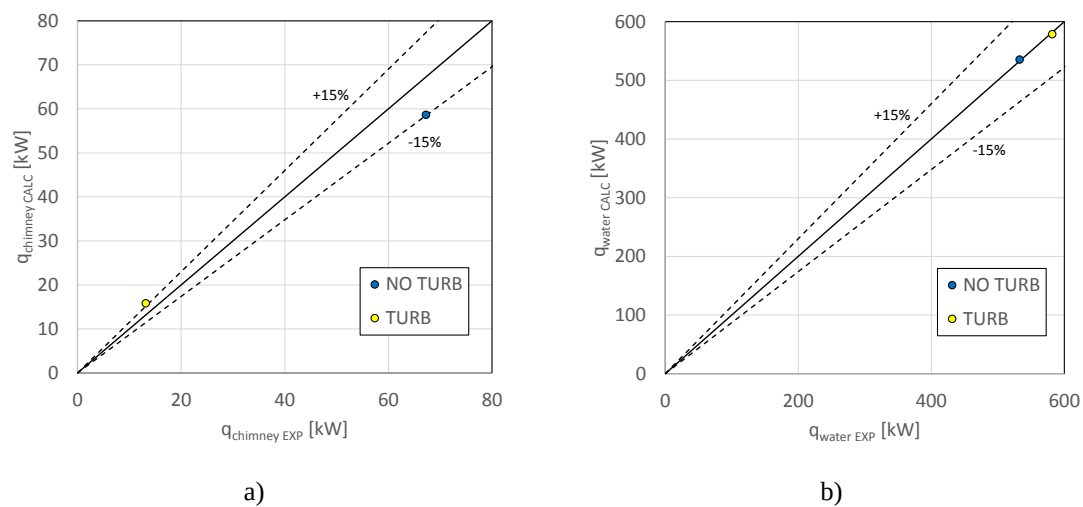


Figure 1.5.2. Experimental versus calculated heat flow rates lost at the chimney (a) and acquired by the water (b) for the 600 kW heat generator working with and without turbulators inside the pipes.

Even when simulating a heat generator having thermal power at the burner one order of magnitude higher, the experimental data are predicted within $\pm 15\%$ error band.

Finally, the model has been used for simulating performances of the 600 kW heat generator varying the number of tubes in the second and the third pass. Simulations were performed with and without turbulators inside the last pass, which has been simulated as formed by 50, 55 and 60 tubes. The total number of tubes in the system (second plus third pass) is fixed and equal to 90, thus the second pass has been simulated as divided into 40, 35 and 30 pipes, respectively. Pipes forming the second pass have the same geometrical characteristics of those forming the last pass.

Simulations were performed giving as input data to the model the values reported in Table 1.5.1 for the case “WITHOUT TURBULATORS”.

Figure 1.5.3 reports the results of the simulations, showing the heat lost at the chimney and the one acquired by the water as a function of the number of tubes composing the third pass. Tendency lines are also reported.

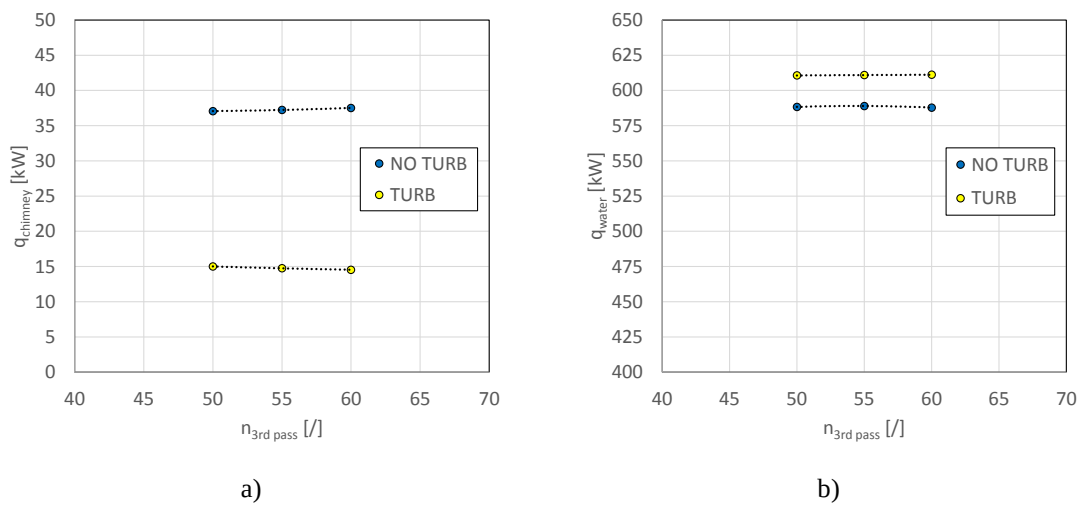


Figure 1.5.3. Heat flow rate lost at the chimney (a) and acquired by the water (b) as a function of the number of tubes in the last flue gases pass with and without turbulators for the 600 kW heat generator.

When working with the standard configuration (90 tubes in the third pass and one 248 mm diameter tube in the first inversion) thermal losses at the chimney are equal to 58.65 kW when working without turbulators and to 15.82 kW when inserting them. Thermal power acquired by the water in the tank is equal to 537.07 kW and to 609.84 kW, respectively.

When using turbulators, reducing the number of tubes in the third pass from 90 to 60 leads to a slight enhancement of the system performance. It has to be pointed out that a reduction of the number of tubes in the last pass means a reduction of the number of turbulence generator inserts

inside them, being these a great-impact element in the final cost of the heat generator. Thus, this analysis showed that a reduction of the number of pipes in the last flue gases pass is a possible and effective way to reduce system costs maintaining (or even increasing) system performances. When working without turbulators, reducing the number of tubes in the third pass leads to a significant enhancement of the heat generator performances. The new geometries present thermal losses at the chimney $\approx 35\%$ lower than the standard system, leading to an enhancement of the thermal efficiency of the system.

Figure 1.5.4 shows the thermal efficiency of the heat generator as a function of the number of pipes in the last flue gases pass, for the cases with and without turbulators. Tendency lines are also reported.

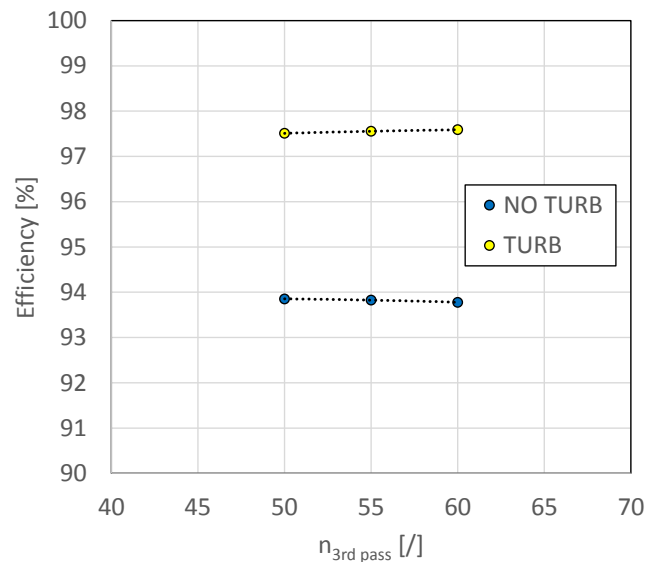


Figure 1.5.4. Heat generator thermal efficiency as a function of the number of tubes in the last flue gases pass with and without turbulators for the 600 kW heat generator.

The standard system presented an efficiency equal to 90.3% when working without turbulators and equal to 97.4% when inserting them. The Figure shows that the low-pipes-number solutions with turbulators present an efficiency which is similar (even slightly higher) to the standard solution, whit a reduction of the costs related to the inserts number. On the other hand, the simulations showed that by reducing the number of tubes in the last pass it is possible to have a significant enhancement of the system efficiency when working without turbulators.

However, the previous analysis accounts only for the thermal performances of the heat generator, without considering pressure losses in the flue gases line. When reducing the number of tubes in

the last pass, substituting the first inversion with a series of pipes, pressure losses inside the second and the third pass enhances.

Frictional pressure drop inside the tubes can be evaluated as follow

$$\Delta p = 2 f L \frac{G^2}{\rho D} \quad (1.36)$$

where D and L are the tube diameter and length, G is flue gases specific mass flow rate, ρ is flue gases density and f is the friction factor, calculated as

$$f = \begin{cases} \kappa_{TURB} \frac{16}{Re} & Re \leq 2100 \\ \kappa_{TURB} 0.0791 \cdot Re^{-0.25} & Re > 2100 \end{cases} \quad (1.37)$$

with Re flue gases Reynolds number.

When evaluating frictional losses in the absence of turbulators the corrective factor κ_{TURB} is equal to 1. When turbulators are inserted, κ_{TURB} is calculated through the same CFD analysis presented in Section 1.3.7, by comparing the frictional losses in the fitted tube with those in the plain one (Eq. (1.37) with $\kappa_{TURB} = 1$).

Figure 1.5.5 shows the results of the CFD simulations in terms of frictional losses.

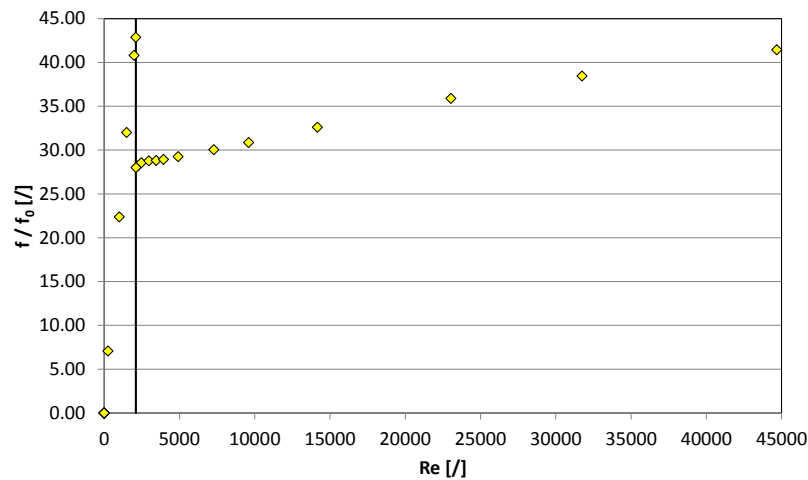


Figure 1.5.5. Ratio of the friction factor in the tube with turbulator to the one in the plain tube as a function of the Reynolds number of the flue gases.

From the results in Figure 1.5.5 the correlation to evaluate the corrective factor

$\kappa_{TURB} = f / f_0$, as a function of the flue gases Reynolds number, can thus be obtained:

$$\kappa_{TURB} = \begin{cases} -1.084 \cdot 10^{-6} Re^2 + 2.1913 \cdot 10^{-2} Re + 1.642; & \text{if } Re \leq 2100 \\ -6.481 \cdot 10^{-15} Re^4 + 1.545 \cdot 10^{-10} Re^3 - 1.285 \cdot 10^{-6} Re^2 + 4.701 \cdot 10^{-3} Re + 22.57; & \text{if } 2100 < Re \leq 10000 \\ -3.286 \cdot 10^{-9} Re^2 + 4.811 \cdot 10^{-4} Re + 26.52; & \text{if } Re > 10000 \end{cases} \quad (1.38)$$

Figure 1.5.6 shows flue gases pressure losses inside the second and the third pass as a function of the number of pipes of the last passage. Tendency lines are also reported.

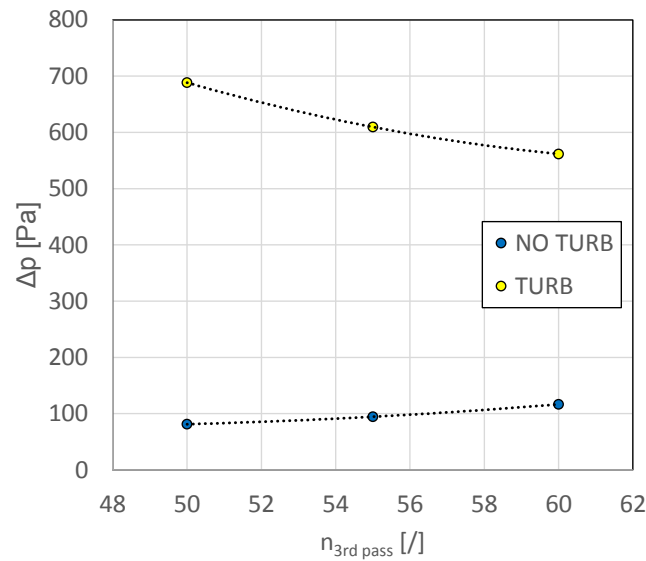


Figure 1.5.6. Flue gases pressure losses in the first inversion and in the pipes as a function of the number of tubes in the last pass, for the 600 kW heat generator. Trends for the cases with and without turbulators are reported.

When analyzing the standard geometry pressure losses resulted to be 24.3 Pa for the case without turbulators and 299.8 Pa for the case with turbulators. Reducing the number of pipes composing the third pass leads to a significant enhancement of the frictional pressure losses of the flue gases in the system, which means a higher pumping power consumption.

1.6 Conclusions

In the first Chapter of the present Thesis an experimental analysis of the working conditions and the heat transfer inside a three-pass fire tube heat generator, fed with natural gas, is presented. Water and flue gases internal temperatures were measured in the system during stationary and

dynamic regime operation. Tests were performed varying fuel mass flow rate, air index and inlet water temperature and mass flux.

The heat generator was tested with and without turbulators inserted inside the last flue gases pass. The experimental analysis showed the effect of turbulators presence on the temperature of the flue gases at the chimney. When working with turbulators inside the tubes, the exit temperatures of flue gases are 110 K – 120 K lower than those obtained, at similar operating conditions, without them. This means lower thermal losses at the chimney, thus higher efficiencies (+3% to +10%).

A dynamic model for predicting the behavior of the fire tube heat generator was developed and implemented. The model, implemented using MatLab/Simulink, is characterized by a subsystems structure that makes it easily adaptable to other configurations and geometries.

Effect of the turbulators presence was also accounted in the model through CFD analysis. The equations implemented in the dynamic model were presented. A deeper analysis of the enhancement of single-phase flue gases convective heat transfer will be presented in Sec. 2 of the present work.

The model has been validated with the heat generator working with a thermal power at the burner ranging between 70 kW and 87 kW. The comparison between experimental and calculated data provides good agreement (error band within $\pm 10\%$) both in stationary and transient regime.

The model has been used to verify the behavior of a larger scale heat generator, when modifying its geometrical configuration for costs reduction. Experimental global heat transfer data were used to verify the model results in the standard geometrical configuration. Then, the number of pipes in the third flue gases pass was reduced to increase flue gases velocity and thus reduce the number of turbulators inside the system, which allows important savings in the heat generator cost. The model showed that reducing the number of pipes, thus the number of turbulators, it is possible to achieve results similar (or even slightly better) than those obtained with the standard geometry.

Experimental data and modeling of fire tube generators are very rare in the literature, thus this Thesis provides relevant information with a validated model for such systems. The model can be used to predict the system behavior when operating with varying thermal power at the burner, varying conditions of the water inside the tank and with different geometries. Such a model can thus be used to optimize the stationary and dynamic working conditions and the characteristics of fire tube heat generators.

Nomenclature

A = Area, m^2

$/CO_2/$ = Carbon dioxide percentage in dry flue gases, %

$/CO_2/_{st}$ = Stoichiometric carbon dioxide percentage in dry flue gases, %

c_p = Isobaric specific heat capacity, $J\ kg^{-1}\ K^{-1}$

D = Diameter, m

$d/d\tau$ = Variation over time, s^{-1}

f = Friction factor, /

f_0 = Friction factor for plain tube, /

G = Specific mass flow rate, $kg\ m^{-2}\ s^{-1}$

H_U = Net heating value, $J\ kg^{-1}$

K = Corrective factor, /

k_{TURB} = Turbulators heat transfer corrective factor, /

L = Length, m

m = Mass flow rate, $kg\ s^{-1}$

$n_{3rd\ pass}$ = Number of tubes in the third flue gases pass, /

Nu = Nusselt number, /

Nu_0 = Nusselt number for plain tube, /

$Pr = \frac{c_p \mu}{\lambda}$ = Prandtl number, /

q = Heat flow rate, W

$\bar{q}$ = Heat flux, $W\ m^{-2}$

Ra = Rayleigh number, /

$Re = \frac{GD}{\mu}$ = Reynolds number, /

s = Thickness, m

S = Cross section area, m^2

T = Temperature, $^{\circ}C$

$\bar{T}$ = Mean temperature, $^{\circ}C$

U = Energy content, J

u_A = Type A uncertainty, %

u_B = Type B uncertainty, %

u_c = Combined standard uncertainty, %

u_M = Expanded uncertainty, %

V = Volume, m^3

Greek Symbols

α = Heat transfer coefficient, $W\ m^{-2}\ K^{-1}$

Δp = Pressure losses, Pa

ΔT_{ml} = Mean logarithmic temperature difference, K

ε = Air index, %

= Flame emissivity, /

K_{TURB} = Turbulators frictional corrective factor, /

λ = Thermal conductivity, $W\ m^{-1}\ K^{-1}$

μ = Viscosity, $Pa\ s^{-1}$

ρ = Density, $kg\ m^{-3}$

σ = Stefan-Boltzmann constant, $W\ m^{-2}\ K^{-4}$

Subscripts

C = Convective

CALC = Calculated

CO2 = Carbon dioxide

env = Environment

EXP = Experimental

ext = External

fg = Flue gases

fg->walls = From the flue gases to the walls

fur = Furnace

H2O = Steam

IN = Inlet

int = Internal

inv1 = First inversion

m = Mean

OUT = Outlet

R = Radiative

ref = Reference

walls->water = From the walls to the water

2 SINGLE-PHASE HEAT TRANSFER ENHANCEMENT BY MEANS OF TURBULATORS

Single phase heat transfer enhancement by means of turbulence generator inserts is an important field of research for heat transfer industry. In particular, inserting turbulators inside the last pass of fire tube heat generators allows to increase the system efficiency by enhancing flue gases convective heat transfer, reducing flue gases exit temperature thus associated thermal losses at the chimney.

When evaluating turbulators behavior it has to be reminded that they lead to an enhancement of both convective heat transfer coefficient and frictional losses along the tubes, thus their performances have to be evaluated through an enhancement factor which accounts for both the positive and the negative effects.

For practical industrial applications, in order to have inserts which are simple and cheap to manufacture, pressing methods are widely used for turbulators production. In the following Paragraphs, after a review of the most common solutions presented in the literature, performance enhancement of single wave and double wave geometries (which can be produced through pressing process) is evaluated by means of CFD simulations. Effects of several geometrical parameters, such as turbulator position inside the tube and pipe diameter, are analyzed with the simulations. Moreover, since these very kind of inters are actually used in heat generator industry, equations for predicting the heat transfer enhancement, the pressure losses enhancement and the global performance of the turbulators are presented.

Finally, double wave geometry has been updated with the aim to increase its performances. The new geometry and the results of the new simulations are presented at the end of this Chapter.

2.1 Introduction

In the past decade, heat transfer enhancement technology has been developed and widely applied to heat exchanger applications; as for example in refrigeration, automotives, process industry, solar water heater, etc (Webb, 1994; Kumar and Prasad, 2000).

The aim of augmentative heat transfer is to accommodate high heat fluxes (or heat transfer coefficients). To date, there have been a large number of attempts to reduce the size and costs of heat exchangers. The most significant variables in reducing the size and cost of a heat exchanger

are basically the heat transfer coefficient and pressure drop. An increase in the heat transfer coefficient generally leads to another advantage of reducing the temperature driving force, which increases the second law efficiency and decreases entropy generation. Thus, research in this area captivated the interest of a number of researchers.

The great attempt on utilizing different methods is to increase the heat transfer rate through the compulsory force convection. Meanwhile, it is found that this way can reduce the sizes of the heat exchanger device and save up energy. In general, the enhancing heat transfer techniques can be divided into two groups. One is the passive method, without stimulation by an external power, such as a surface coating, extended surfaces, swirl flow devices, convoluted (twisted) tubes and additives for liquid and gases. The other is the active method, which requires extra external power sources, as for example mechanical aids, surface-fluid vibration, injection and suction of the fluid, jet impingement and use of electrostatic fields (Eiamsa-ard and Promvonge, 2005).

Passive methods such as swirl flow devices, which cause an augmentation of the fluid turbulence that leads to an increase in the convective heat transfer coefficient, are widely used in heat generator industry. As an example, in a standard three pass fire tube heat generator these inserts, named turbulators, are placed inside the pipes forming the last pass to enhance the system thermal performances. It has been already shown in Chapter 1 of the present Thesis that a heat generator performs definitely better when working with turbulators inside the pipes than in the case of empty tubes, because of lower losses at the chimney due to the lower exit temperature of the flue gases.

The presence of turbulence generator inserts inside the tubes enhances the thermal performance of the system but at the same time it increases the pressure losses along the flue gases line, meaning a higher pumping power consumption. Thus, both thermal and frictional performances have to be taken into account when studying turbulators.

The thermal performance ratio η (a.k.a. enhancement efficiency) can be used to compare the convective heat transfer coefficient in a tube equipped with a turbulator (α) against the one in a plain (empty) tube (α_0), under the same pumping power.

For constant pumping power it is

$$(\dot{V}\Delta p)_0 = (\dot{V}\Delta p)_{TURB} \quad (2.1)$$

which in terms of friction factor and Reynolds number can be written as

$$(f \cdot Re^3)_0 = (f \cdot Re^3)_{TURB} \quad (2.2)$$

thus, the thermal performance ratio can be expressed as

$$\eta = \left| \frac{\alpha}{\alpha_0} \right|_{PP} = \left| \frac{Nu}{Nu_0} \right|_{PP} = \left(\frac{Nu}{Nu_0} \right) \left(\frac{f_0}{f} \right)^{1/3} \quad (2.3)$$

If the thermal performance ratio is bigger than one the turbulator leads to an effective increase of the system performances, otherwise the advantage related to the heat transfer enhancement is lower than the disadvantage related to the increased pumping power.

Twisted and helical geometries are, among the others, those mostly investigated in the literature. These kind of inserts performs quite well, as it will be shown in Section 2.2. However, for practical applications, simpler and easier-to-manufacture geometries have to be thought. Because of this, heat generators manufactures focused on “wave” turbulators, which can be produced through a simple press process. Thus, in the following Paragraphs the wave turbulators used by the industry will be presented, modeled and simulated, and the theoretical results will be discussed. Finally, an alternative solution will be studied and compared against the classical ones.

2.2 State of the art about turbulators analysis

Several works have been published in the recent years regarding experimental and theoretical analysis of the presence of a turbulator inside a tube, mainly with water or air as working fluid.

Twisted tape geometry has been widely treated by many authors. Eiamsa-ard *et al.* (2010a) experimentally studied the performances of a tube equipped with one or two parallel twisted tape inserts. Moreover, comparison between full length and regularly spaced turbulators was shown. The authors investigated the effect of the twist ratio, i.e. the ratio between the tape pitch length and width, as well as the effect of the space ratio, i.e. the ratio between the space between the inserts and the tube diameter, on the thermal and frictional performances of the system. Tests were performed using water as working fluid, varying the fluid Reynolds number between 4000 and 19000.

Figure 2.2.1 shows the enhancement efficiency obtained by Eiamsa-ard *et al.* (2010a). Left graph shows the comparison between the tube fitted with one (single) or two parallel (dual) twisted tape turbulators, for different twist ratio (y/w), as a function of the fluid Reynolds number. Right graph shows the comparison between the results obtained with the regularly spaced double twisted tape inserts, at different space ratio (s/D).

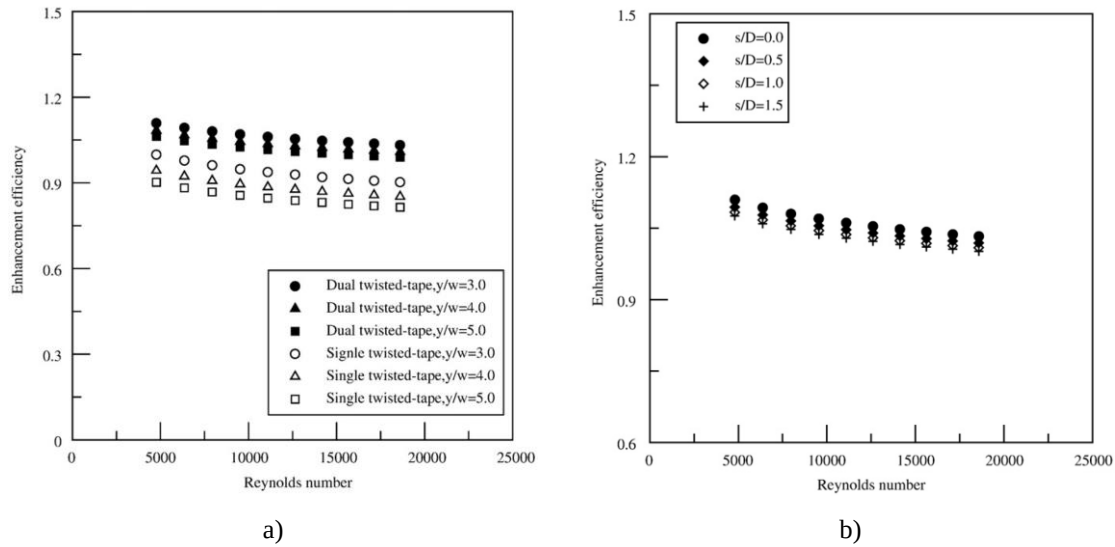


Figure 2.2.1. Enhancement efficiency as a function of the fluid Reynolds number. a) comparison between single and dual full length twisted tape turbulators at different twist ratio y/w . b) comparison between regularly spaced dual twisted tape turbulators ($y/w = 3$) at different space ratio s/d . Data from Eiamsa-ard *et al.* (2010a).

The analysis showed that the classical single twisted tape turbulator leads to an augmentation of the thermal performance of the system, but this positive effect is lower than the negative component related to the increased frictional losses, which leads to a thermal performance ratio of the single inserts lower than 1. The inserts enhancement efficiency increases when reducing the twist ratio, but the lower the twist ratio the more the turbulator is difficult (thus expensive) to manufacture.

The authors found that it is possible to get $\eta > 1$ by placing two parallel inserts inside the tube. On the other hand, this means doubling the number of turbulators inside the system. To reduce dual tape costs, maintaining efficiencies greater than one, the inserts can be regularly spaced with small space ratios.

Since standard twisted tape turbulators are not able to reach high thermal performance ratio, authors in the literature tried several ways to modify them with the aim to increase η . However, tested geometries are, for the majority of the cases, too complex and expensive to manufacture for practical applications.

As an example, Chang *et al.* (2007a) experimentally studied heat transfer and pressure losses enhancement in a tube fitted with serrated twisted tape turbulators having different twist ratio (y). The authors compared the results with those obtained using smooth twist inserts. Tests were performed using air as working fluid, varying the Reynolds number between 5000 and 25000. Figure 2.2.2 shows the thermal performance ratio obtained by Chang *et al.* (2007a).

◆	Serrated twisted tape with $y=1.56$		
■	Serrated twisted tape with $y=1.88$		
▲	Serrated twisted tape with $y=2.81$		
●	Serrated twisted tape with $y=\infty$		
◇	Smooth twisted tape with $y=1.56$		
□	Smooth twisted tape with $y=1.88$		
△	Smooth twisted tape with $y=2.81$		
○	Smooth twisted tape with $y=\infty$		

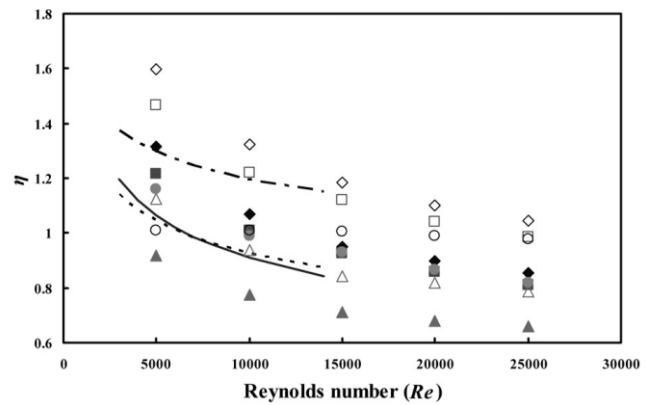


Figure 2.2.2. Thermal performance ratio of the serrated and smooth twisted tape turbulators as a function of the fluid Reynolds number, for different twist ratio y . Data from Chang *et al.* (2007a).

The authors showed that the serrated geometries lead to an increase of the thermal performance as compared to the smooth ones, but, if considering both thermal and frictional enhancement, smooth inserts perform better. Even if the serrated solutions present Nusselt numbers 25% - 70% higher than the smooth ones the increase of the friction factor leads to lower global performances. By comparing results in Figures 2.2.1 and 2.2.2 it is possible to see that by reducing the twist ratio of the smooth turbulators to very low values (e.g. 1.56) it is possible to get thermal performance ratios bigger than one. However, as already said, low twist ratio elements are difficult and expensive to manufacture.

Chang *et al.* (2007b) experimentally investigated heat transfer and pressure losses enhancement in a tube fitted with broken twisted tape inserts. The authors compared the data with those acquired with the serrated turbulators and with those calculated by Manglik and Bergles (1993 a,b) for smooth twist inserts. Tests were performed at various turbulators twist ratio (y), using air flowing with Re ranging between 1000 and 40000. Figure 2.2.3 shows the results by Chang *et al.* (2007b) in terms of turbulator enhancement efficiency.

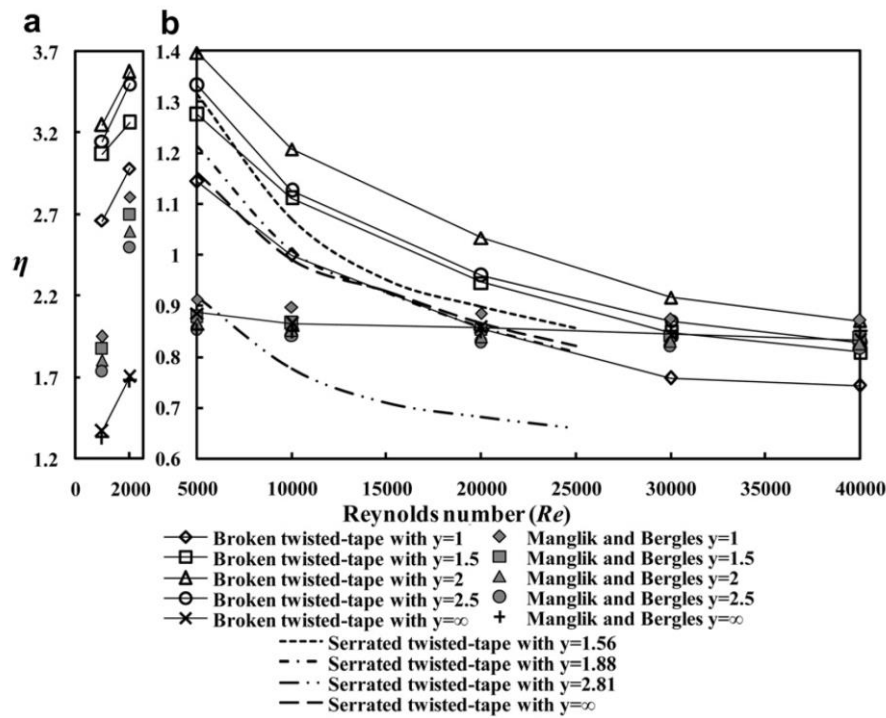


Figure 2.2.3. Thermal performance ratio of the broken twisted tape turbulators as a function of the fluid Reynolds number, for different twist ratio y , compared against values for the serrated and smooth geometries. Data from Chang *et al.* (2007b).

The tube fitted with the broken twisted tape inserts presents higher η factor than those fitted with the continuous and serrated elements, for a fixed twist ratio. Nevertheless, it is worthy to say that such a geometry is extremely complicated to realize, thus it is not a practical solution for heat generators applications.

From figure 2.2.3 it is possible to see that turbulators performances are extremely different between laminar and turbulent fluid flow regimes. When $Re < 2100$ the thermal performance ratio of the broken turbulators ranges in between 2.7 and 3.7 (for twist ratio $\neq \infty$) and the one of the smooth inserts ranges in between 1.7 and 2.8. For $Re > 2100$ the enhancement efficiency drastically reduces and, for the smooth inserts, it is always lower than 1. This kind of trend was also reported by Wongcharee and Eiamsa-ard (2011) and by Eiamsa-ard *et al.* (2010b).

Rahimi *et al.* (2009) experimentally analyzed heat transfer coefficient and friction factor enhancement in a tube equipped with twisted tape turbulators having different shape. The authors calculated the thermal performance ratio of classic, perforated, notched and jagged twisted tape inserts, using water as working fluid and varying the Reynolds number in between 3000 and 12000. The results of Rahimi *et al.* (2009) analysis are shown in Figure 2.2.4.

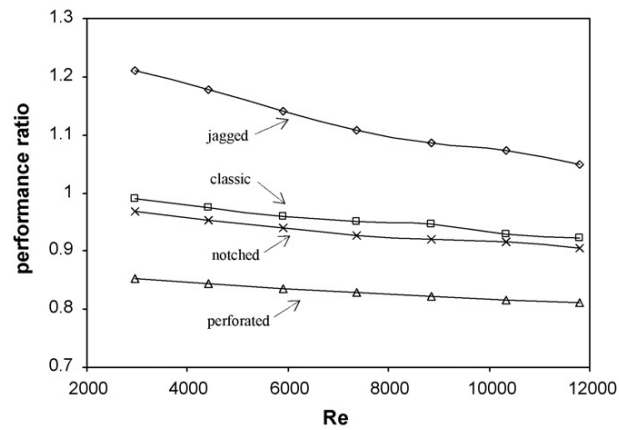


Figure 2.2.4. Thermal performance ratio of different types of twisted tape turbulators having twist ratio equal to 3. Data from Rahimi *et al.* (2009).

The Figure shows that the jagged solution is the only one which leads to enhancement efficiency greater than 1 in the actual range of Re .

Helical tape geometries have also been tested in the literature (Sivashanmugan and Suresh, 2006; Sivashanmugan and Suresh, 2007; Eiamsa-ard and Promvonge, 2005). Despite the complexity of the turbulator shape these geometries did not lead to substantial performance enhancement as compared to twisted tape inserts.

Louvered strip inserts were tested by Eiamsa-ard *et al.* (2008) and by Eiamsa-ard and Promvonge (2011). Tests were performed both with water and air as working fluid, varying louvers geometry and inclination angle as well as fluid flow direction. The authors found thermal performance ratios varying between 1.05 and 1.2 at $Re = 4000$ and between 0.9 and 1.05 at $Re = 20000$.

2.3 Actual geometries

Turbulator geometry for practical industrial applications should be the simplest and cheapest to manufacture as possible. Because of this, solutions obtained by pressing methods are mostly used. The heat generator presented in Section 1.2.1 of the present Thesis is usually equipped with turbulators having “single-wave” or “double-wave” geometry. The following Figures show a sketch of the two shapes.

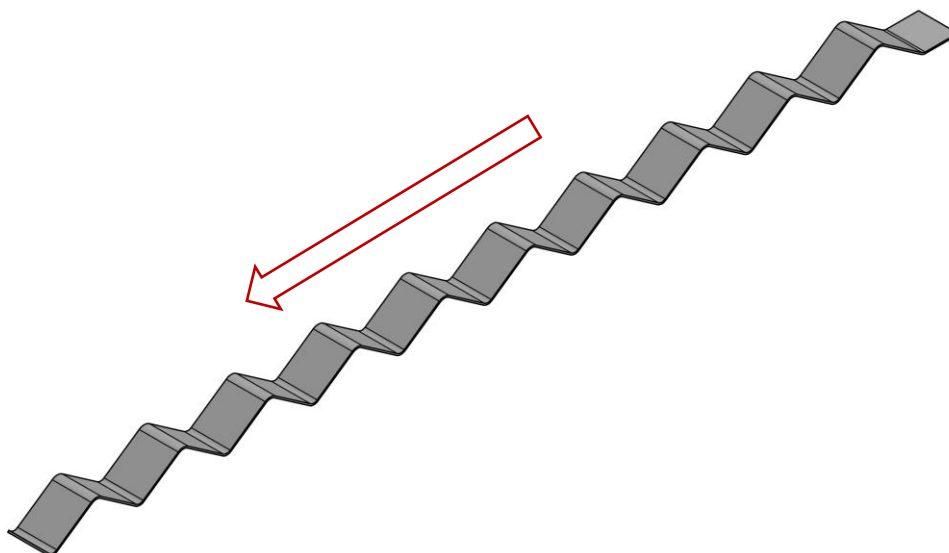


Figure 2.3.1. Sketch of the single-wave turbulator. Flue gases flow direction is showed by the arrow.

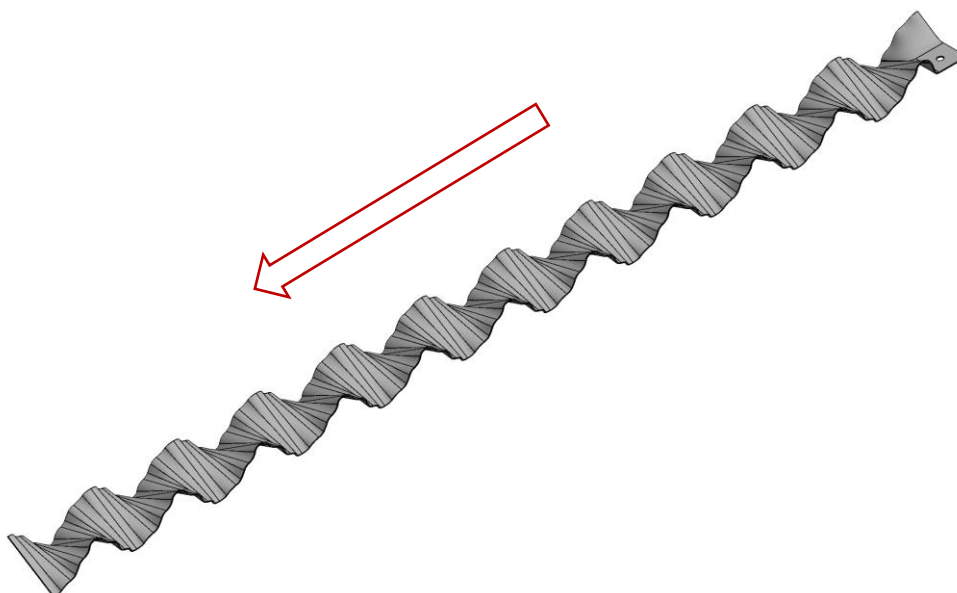


Figure 2.3.2. Sketch of the double-wave turbulator. Flue gases flow direction is showed by the arrow.

The length and width of the single wave turbulator are 617.5 mm and 26.5 mm, respectively. The insert is composed by 10 waves 20 mm high. Distance between two wave peaks is equal to 59 mm.

Double wave configuration can be considered as two staggered single wave turbulators placed close to each other. The width of the double wave insert is equal to 35.5 mm and its length is equal to 660 mm. The turbulator is composed by 11 double waves having a peak distance of 60 mm.

The two elements are simulated while placed inside a tube where flue gases flow. Tube length is fixed and equal to 826 mm. Tube internal diameter (D) as well as distance from the entrance of the pipe to the beginning of the turbulator (z) were varied. A summary of the simulated geometries is reported in Table 2.3.1.

SIMULATION n.	1	2	3	4	5
<i>WAVE TYPE</i>	<i>SINGLE</i>	<i>SINGLE</i>	<i>SINGLE</i>	<i>DOUBLE</i>	<i>DOUBLE</i>
D [mm]	36.4	36.4	42.5	42.5	42.5
z [mm]	85	25	85	85	25

Table 2.3.1. Simulated geometry.

A sketch of the geometry simulated in the case study n. 1 is shown in Figure 2.3.3. Since the single wave turbulator is vertically symmetric all the simulations regarding this geometry have been modeled considering only half tube.

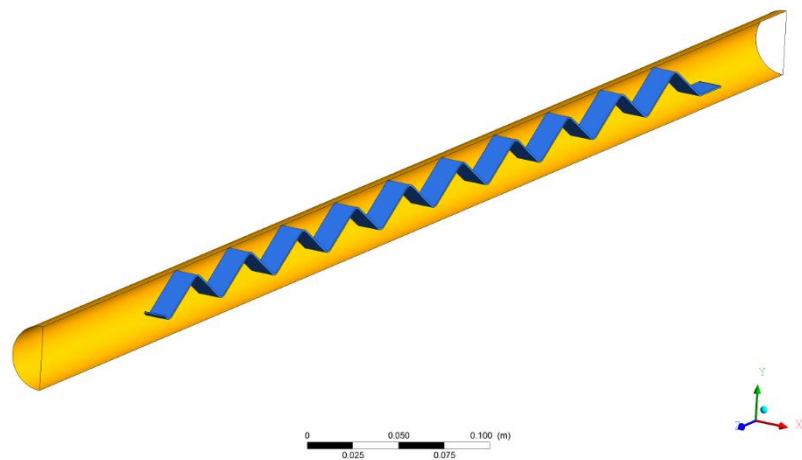


Figure 2.3.3. Geometry of simulation n.1.

2.4 CFD modeling

The fitted tube has been designed using AutoDesk INVENTOR 2014 software, while CFD analysis has been performed using ANSYS/FLUENT simulation tools. Simulations have been performed using a multi-CPU processor equipped with 16GB RAM. Typical simulation time is 1 – 2 days.

The system has been meshed using ICEM package. The (half) tube fitted with the single wave turbulator has been discretized in $\approx 4 \cdot 10^6$ elements, while the one fitted with the double-wave insert has been discretized in $\approx 9 \cdot 10^6$ elements. Figure 2.4.1 shows a sketch of the single-wave meshed tube.

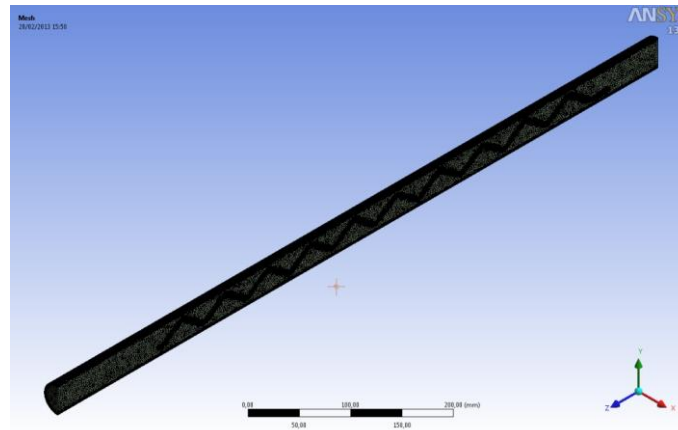


Figure 2.4.1. Mesh of the tube fitted with the single-wave turbulator.

Imposed boundary conditions are: the conditions ($u_{fg,IN}$, $T_{fg,IN}$) of the flue gases entering the tube, the temperature of the wall (T_{wall}) and the outlet relative pressure (p_{OUT}). Boundary conditions are constant for all the simulations, except for flue gases inlet velocity which is varied to cover a range of flue gases Reynolds number. The following Table reports a summary of the simulated conditions.

$u_{fg,IN}$ [m s^{-1}]	0.5 – 100
$T_{fg,IN}$ [$^{\circ}\text{C}$]	600
T_{wall} [$^{\circ}\text{C}$]	85.5
p_{OUT} [bar]	0

Table 2.4.1. Boundary conditions of the CFD simulations.

Flue gases thermodynamic properties are evaluated as proposed in Annaratone (2008). For the single-wave turbulator, the vertical middle plane is imposed symmetrical.

Eiamsa-ard *et al.* (2009) numerically studied twisted tape turbulators by means of CFD simulations, testing different viscosity models. The authors compared their predicted results with experimental values obtained by Manglik and Bergles (1993b), as shown in Figure 2.4.2.

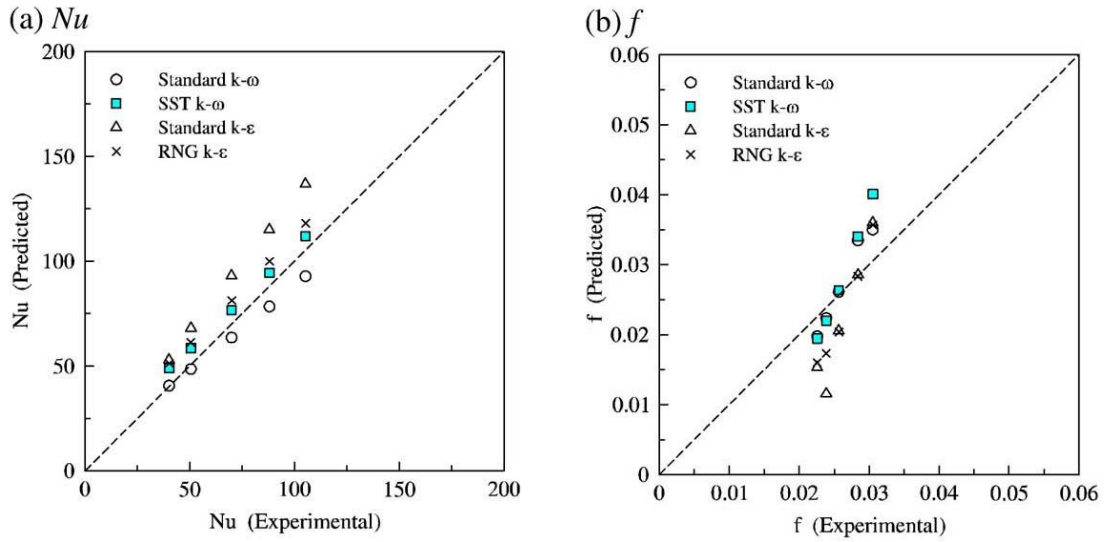


Figure 2.4.2. Comparison between experimental and CFD simulated Nusselt number (left) and friction factor (right) for different turbulence models. Graphs from Eiamsa-ard et al. (2009).

The authors found the Shear Stress Transport (SST) k - ω turbulence model to be the one that gives the best predictions. Calculated values fall within the $\pm 12.2\%$ error band for the Nusselt number and within the $\pm 6.4\%$ error band for the friction factor. Thus, SST k - ω turbulence model with low Re correction has been used in the present analysis.

2.5 Data reduction

Simulations output are the temperature of the flue gases at the exit of the tube ($T_{fg,OUT}$), the exchanged heat flow rate (q) and the inlet pressure (p_{IN}).

Once inlet and outlet flue gases temperatures and wall temperature are known it is possible to calculate the mean logarithmic temperature difference between the gases and the wall ΔT_{ml} . Knowing the heat transfer area A it is thus possible to evaluate the convective heat transfer coefficient between the fluid and the wall

$$\alpha = \frac{q}{A \cdot \Delta T_{ml}} \quad (2.4)$$

Evaluating fluid thermodynamic properties at the mean condition between the inlet and the outlet of the tube it is possible to calculate the Nusselt number of the fitted tube as

$$Nu = \frac{\alpha D}{\lambda_{fg}} \quad (2.5)$$

with D internal diameter of the tube.

The friction factor inside the tube can be evaluated as

$$f = \frac{1}{2} \frac{D}{\rho_{fg} u_{fg}^2} \frac{\Delta p}{L} \quad (2.6)$$

where $\Delta p = p_{IN} - p_{OUT}$ is the pressure drop between the inlet and the outlet of the tube and L is the tube length.

Knowing flue gases velocity and thermodynamic properties it is possible to evaluate the fluid Reynolds and Prandlt numbers

$$Re = \frac{\rho_{fg} u_{fg} D}{\mu_{fg}} \quad (2.7)$$

$$Pr = \frac{c_{p, fg} \mu_{fg}}{\lambda_{fg}} \quad (2.8)$$

In Eqs. (2.6) and (2.7) the velocity of the flue gases is corrected by considering the flue gases density at the mean temperature between the inlet and the outlet. It can be seen that Reynolds number in Eq. (2.7), which will be used for results comparison, is evaluated as for plain tubes.

Once Reynolds and Prandlt numbers are known it is possible to evaluate the Nusselt number and the friction factor for the empty tube as

$$Nu_0 = \begin{cases} 1.86 \left(Re \cdot Pr \frac{D}{L} \right)^{1/3} \left(\frac{\mu_{fg}}{\mu_{walls}} \right)^{0.14} ; & \text{if } Re < 2100 \\ 0.023 Re^{0.8} Pr^{0.4} ; & \text{if } Re \geq 2100 \end{cases} \quad (2.9)$$

$$f_0 = \begin{cases} 16/Re ; & \text{if } Re < 2100 \\ 0.0791 \cdot Re^{-0.25} ; & \text{if } Re \geq 2100 \end{cases} \quad (2.10)$$

Once predicted values for the plain tube are known it is finally possible to evaluate the thermal enhancement (Nu/Nu_0) and the frictional enhancement (f/f_0) related to the turbulator presence, and thus the thermal performance ratio as in Eq. (2.3).

2.6 Results

In this Section, the theoretical results of the five simulations previously introduced are presented. Moreover, a preliminary validation of the calculated output using the heat generator model and the experimental data acquired in the three pass fire tube generator equipped with the ten wave turbulator (both presented in Chapter 1 of the present Thesis) is reported.

2.6.1 Theoretical analysis

Figure 2.6.1 shows the results obtained with the single wave turbulator placed 85 mm inside the 36.4 mm tube (Simulation n. 1 of Table 2.3.1), in terms of thermal enhancement, frictional enhancement and enhancement efficiency.

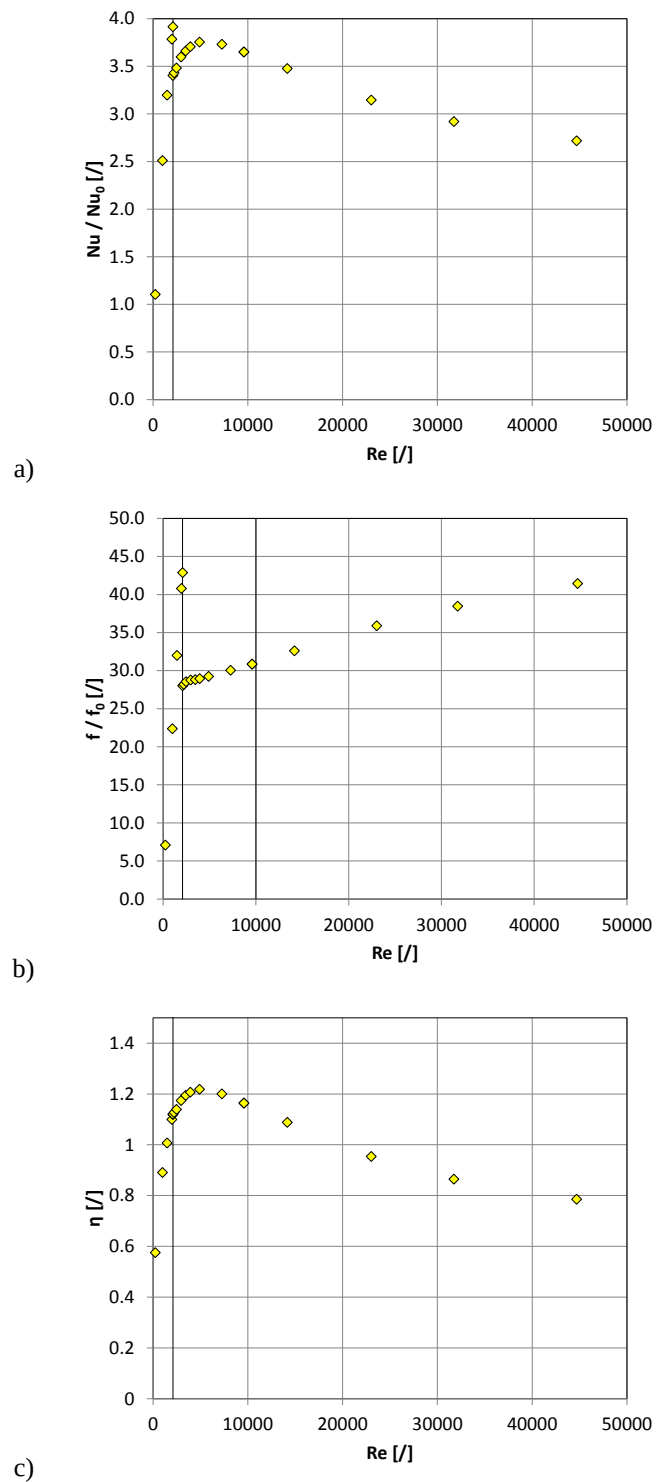


Figure 2.6.1. Results of CFD simulation n.1 in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c).

Turbulator presence increases flue gases Nusselt number. The enhancement follows a linear trend while the fluid flows in laminar regime. Then, for $Re > 2100$, the ratio between the actual and the plain tube Nusselt numbers increases until it reach a maximum for $Re \approx 5000$, over which it tends to decrease.

In the laminar zone friction factor enhancement trend is similar to Nusselt number ratio trend, since it increases linearly with Re , while in the transition and turbulent zones the behavior is different. When $Re > 2100$ the frictional enhancement increases almost linearly, with a slope which is gentler than the one relative to the $Re < 2100$ zone.

By combining the Nu and f enhancement it is possible to get the trend of the thermal performance ratio for the simulated case. As it can be seen from Figure 2.6.1c η increases for Reynolds numbers lower than ≈ 5000 , reaching a maximum value equal to 1.22, and then decreases. It is possible to see that the enhancement efficiency is bigger than one in the range $Re = 1400 - 20000$, while for extremely low and extremely high Reynolds numbers (thus flue gases velocities) introducing such a turbulator is not convenient.

By comparing simulation results with the literature data presented in Section 2.2 it is possible to infer that, even if the single wave geometry is much more simple to manufacture than the twisted tape one (especially at low twist ratio), this presents thermal performance ratios similar to those obtained using the twisted inserts, in the transient and turbulent zones. On the contrary, for $Re < 2100$, more complex solutions perform better.

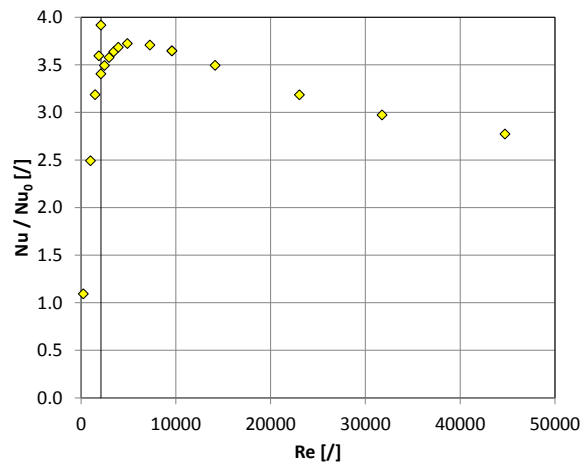
Interpolating the data presented in Figure 2.6.1 it is finally possible to develop predictive equations to get the thermal enhancement, the frictional enhancement and the thermal performance ratio for the single wave turbulator placed 85 mm inside the 36.4 mm tube, as a function of the flue gases Reynolds number. The equations presented hereinafter are different for the laminar ($Re < 2100$), transient ($2100 < Re < 10000$) and turbulent ($Re > 10000$) flow regimes.

$$\frac{Nu}{Nu_0} = \begin{cases} -3.05 \cdot 10^{-7} Re^2 + 2.23 \cdot 10^{-3} Re + 0.569; & \text{if } Re \leq 2100 \\ -3.98 \cdot 10^{-16} Re^4 + 1.23 \cdot 10^{-11} Re^3 - 1.46 \cdot 10^{-7} Re^2 + 7.51 \cdot 10^{-4} Re + 2.356; & \text{if } 2100 < Re < 10000 \\ 5.05 \cdot 10^{-10} Re^2 - 5.42 \cdot 10^{-5} Re + 4.132; & \text{if } Re \geq 10000 \end{cases} \quad (2.11)$$

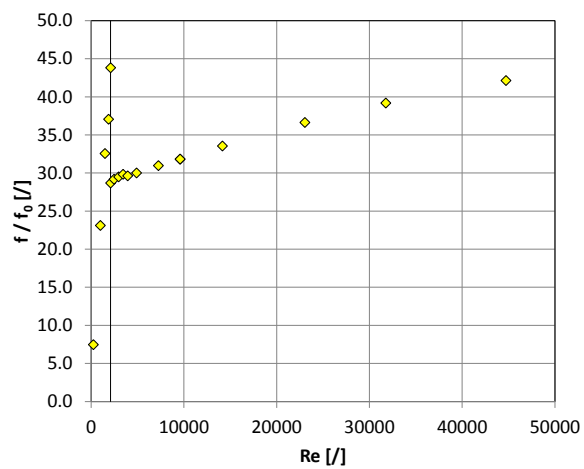
$$\frac{f}{f_0} = \begin{cases} -1.08 \cdot 10^{-6} Re^2 + 2.19 \cdot 10^{-2} Re + 1.642; & \text{if } Re \leq 2100 \\ 7.03 \cdot 10^{-12} Re^3 - 1.29 \cdot 10^{-7} Re^2 + 1.05 \cdot 10^{-3} Re + 26.458; & \text{if } 2100 < Re < 10000 \\ -3.29 \cdot 10^{-9} Re^2 + 4.81 \cdot 10^{-4} Re + 26.516; & \text{if } Re \geq 10000 \end{cases} \quad (2.12)$$

$$\eta = \begin{cases} -1.05 \cdot 10^{-7} Re^2 + 5.37 \cdot 10^{-4} Re + 0.449; & \text{if } Re \leq 2100 \\ -4.67 \cdot 10^{-17} Re^4 + 2.05 \cdot 10^{-12} Re^3 - 3.12 \cdot 10^{-8} Re^2 + 1.85 \cdot 10^{-4} Re + 0.848; & \text{if } 2100 < Re < 10000 \\ 2.18 \cdot 10^{-10} Re^2 - 2.26 \cdot 10^{-5} Re + 1.363; & \text{if } Re \geq 10000 \end{cases} \quad (2.13)$$

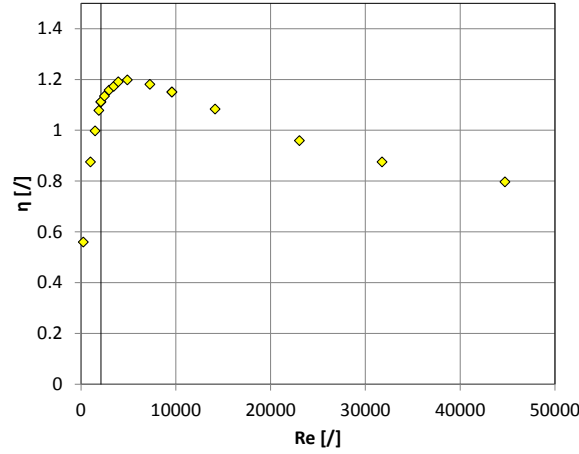
Figure 2.6.2 shows the results obtained with simulation n.2 (single wave turbulator, $D = 36.4$ mm, $z = 25$ mm).



a)



b)



c)

Figure 2.6.2. Results of CFD simulation n.2 in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c).

Trends are similar to those obtained for the first simulation. Equations for the single wave turbulator placed 25 mm inside the 36.4 mm tube are reported hereinafter.

$$\frac{Nu}{Nu_0} = \begin{cases} -2.98 \cdot 10^{-7} Re^2 + 2.20 \cdot 10^{-3} Re + 0.569; & \text{if } Re \leq 2100 \\ -3.33 \cdot 10^{-16} Re^4 + 1.05 \cdot 10^{-11} Re^3 - 1.26 \cdot 10^{-7} Re^2 + 6.55 \cdot 10^{-4} Re + 2.495; & \text{if } 2100 < Re < 10000 \\ 4.32 \cdot 10^{-10} Re^2 - 4.86 \cdot 10^{-5} Re + 4.082; & \text{if } Re \geq 10000 \end{cases} \quad (2.14)$$

$$\frac{f}{f_0} = \begin{cases} -1.24 \cdot 10^{-6} Re^2 + 2.19 \cdot 10^{-2} Re + 2.205; & \text{if } Re \leq 2100 \\ 7.64 \cdot 10^{-12} Re^3 - 1.41 \cdot 10^{-7} Re^2 + 1.16 \cdot 10^{-3} Re + 26.963; & \text{if } 2100 < Re < 10000 \\ -2.92 \cdot 10^{-9} Re^2 + 4.53 \cdot 10^{-4} Re + 27.738; & \text{if } Re \geq 10000 \end{cases} \quad (2.15)$$

$$\eta = \begin{cases} -1.03 \cdot 10^{-7} Re^2 + 5.38 \cdot 10^{-4} Re + 0.433; & \text{if } Re \leq 2100 \\ 8.62 \cdot 10^{-13} Re^3 - 1.98 \cdot 10^{-8} Re^2 + 1.37 \cdot 10^{-4} Re + 0.905; & \text{if } 2100 < Re < 10000 \\ 1.87 \cdot 10^{-10} Re^2 - 2.03 \cdot 10^{-5} Re + 1.33; & \text{if } Re \geq 10000 \end{cases} \quad (2.16)$$

Figure 2.6.3 compares results obtained with the single wave turbulator placed 85 mm and 25 mm apart the tube entrance (i.e. it compares simulations n.1 and n.2).

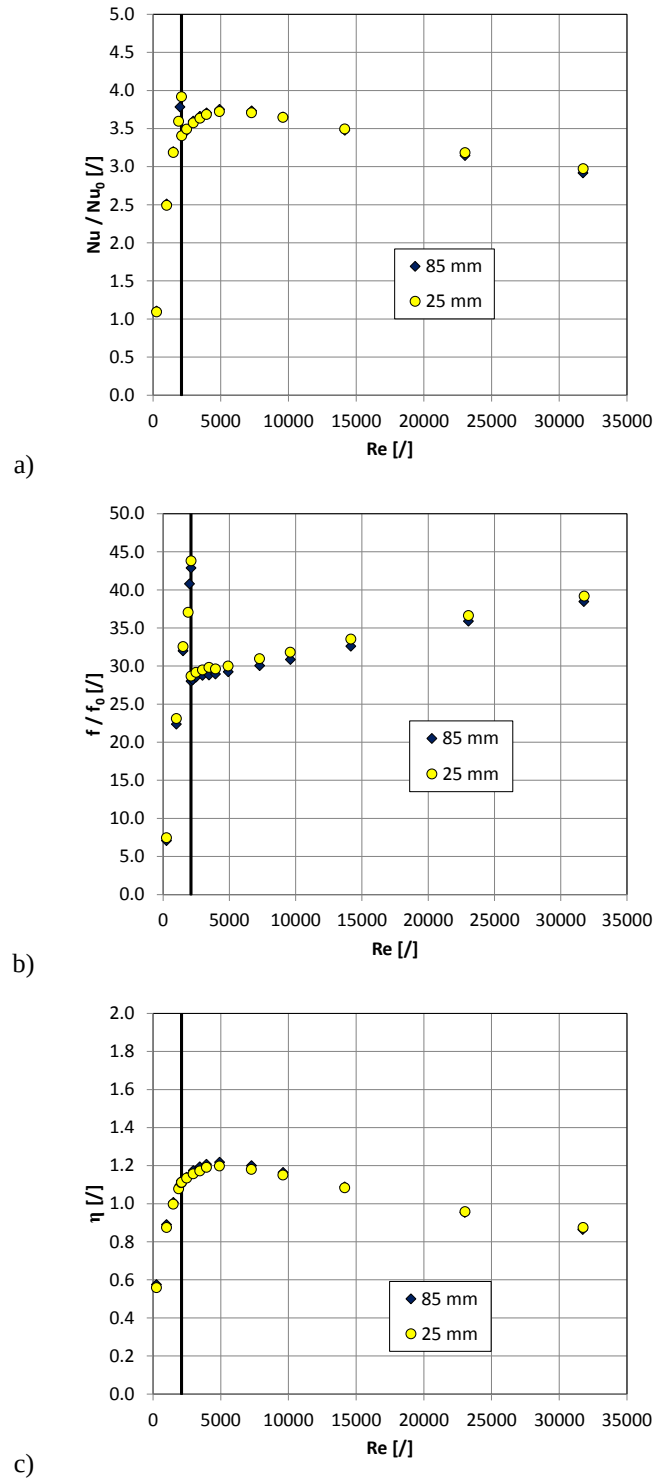
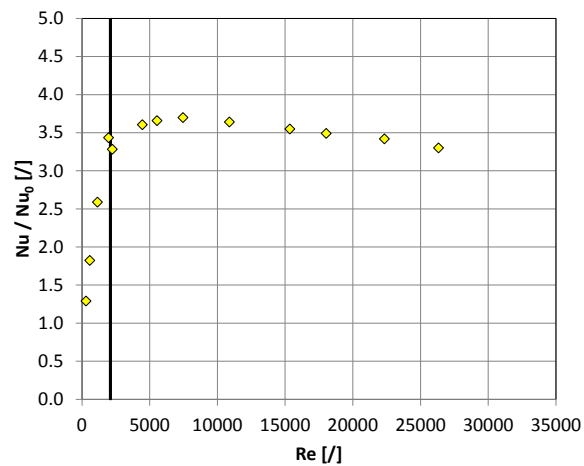


Figure 2.6.3. Comparison between the results of simulations n.1 ($z = 85$ mm) and n.2 ($z = 25$ mm) in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c).

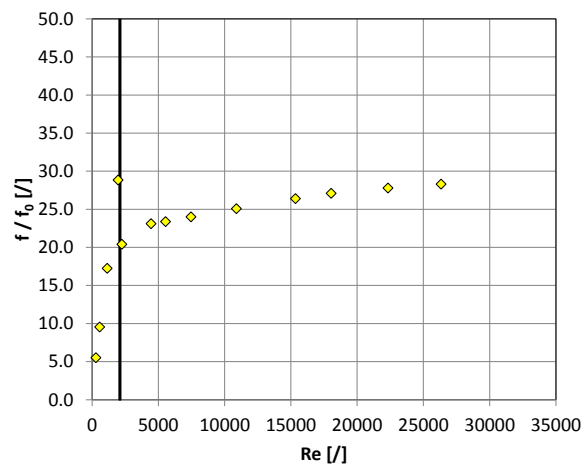
The graphs in the Figure show than by moving the turbulator closer to the tube entrance no sensible effect on its performances is achieved. Nusselt number ratios between the two cases are almost identical. The turbulator placed 85 mm apart from the tube entrance performs slightly

better than the one placed 25 mm inside in terms of frictional losses, which reflects in a slightly higher thermal performance ratio.

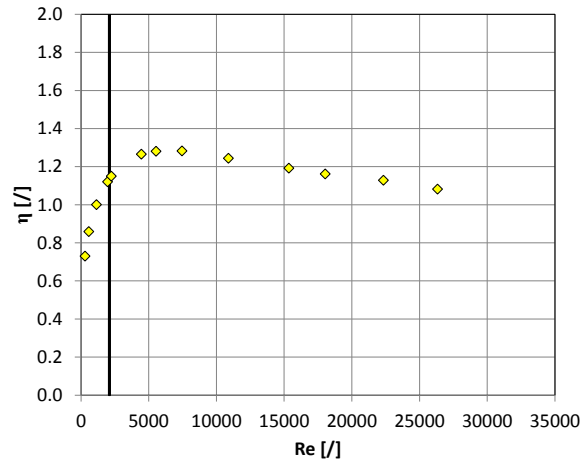
Figure 2.6.4 shows the results of the simulation n.3, which refers to the single wave turbulator placed 85 mm inside the bigger tube ($D = 42.5$). Figure 2.6.5 shows the comparison between the results obtained with the same turbulator into the two different tubes (i.e. comparison between simulations n.1 and n.3).



a)

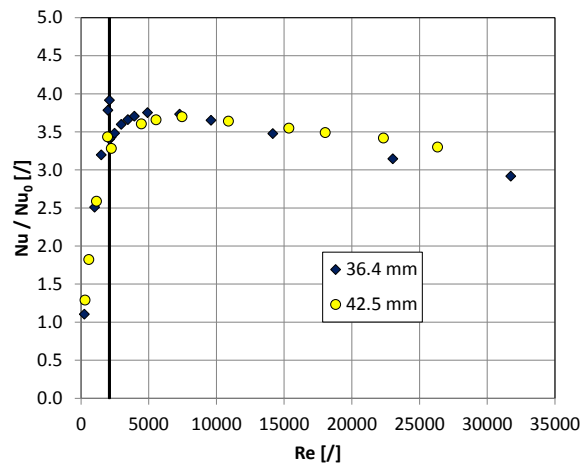


b)

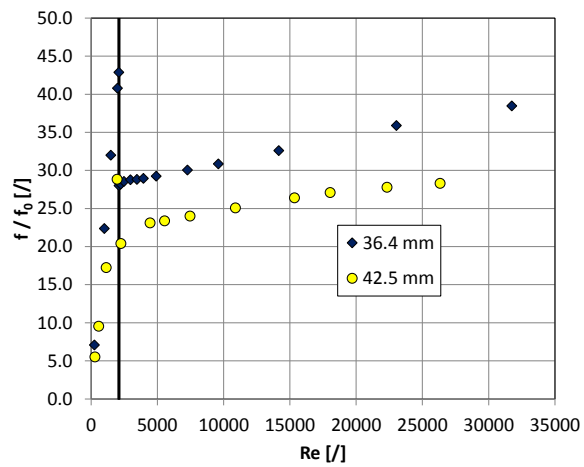


c)

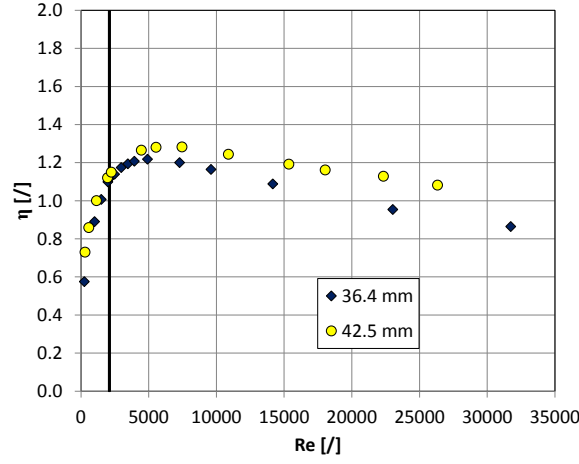
Figure 2.6.4. Results of CFD simulation n.3 in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c).



a)



b)



c)

Figure 2.6.5. Comparison between the results of simulations n.1 ($D = 36.4$ mm) and n.3 ($D = 42.5$ mm) in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). In both cases turbulator is placed 85 mm inside the tube entrance.

Trends obtained with simulation n3. are similar to the previous ones.

Moreover, Figure 2.6.5 shows that the Nusselt number enhancement due to the turbulator presence is not significantly affected by an increase in the tube internal diameter. However, Nu/Nu_0 ratio for the 42.5 mm tube is slightly lower than the one for the smaller tube in the range $Re \approx 1200 - 10000$, and slightly higher outside this zone.

Nevertheless, frictional losses inside the 42.5 mm tube are much lower than those inside the 36.4 mm tube. This leads to a thermal performance ratio $\approx 20\%$ higher when using the bigger tube, especially when the fluid is flowing in the transient or turbulent regime.

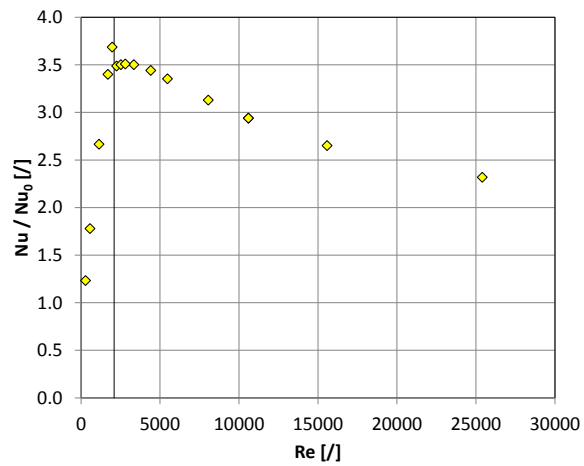
In the following Equations correlations for the 42.5 mm tube equipped with the single wave turbulator are reported.

$$\frac{Nu}{Nu_0} = \begin{cases} -3.24 \cdot 10^{-7} Re^2 + 2.00 \cdot 10^{-3} Re + 0.752; & \text{if } Re \leq 2100 \\ 1.55 \cdot 10^{-12} Re^3 - 4.29 \cdot 10^{-8} Re^2 + 3.75 \cdot 10^{-4} Re + 2.642; & \text{if } 2100 < Re < 10000 \\ -2.59 \cdot 10^{-10} Re^2 - 1.17 \cdot 10^{-5} Re + 3.795; & \text{if } Re \geq 10000 \end{cases} \quad (2.17)$$

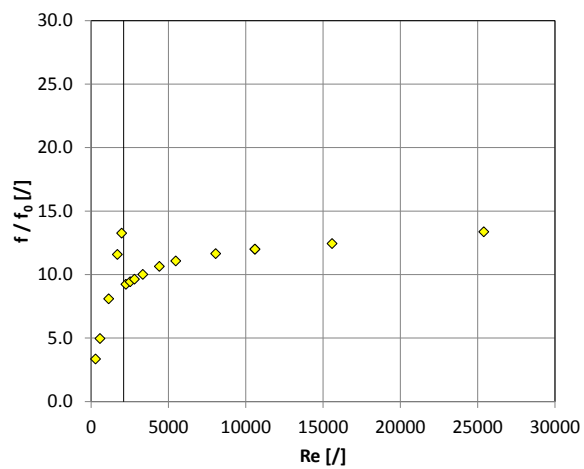
$$\frac{f}{f_0} = \begin{cases} 1.69 \cdot 10^{-8} Re^2 + 1.39 \cdot 10^{-2} Re + 1.518; & \text{if } Re \leq 2100 \\ 1.90 \cdot 10^{-11} Re^3 - 4.31 \cdot 10^{-7} Re^2 + 3.38 \cdot 10^{-3} Re + 14.793; & \text{if } 2100 < Re < 10000 \\ -8.23 \cdot 10^{-9} Re^2 + 5.14 \cdot 10^{-4} Re + 20.465; & \text{if } Re \geq 10000 \end{cases} \quad (2.18)$$

$$\eta = \begin{cases} -1.12 \cdot 10^{-7} Re^2 + 4.81 \cdot 10^{-4} Re + 0.607; & \text{if } Re \leq 2100 \\ 6.92 \cdot 10^{-13} Re^3 - 1.84 \cdot 10^{-8} Re^2 + 1.50 \cdot 10^{-4} Re + 0.899; & \text{if } 2100 < Re < 10000 \\ 7.23 \cdot 10^{-11} Re^2 - 1.29 \cdot 10^{-5} Re + 1.375; & \text{if } Re \geq 10000 \end{cases} \quad (2.19)$$

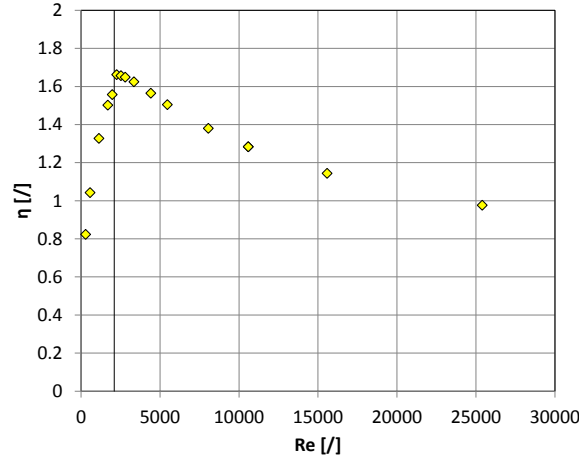
The following Figure presents the data obtained with the double wave turbulator, equipped 85 mm inside the 42.5 mm tube (simulation n.4).



a)



b)



c)

Figure 2.6.6. Results of CFD simulation n.4 in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c).

As it can be seen from the Figure, the trend obtained with the double wave insert is similar to the one got with the single wave turbulator, but the maximum thermal enhancement and enhancement efficiency are achieved with the doubled geometry for a lower value of Re (≈ 2500). Equations for the double wave turbulator obtained from simulation n.4 are reported hereinafter.

$$\frac{Nu}{Nu_0} = \begin{cases} -2.89 \cdot 10^{-7} Re^2 + 2.10 \cdot 10^{-3} Re + 0.658; & \text{if } Re \leq 2100 \\ 3.71 \cdot 10^{-12} Re^3 - 6.58 \cdot 10^{-8} Re^2 + 2.90 \cdot 10^{-4} Re + 3.129; & \text{if } 2100 < Re < 10000 \\ 1.71 \cdot 10^{-9} Re^2 - 1.04 \cdot 10^{-4} Re + 3.853; & \text{if } Re \geq 10000 \end{cases} \quad (2.20)$$

$$\frac{f}{f_0} = \begin{cases} 2.95 \cdot 10^{-7} Re^2 + 5.20 \cdot 10^{-3} Re + 1.851; & \text{if } Re \leq 2100 \\ -6.23 \cdot 10^{-8} Re^2 + 1.06 \cdot 10^{-3} Re + 7.162; & \text{if } 2100 < Re < 10000 \\ -6.84 \cdot 10^{-10} Re^2 + 1.20 \cdot 10^{-4} Re + 10.761; & \text{if } Re \geq 10000 \end{cases} \quad (2.21)$$

$$\eta = \begin{cases} -1.94 \cdot 10^{-7} Re^2 + 8.65 \cdot 10^{-4} Re + 0.598; & \text{if } Re \leq 2100 \\ -3.17 \cdot 10^{-10} Re^2 - 4.71 \cdot 10^{-4} Re + 1.778; & \text{if } 2100 < Re < 10000 \\ 8.34 \cdot 10^{-10} Re^2 - 5.10 \cdot 10^{-5} Re + 1.735; & \text{if } Re \geq 10000 \end{cases} \quad (2.22)$$

Figure 2.6.7 shows the comparison between the results obtained with the single and the double wave turbulator, both installed 85 mm apart the 42.5 mm tube entrance.

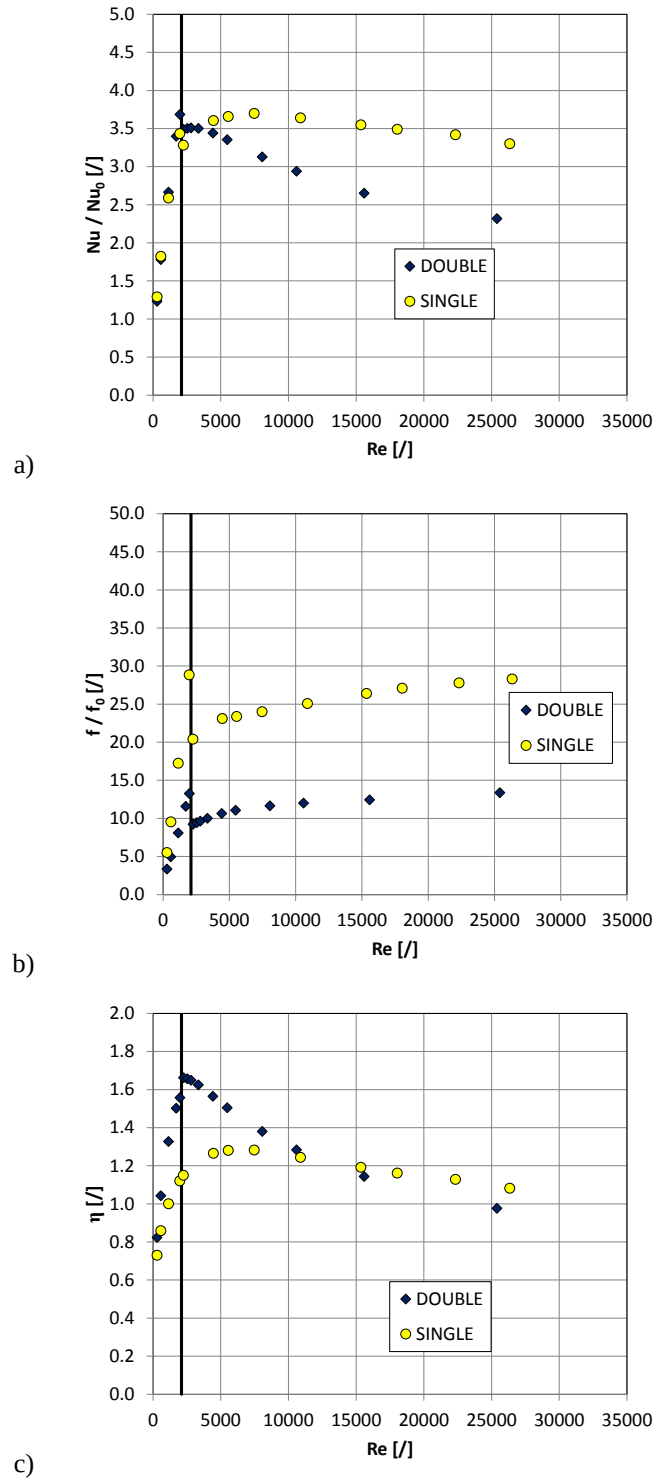


Figure 2.6.7. Comparison between the results of simulations n.3 (single wave turbulator) and n.4 (double wave turbulator) in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). In both cases turbulator is placed 85 mm inside the tube (42.5 mm diameter) entrance.

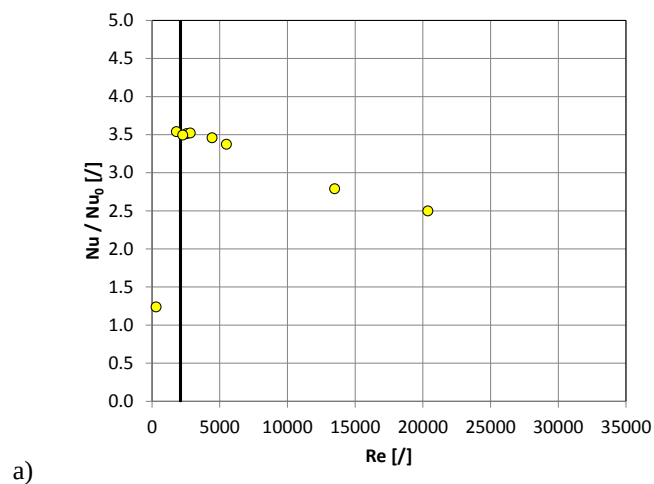
The Figure shows that the two turbulators perform similarly, in terms of Nu/Nu_0 ratio, when the flue gases flow in laminar regime while, for $Re > 2100$, the thermal enhancement of the single

wave insert is higher than the one of the double system, the more the bigger is the Reynolds number.

Nevertheless, the double wave turbulator presents much lower friction factor enhancement than the single geometry, both in laminar and turbulent regimes. This reflects in a higher thermal performance ratio, as can be seen in Figure 2.6.7c, in the laminar and transient fluid flow regimes. The enhancement efficiency of the double wave insert has a peak value of about 1.65 at $Re \approx 2500$. When reaching high Reynolds number values (i.e. bigger than 10000) the reduced thermal performance of the double wave system compared to the single wave one is no more balanced by the lower frictional losses, thus the single wave turbulator presents higher thermal performance ratio.

Comparing the results obtained with the double wave geometry with those found in the literature (Section 2.2) it is possible to infer that this kind of turbulator performs very well, especially in the transient and turbulent flow regions.

Results of simulation n.5 are hereinafter reported. Since theoretical analysis of the effect of moving the turbulator closer to the tube entrance was already performed through simulations n.1 and n.2, only a limited number of points were collected for simulation n.5, with the aim to verify the trend also with the double wave geometry.



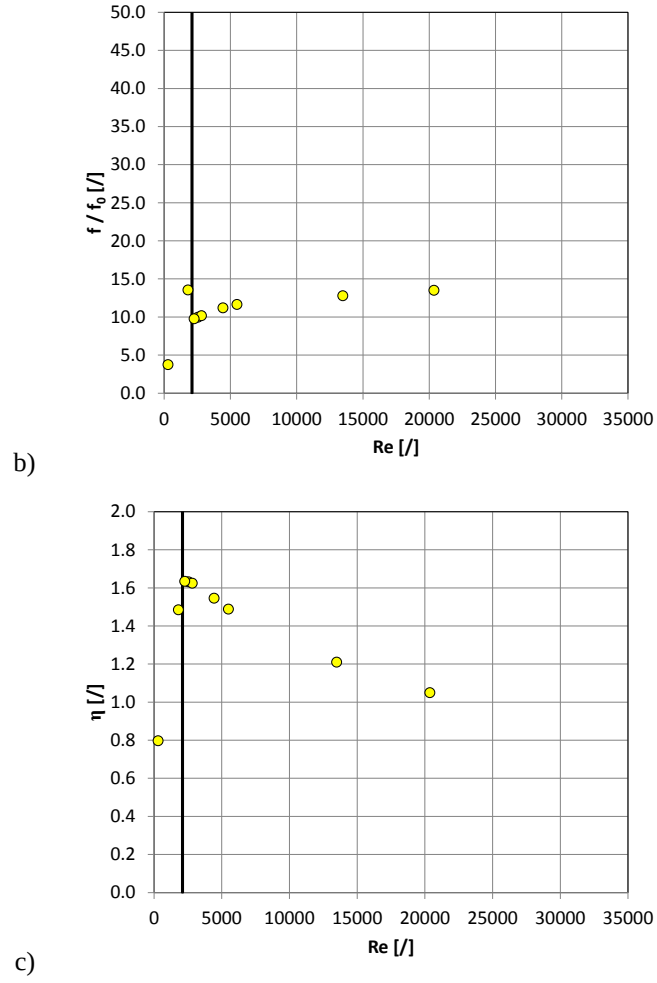


Figure 2.6.8. Results of CFD simulation n.5 in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c).

Equations for the double wave turbulator placed at $z = 25$ mm are the following.

$$\frac{Nu}{Nu_0} = \begin{cases} 1.54 \cdot 10^{-3} Re + 0.795; & \text{if } Re \leq 2100 \\ 3.08 \cdot 10^{-13} Re^3 - 9.85 \cdot 10^{-9} Re^2 + 2.34 \cdot 10^{-5} Re + 3.509; & \text{if } Re > 2100 \end{cases} \quad (2.23)$$

$$\frac{f}{f_0} = \begin{cases} 6.54 \cdot 10^{-3} Re + 1.858; & \text{if } Re \leq 2100 \\ -1.37 \cdot 10^{-8} Re^2 + 4.95 \cdot 10^{-4} Re + 8.97; & \text{if } Re > 2100 \end{cases} \quad (2.24)$$

$$\eta = \begin{cases} 4.59 \cdot 10^{-4} Re + 0.665; & \text{if } Re \leq 2100 \\ 8.66 \cdot 10^{-10} Re^2 - 5.23 \cdot 10^{-5} Re + 1.757; & \text{if } Re > 2100 \end{cases} \quad (2.25)$$

Finally, comparison between the simulations performed with the double wave insert placed 85 mm and 25 mm inside the tube (simulations n.4 and n.5) is shown in the following Figure.

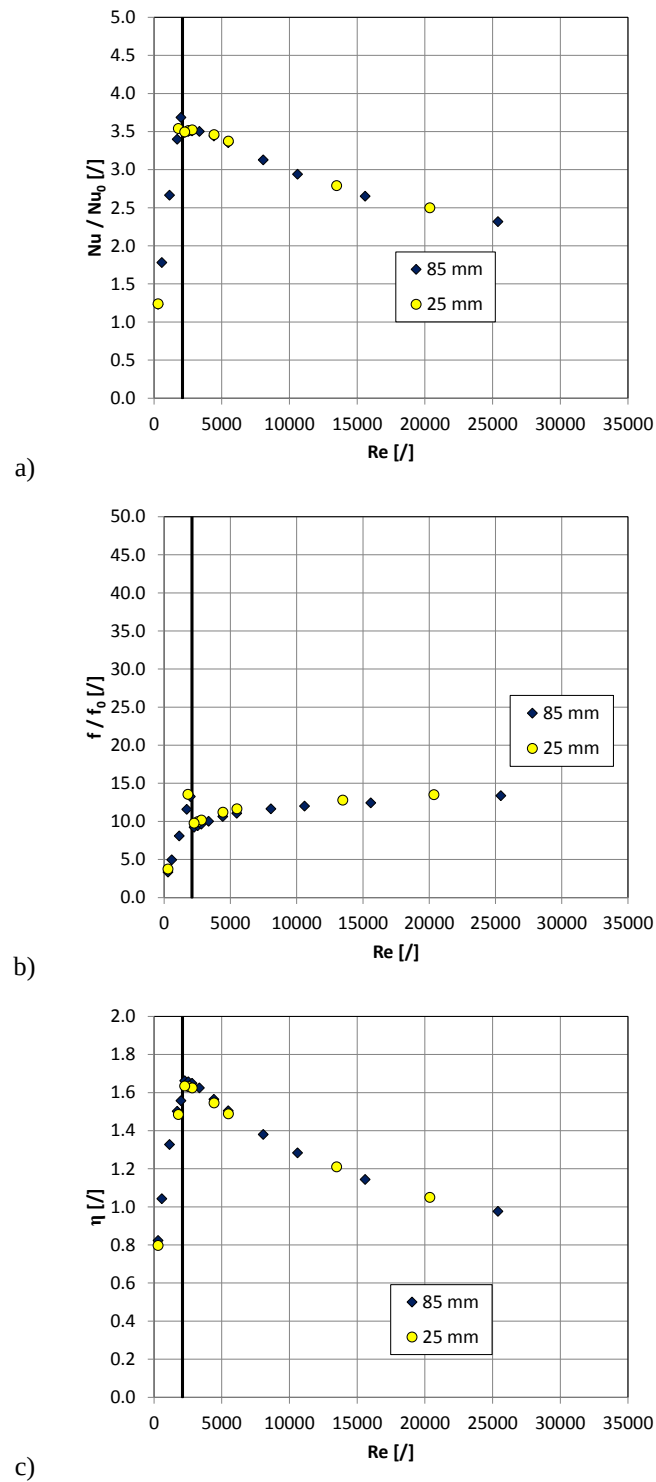


Figure 2.6.9. Comparison between the results of simulations n.4 ($z = 85$ mm) and n.5 ($z = 25$ mm) in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). Simulations refer to the double wave turbulator inside the 42.5 mm tube.

Trends are those deducted after analysis of simulations n.1 and n.2. No significant influence of the position of the double wave turbulator inside the tube has been found.

2.6.2 Validation of the results through the heat generator model

The simulations presented in Section 2.6.1 have been preliminary validated by introducing in the heat generator model presented in Chapter 1 of the present Thesis the equations for the enhancement of the Nusselt number due to the inserts presence, comparing the results of the Simulink simulations with the experimental data.

Model validation when considering turbulators presence has been presented in Section 1.4, confirming the accuracy of the equations obtained through the CFD analysis for the ten wave turbulator placed 85 mm apart the entrance of the 36.4 mm tube. From this results it is inferred that the simulation model is accurate and can be used also for the other configurations.

2.7 Modeling of a new geometry

In this Section, a modified geometry for the double wave turbulator is presented, with the aim of increasing the insert performances maintaining a configuration which is simple and economic to manufacture (thus that can be manufactured by pressing methods). After the description of the new shape the CFD simulations performed with the modified geometry is introduced. Finally, simulations results are discussed.

2.7.1 Description of the geometry

Figure 2.7.1.1 shows a frontal sketch of the actual double wave turbulator.

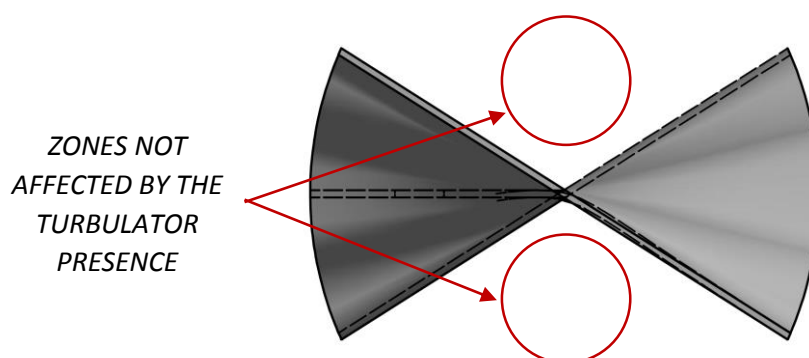


Figure 2.7.1.1. Frontal view of the actual double wave turbulator. Zones not affected by the turbulator presence are highlighted.

The Figure shows that the insert is characterized by two central zones not affected by the turbulator presence. By analyzing the turbulators geometries presented in the literature (Section

2.2) louvered strips turbulators turned out to be feasible solutions able to give good results maintain manufacture simplicity.

Thus, it has been decided to combine the double wave shape with the louvered strip concept found in the literature. The modified double wave turbulator geometry is presented in Figure 2.7.1.2 and 2.7.1.3.

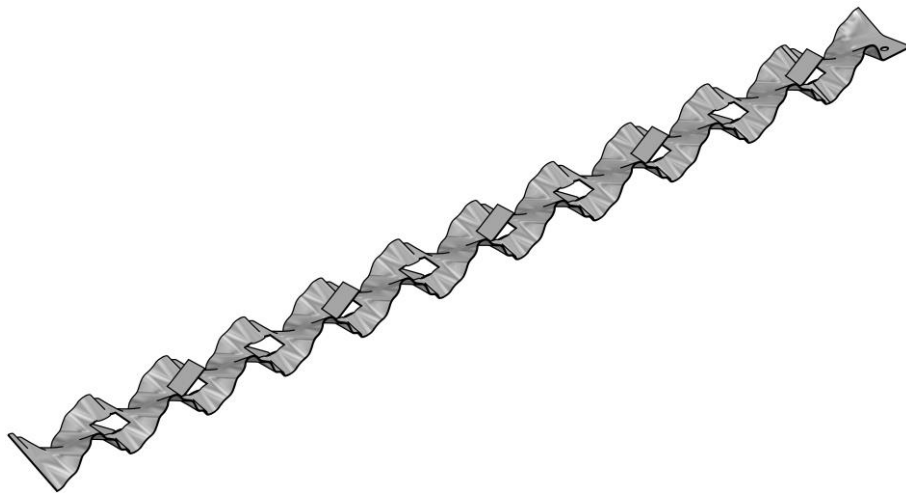


Figure 2.7.1.2. Sketch of the modified double wave turbulator.

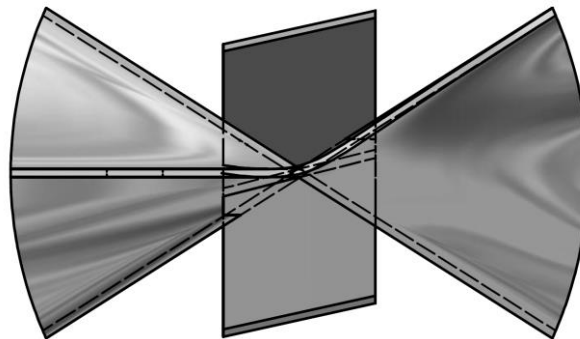


Figure 2.7.1.3. Frontal view of the modified double wave turbulator.

This kind of geometry can still be realized through a pressing process, but it is believed to enhance the double wave performances avoiding the two zones not affected by the turbulator presence.

The turbulator length, width and distance between the wave peaks is not affected by the geometry modification. Louvered strips have an inclination angle equal to 30° and a length of 20 mm. The modified insert is characterized by 10 strips 40 mm spaced.

2.7.2 CFD model

The modified geometry needs to be meshed carefully close to the strips to perform precise CFD simulations of the insert behavior. Thus, because of the extremely high number of mesh elements needed by this kind of geometry it was not possible to solve it as it is.

Nevertheless, with ANSYS/FLUENT it is possible to introduce periodical boundary conditions, so it is possible to investigate the behavior of a part of the turbulator exposed to flue gases flowing in fully developed flow conditions. Thus, it has been decided to perform periodical simulations over a part (120 mm long) of the turbulator, this way reducing mesh size maintaining simulation accuracy.

These simulations are not representative of practical cases, because it is supposed that the velocity profile along all the turbulator is always fully developed and that the turbulator covers all the tube length. Because of this, a direct comparison between the periodic simulations performed with the new geometry and the non-periodic ones performed with the standard shape (presented in Section 2.6) is not correct. Thus, periodical simulations were also performed with the standard double wave turbulator for comparison purposes.

Figure 2.7.2.1 shows the periodical modified geometry simulated with ANSYS/FLUENT. An identical configuration was used for the classical double wave insert.

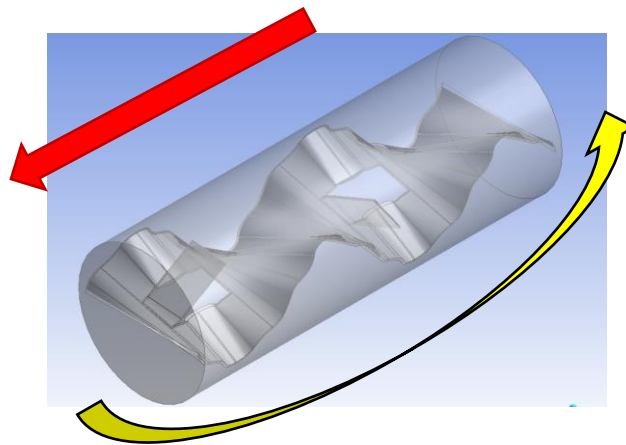


Figure 2.7.2.1. Modified double wave geometry used for periodical ANSYS/FLUENT simulations. Flue gases flow direction (in countercurrent) and velocity profile connection are shown by the red and yellow arrow, respectively.

Both periodical geometries were meshed into $\approx 10 \cdot 10^6$ elements. Simulation time was about 2 – 3 days for each test condition. Again, SST $k - \omega$ turbulence model with low Re correction was used. Periodical velocity boundary conditions are imposed at the inlet and at the outlet of the tube, which is supposed to be composed of seven periodical elements. Bulk temperature of the flue gases at the inlet is equal to 600 K. Walls temperature is equal to 85.5°C while flue gases mass flow rate m_{fg} is varied from 0.0005 kg s⁻¹ to 0.01 kg s⁻¹ to vary flue gases Reynolds number.

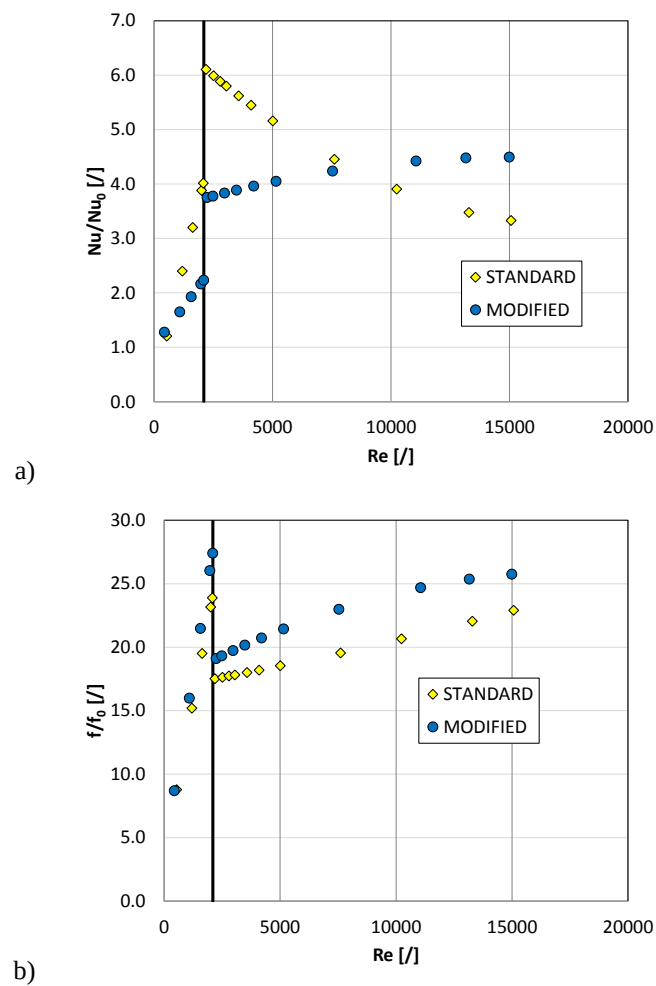
Data reduction is presented in Section 2.5. Flue gases velocity, used in Eqs. (2.6) and (2.7) is evaluated as

$$u_{fg} = \frac{m_{fg}}{\rho_{fg} S} \quad (2.26)$$

Periodical simulations output are the temperature of the flue gases at the exit, the exchanged heat flow rate and the pressure losses between the inlet and the outlet.

2.7.3 Results

Figure 2.7.3.1 shows the results of the periodic simulations performed with the standard and the modified double wave turbulator.



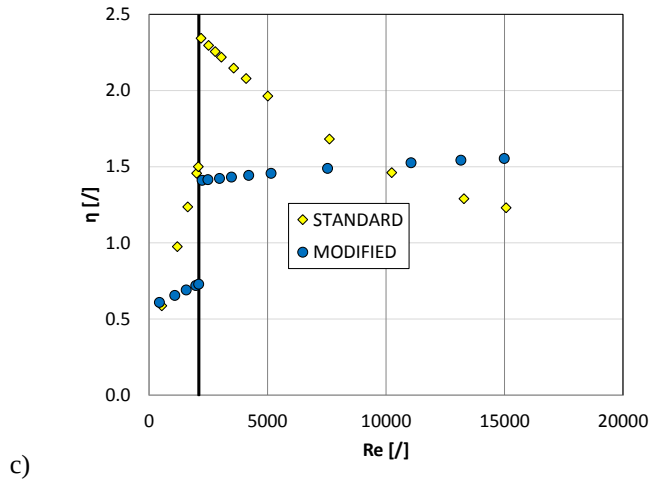


Figure 2.7.3.1. Comparison between the results of the periodic simulations performed with the standard and the modified double wave insert in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c).

The simulations showed that the modified geometry performs better than the new one for very low Reynolds number (less than 600) as well as for fully developed turbulent regime ($Re > 10000$), while for $600 < Re < 10000$ the standard geometry gives better results. High Reynolds numbers (bigger than 10000) are achieved when using large-scale heat generators having high thermal power at the burner (e.g. 2.5 MW heat generator), or when using medium-scale heat generators with a limited number of pipes in the last pass (e.g. 600 kW heat generator with 50 tubes in the last pass, as shown in Section 1.5).

Finally, the behavior of the modified geometry when reversing the flue gases flow direction has been simulated. The following Figure reports a sketch of this periodic simulation.

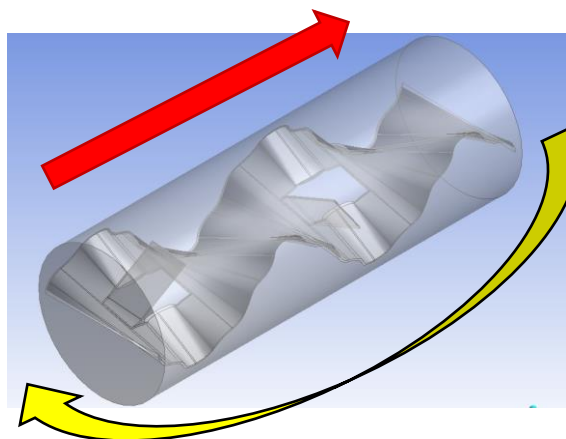


Figure 2.7.3.2. Modified double wave geometry used for periodical ANSYS/FLUENT simulations with opposite flue gases flow direction (co-current flow direction). Flue gases flow direction and velocity profile connection are shown by the red and yellow arrow, respectively.

Comparison between the results obtained with the two flowing directions are reported hereinafter.

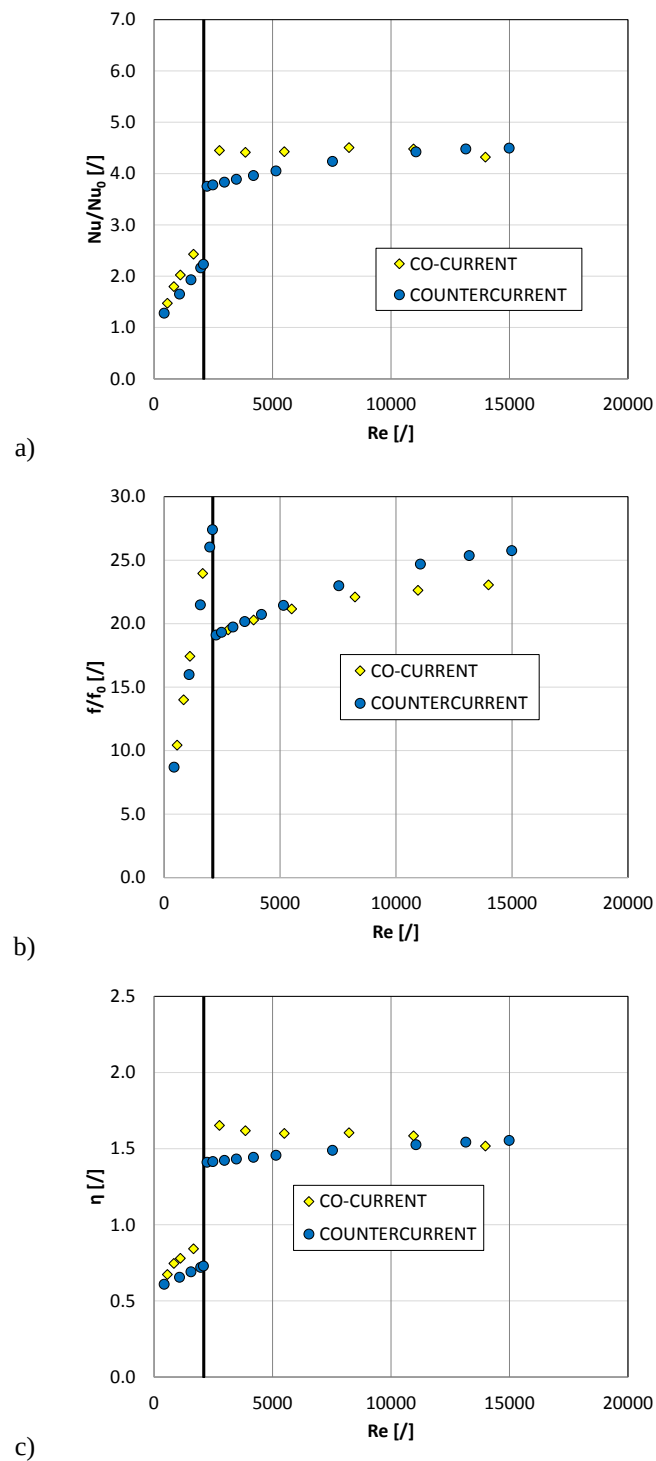


Figure 2.7.3.3. Comparison between the results of the periodic simulations performed with the modified double wave insert in terms of Nusselt number enhancement (a), friction factor enhancement (b) and thermal performance ratio (c). Results refer to co-current and countercurrent flow directions.

It can be seen that by reversing the turbulator inside the tube better results are achieved, for Reynolds numbers lower than ≈ 13000 . The performance enhancement is mainly due to better thermal performances of the co-current geometry, while in terms of frictional losses the two shapes perform similarly.

By comparing results in Figures 2.7.3.1 and 2.7.3.3 it is possible to see that the modified co-current solution performs better than the standard one for $Re > 8000$, thus for a larger Reynolds number range in comparison to the countercurrent situation.

2.8 Conclusions

In this Chapter, CFD analysis of heat transfer and pressure losses due to the introduction of turbulence generators inside a tube where flue gases flow has been presented.

Twisted tape turbulators are widely studied in the literature because they provide good performance, but such a geometry is quite complicated and expensive to manufacture, thus it is not applied in the heat generator industry. On the contrary, inserts which can be manufactured through a pressing process, such as the single wave and the double wave geometries presented in Section 2.3, are actually used in heat generator systems. Thus, heat transfer and pressure losses of a tube equipped with these two geometries have been simulated by means of ANSYS/FLUENT to evaluate the turbulators performance. Simulations were carried out varying the turbulators position inside the tube as well as the pipe diameter.

Results showed that the use of turbulator enhances the convective heat transfer coefficient inside the tube, but it also increases frictional pressure losses. To evaluate the global performance of the fitted tube, the thermal performance ratio (η), which accounts for heat transfer enhancement and the increased pumping power required because of increased pressure losses, has been introduced and calculated.

When inserted in a small diameter tube ($D = 36.4$ mm) the single wave turbulator presents $\eta > 1$ only if flue gases flow with $\approx 1500 < Re < \approx 20000$, while outside this range the negative effect of the enhanced frictional losses is higher than the positive effect of increased heat transfer. When placing the turbulator inside a bigger diameter tube ($D = 42.5$ mm) it has been found that the reduced frictional losses lead to a wider range of operating conditions with $\eta > 1$ ($\approx 1400 < Re < \approx 30000$). No significant influence of the turbulator position inside the tube was found.

When comparing single wave and double wave turbulators, it is shown that the latter presents higher thermal performance ratio in laminar and transient flow regimes, because of the reduced

frictional losses. Instead, in fully developed turbulent regime, the single wave insert performs better, because of the higher Nusselt number enhancement.

Since these simulations are representative of real conditions inside fire tube heat generators, equations to predict turbulators performance have been deducted.

Finally, the double wave geometry was modified inserting central jags to better influence the flue gases flowing in the central part of the tube. A portion of the fitted tube has been studied through CFD periodical simulations finding that the jagged double wave turbulator performs better than the standard one for fully developed turbulent flue gases flow regime, while for $Re < 8000$ the previous geometry is better. Thus, the modified insert can be effectively used in large scale (e.g. 2.5 MW) heat generators or in medium scale (e.g. 600 kW) heat generators with a limited number of tubes in the last fitted pass, while for small scale (e.g. 90 kW) heat generators the actual geometry should be used.

Nomenclature

A = Area, m^2

c_p = Isobaric specific heat capacity, $\text{J kg}^{-1} \text{K}^{-1}$

D = Diameter, m

f = Friction factor, /

L = Length, m

m = Mass flow rate, kg s^{-1}

Nu = Nusselt number, /

p = Pressure, bar

Pr = Prandtl number, /

q = Heat flow rate, W

Re = Reynolds number, /

S = Cross section, m^2

T = Temperature, K

u = Velocity, m s^{-1}

V = Volumetric flow rate, $\text{m}^3 \text{s}^{-1}$

z = Distance between the tube entrance and the beginning of the turbulator, m

Greek Symbols

α = Heat transfer coefficient, $\text{W m}^{-2} \text{K}^{-1}$

Δp = Pressure drop, Pa

ΔT_{ml} = Mean logarithmic temperature difference, K

η = Thermal performance ratio, /

λ = Thermal conductivity, $\text{W m}^{-1} \text{K}^{-1}$

μ = Dynamic viscosity, Pa s

ρ = Density, kg m^{-3}

Subscripts

0 = Plain tube

fg = Flue gases

IN = Inlet

OUT = Outlet

PP = Pumping power

TURB = Turbulator

3 TWO-PHASE HEAT TRANSFER ENHANCEMENT BY MEANS OF NANO-ENGINEERED SURFACES

Condensing heat generators are of great interest for the industry because of the possibility of achieving extremely high efficiency through the recovery of the latent heat in the exhaust products. Two-phase heat transfer in heat generators is thus an interesting field of research.

When steam condenses over a surface it usually forms a liquid layer which introduces a mayor heat transfer resistance to the two-phase process. Reducing or eventually avoiding the condensate layer formation will thus lead to an enhancement of the condensation heat transfer coefficient, increasing the heat transfer rate per unit area. This way, it will be possible to get higher performance of the system maintaining the same geometry, or it will be possible to reduce the heat transfer area (thus the system costs) maintaining the same net effect.

Because of this, the following Chapter will focus on the enhancement of the condensation heat transfer coefficient by means of nano-engineered surfaces, with the aim of removing (dropwise condensation mode) or eventually reducing (filmwise condensation with reduced thickness) the condensate layer over the wall. For this purpose, surface wetting properties of aluminum substrates (a material which is of great interest for the heat transfer industry) have been modified for studying the behavior of conventional, superhydrophilic, hydrophobic and superhydrophobic surfaces during pure steam condensation at various vapor velocity. For the sake of experimental simplicity, in the following investigation pure steam is condensed over the surface, without presence of non-condensable gases.

3.1 Introduction

Owing to their high water repellency, hydrophobic and superhydrophobic surfaces have recently been studied as a promising solution to several challenges, such as drag reduction, anti-icing and enhancement of two-phase heat transfer performance (Rothstein, 2010; Antonini *et al.*, 2011; Betz *et al.*, 2013). Among the others, promotion of dropwise condensation mode over filmwise one is a very interesting research field.

Condensation is widely used in many industries and engineering processes, such as power industry and refrigerating process. With the motivation for improved energy efficiency and miniaturization of the heat exchanger, an extensive research has been undertaken in the area of enhanced

condensation heat transfer since the report of Schmidt *et al.* (1930), who firstly recognized that during dropwise condensation the heat transfer coefficients were between 5 and 7 times those found with film condensation.

The ability of dropwise condensation to increase heat transfer coefficients over filmwise condensation is well established. The main mechanism responsible for the heat transfer enhancement in dropwise condensation heat transfer is related to droplets mobility. In fact, the sweeping and renewal in the droplets' growth process, encouraged by poor wall wettability, produces an augmentation of both the heat and the mass transfer coefficient as compared to the film condensation condition, where the liquid layer adjacent to the wall causes a large thermal resistance.

The surface properties, in particular the surface energy of the material and the correlated contact angle phenomena, play a crucial role in determining the onset of condensation modality. Typically, hydrophobic surfaces are expected to promote dropwise condensation, while hydrophilic ones more easily induce filmwise condensation.

Surface wettability is defined by the contact angles of a water drop sitting over it. For a static drop equilibrium contact angle is taken into account while for moving drops the advancing and receding contact angles are taken as a reference. The difference among the last two gives the contact angle hysteresis. Superhydrophobic surfaces present high contact angles, greater than 150° , and low contact angle hysteresis, lower than 10° . These surfaces can be produced by combining two factors: micro-/nano- scale surface roughness and low surface free energy. Proper surface roughness can be obtained through different techniques, as micromachining, micro-contact-printing, chemical etching in aqueous solutions and deep radiative ion etching. Low surface energy can be obtained by coating the substrate with a thin layer of a material with small surface energy, such as organic substances, polymers, and noble metals. These two elements allow water drops to sit over the surface with a quasi-spherical shape and to easily roll-off from it, being this is a key factor for two-phase heat transfer applications. When modifying surface morphology without lowering its free energy the substrate presents very low contact angles (i.e. lower than 30°), thus it is named superhydrophilic. If the contact angle is not that low, but still lower than 90° , the surface is called hydrophilic. On the contrary, hydrophobic surfaces, obtained only by lowering surface free energy, present an advancing contact angle in between 90° and 150° and a contact angle hysteresis bigger than 10° .

The use of organic substances as low-surface energy promoters requires strong, long-term adhesion forces between the coating and the metal substrate. Usually, the thicker is the coating, the better its resistance to corrosion/erosion. However, due to very low thermal conductivity,

thick organic coatings add a heat transfer resistance that deteriorates the condensation performance. Moreover, the coating material, if inadvertently removed from the condenser surface, may contaminate the system. A very interesting method of modifying the wetting properties of a surface is to use Self Assembled Monolayers (SAMs). A SAM is composed of a single layer of organic molecules adsorbed onto a surface to form a coating. Being only a monolayer-thick (10–15 Å), these coatings give negligible heat transfer resistance. In addition, they are very stable and should last over long periods of time (Bonner, 2010).

Nanoengineered surfaces appear to be a viable and promising solution to enhance condensation performance but their feasibility is still to be demonstrated. The present study is focused on aluminum surfaces which have many applications in industrial fields, especially in the heat exchange area.

3.2 Theoretical background

In this Section, the theoretical analysis of the wettability models and of the dropwise condensation phenomena are discussed. First, the Young, Wenzel, and Cassie equations of wetting properties on smooth and rough surfaces are introduced. Then, the theory of contact angles analysis is presented, with particular care to theoretical evaluation of the contact angle hysteresis as a measure of energy dissipation during liquid flow transition between the wetting regimes. Finally, the base theory of the dropwise condensation phenomena is introduced.

3.2.1 Wetting regimes over smooth and rough surfaces

From the thermodynamic point of view, wetting of a rough solid surface is governed by Young, Wenzel, and Cassie equations that relate the contact angle between liquid and solid to interface free energies and to surface roughness.

Figure 3.2.1.1 reports a liquid in contact with a solid in presence of air, with the free surface energies of the solid–liquid, solid–air, and liquid–air interfaces equal to γ_{SL} , γ_{SA} , and γ_{LA} , respectively. When a liquid front comes in contact with a flat solid surface under the contact angle ϑ , the propagation of the liquid front for a small distance ds results in a net energy change of

$$dE = ds \cdot (\gamma_{SL} - \gamma_{SA} + \gamma_{LA} \cdot \cos \vartheta) \quad (3.1)$$

Therefore, for the liquid front to be at equilibrium ($dE/ds=0$), the Young equation should be satisfied:

$$\gamma_{LA} \cdot \cos \vartheta = \gamma_{SA} - \gamma_{SL} \quad (3.2)$$

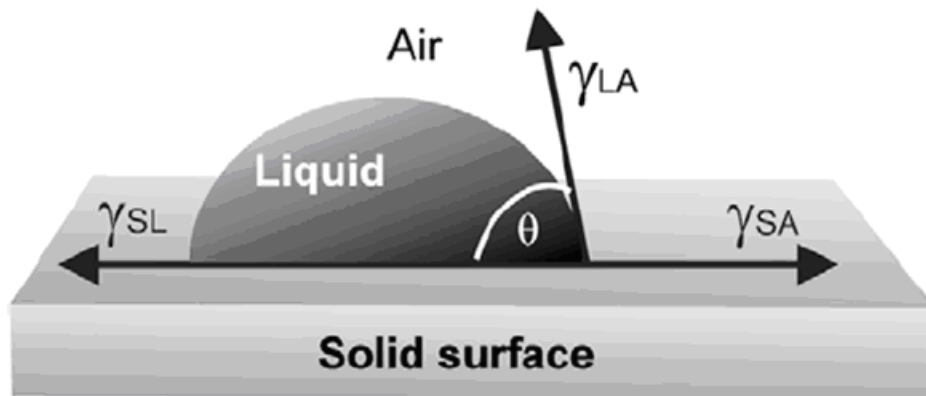


Figure 3.2.1.1. Liquid drop over a solid surface. ϑ represents the contact angle, while γ are the interfacial surfaces energies.

The Young's equation allows the calculation of the contact angle for given values of the interface energies. It is obvious from Eq. (3.2) that three situations are possible:

- If $(\gamma_{SA} - \gamma_{SL})/\gamma_{LA} = 1$, complete wetting takes place with the liquid fully adsorbed by the solid surface ($\vartheta = 0^\circ$).
- If $(\gamma_{SA} - \gamma_{SL})/\gamma_{LA} = -1$, complete rejection of the liquid by the solid surface takes place ($\vartheta = 180^\circ$).
- Finally, if $-1 < (\gamma_{SA} - \gamma_{SL})/\gamma_{LA} < 1$ (most common situation) the intermediate situation of partial wetting takes place ($0 < \vartheta < 180^\circ$).

A liquid that has the contact angle $\vartheta < 90^\circ$ is often referred to as a “wetting liquid,” while that with $\vartheta > 90^\circ$ is a “non-wetting liquid”. Corresponding surfaces are called, in the case of the contact with water, “hydrophilic” and “hydrophobic”, respectively.

The free interface energies can also be viewed as surface tension forces. These forces are applied to the three-phase contact line (the triple line) and directed toward the corresponding interface. The surface tensions are measured in force units per length of the contact line, N m^{-1} , the same units as the interface energy, J m^{-2} . Note that Young's formulation implies that the solid is non-deformable and insoluble, so that only the horizontal projection of the tensions is considered.

Young's equation does not take into account a number of factors, which can significantly affect the contact angle at the micro and nanoscale. It is emphasized that the contact angle provided by the Young's equation is a macroscale parameter, so it is sometimes called the “apparent contact

angle” (APCA). The actual angle under which the liquid–air interface comes in contact with the solid surface at the micro and nanoscale can be different. There are several reasons for that.

First, water molecules tend to form a thin layer upon the surfaces of many materials. This is because of a long-distance Van Der Waals adhesion force that creates the so-called disjoining pressure. This pressure is dependent upon the liquid layer thickness and may lead to the formation of stable thin films or precursors. In this case, the shape of the drop near the triple line gradually transforms from a spherical surface into a precursor layer, and thus the nanoscale contact angle is much smaller than the apparent contact angle.

Second, even carefully prepared atomically smooth surfaces exhibit certain roughness and chemical heterogeneity. Water tends to cover at first the hydrophilic spots with high surface energy and low contact angle. The tilt angle of the surface due to roughness can also contribute to the APCA.

Third, Young’s equation provides the value of the so called static contact angle that ignores any dynamic effects related to the change of the drop’s shape. The very concept of the static contact angle is not well defined. For practical purposes, the contact angle, which is formed after a drop is gently placed upon a surface and stops propagating, is considered the static contact angle. However, depositing the drop involves adding liquid while leaving, which may involve evaporation, so it is difficult to avoid dynamic effects.

Fourth, for a small drop and curved triple lines, the effect of the contact line tension may be significant. Molecules at the surface of a liquid or solid phase have higher energy because they are bonded to fewer molecules than those in the bulk. This leads to surface tension and surface energy. In a similar manner, molecules at the edge have fewer bonds than those at the surface, which leads to line tension and the curvature dependence of the surface energy. This effect becomes important when the radius of curvature is comparable with the Tolman’s length. However, the triple line at the nanoscale can be bending, and the radius of curvature can be very small, so that the line tension effects become important.

Thus, while the contact angle is a convenient macroscale parameter, wetting is governed by interactions at the micro and nanoscale, which determine the contact angle hysteresis and other wetting properties.

The Wenzel equation relates the contact angle of a water drop upon a rough solid surface ϑ , with that upon a smooth surface, ϑ_0 , through the nondimensional surface roughness factor, $Rf \geq 1$, equal to the ratio of the surface area to its flat projection (Figure 3.2.1.2):

$$\cos \vartheta = Rf \cdot \cos \vartheta_0 \quad (3.3)$$

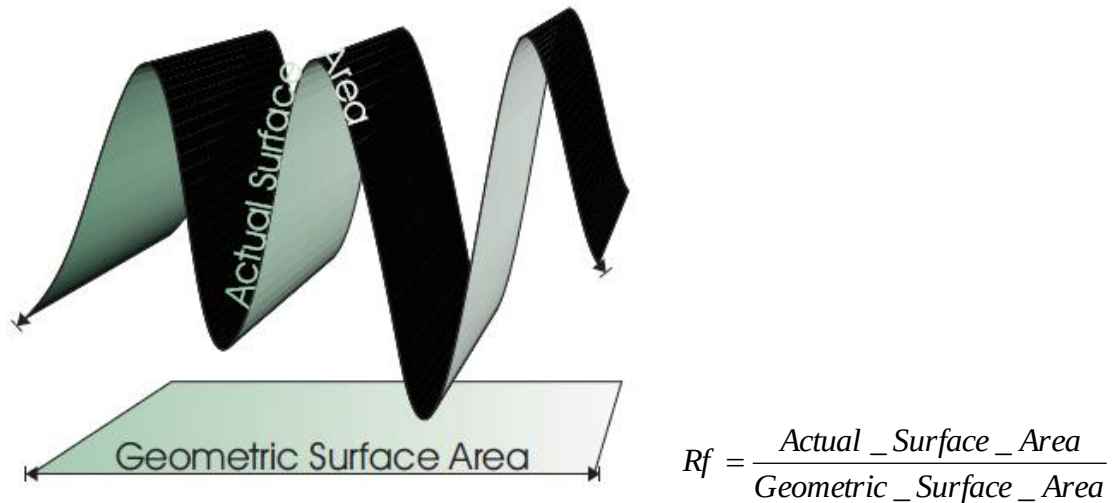


Figure 3.2.1.2. Schematic drawings of the actual and projected area of a rough surface.

The equation was originally derived for the homogeneous solid–liquid interface (no air pockets, Figure 3.2.1.3) using the surface force balance and empirical considerations; however, it was later put in a proper thermodynamic framework. It is important that according to Wenzel model the inherently hydrophilic flat surface will be more hydrophilic when rough, and inherently hydrophobic surface will become more hydrophobic.

For a surface composed of two fractions, one is with the fractional area f_1 and the contact angle ϑ_1 and the other with f_2 and ϑ_2 , respectively $f_1+f_2=1$, the contact angle is given by the Cassie equation:

$$\cos \vartheta = f_1 \cdot \cos \vartheta_1 + f_2 \cdot \cos \vartheta_2 \quad (3.4)$$

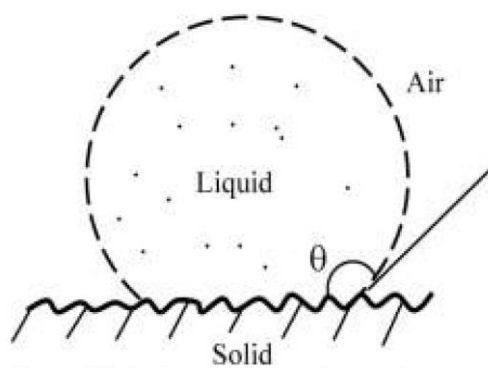


Figure 3.2.1.3. Water drop on a rough substrate.

For the case of a composite interface, consisting of the solid–liquid fraction ($f_1=f_{SL}$, $\vartheta_1=\vartheta_0$) and liquid–vapor fraction ($f_2=1-f_{SL}$, $\cos \vartheta_2=-1$), substituting in the Eq. (3.4) yields the Cassie–Baxter equation:

$$\cos \vartheta = -1 + f_{SL} \cdot (\cos \vartheta_0 + 1) \quad (3.5)$$

Eq. (3.5) predicts that an enhancement of hydrophobicity ($\vartheta > \vartheta_0$) and a jump in the contact angle can often be observed once air trapping occurs.

The same equation could be explained in this way: a drop on a flat solid makes an angle ϑ , while it does not spread at all on a pure film of air (contact angle of π), and the average value it takes on the patchwork is an average on the cosines weighted by the respective proportions of solid and air below the drop f_{SL} .

Gao and McCarthy (2007) showed experimentally that the contact angle of a drop is defined by the triple line and does not depend upon the roughness under the bulk of the drop. The authors concluded that the Wenzel and Cassie equations “should be used with the knowledge of their fault.” The question remained, however, under what circumstances the Wenzel and Cassie equations can be safely used and under what circumstances do they become irrelevant.

Figure 3.2.1.4 summarizes the wetting regimes over a rough surface.

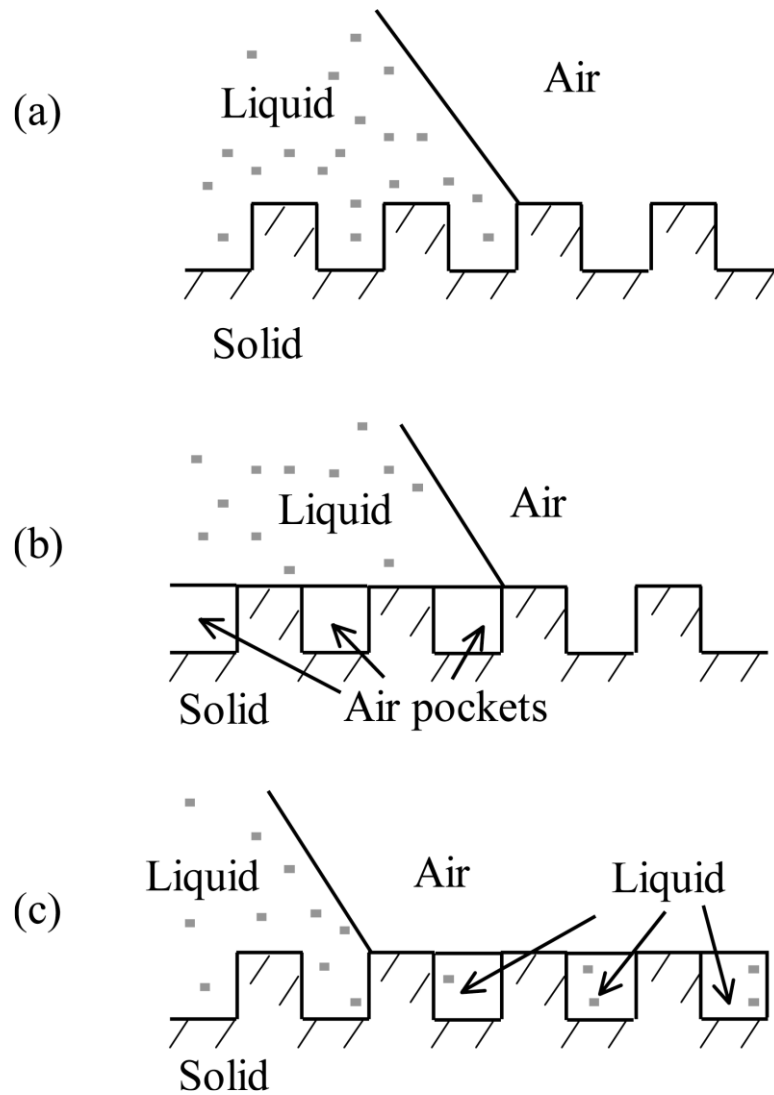


Figure 3.2.1.4. Solid–liquid–air interface (a) homogeneous (Wenzel), (b) composite (Cassie), and (c) with filled holes (Cassie with water penetration).

3.2.2 Contact angles theory

Surface wettability is defined by the contact angles of a water drop sitting over it. For a static drop equilibrium contact angle is taken into account while for moving drops the advancing and receding contact angles are taken as a reference. The dynamic condition is much more interesting for practical applications, thus in this Thesis only advancing and receding angles will be considered. Figure 3.2.2.1 shows the dynamic angles of a drop moving onto a tilted surface.

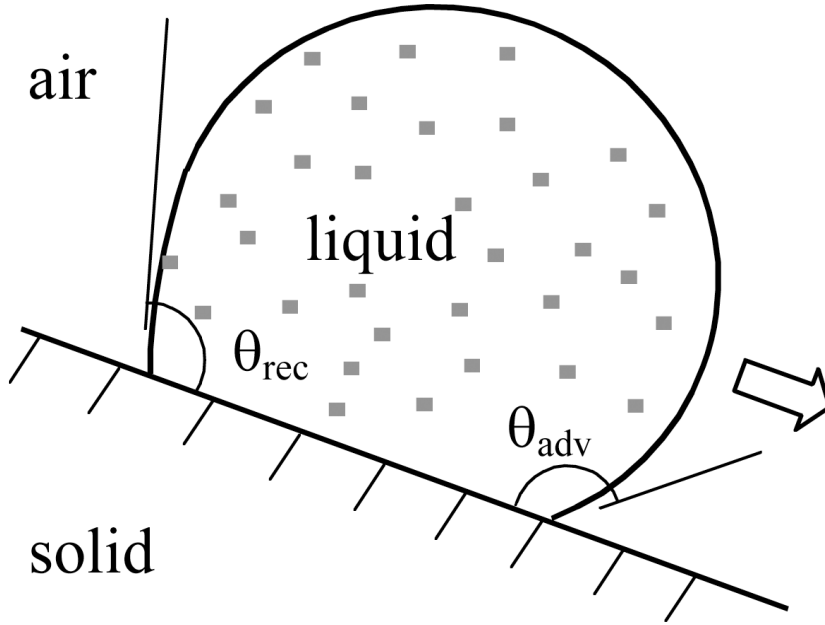


Figure 3.2.2.1. A water drop on a tilted solid surface. The contact angle at the front is the advancing contact angle, ϑ_{adv} , whereas that at the back is the receding contact angle, ϑ_{rec} .

The difference among advancing and receding angles gives the contact angle hysteresis. Superhydrophobic surfaces present high advancing contact angle, greater than 150° , and low contact angle hysteresis, lower than 10° .

Contact angles hysteresis plays a crucial role in determining the mobility of a drop over a surface, thus in influencing the condensation mode.

It has been pointed out that defects on a solid can pin a contact line. As a consequence, droplets on an incline stay at rest; the front and rear contact nonwetting and wetting defects, respectively. The resulting asymmetry in contact angles creates a Laplace pressure difference between the front (of high curvature) and the rear (of smaller curvature) and, thus, a force able to resist gravity provided that the drop is small enough. Both chemical heterogeneities and roughness can act as pinning sites. It has been shown that a force, most often gravity, must be applied to overcome the surface tension forces holding the liquid to the surface. The following equation

$$m \cdot g \cdot \sin \alpha = k \cdot w \cdot \gamma_{LV} \cdot (\cos \vartheta_{Rec} - \cos \vartheta_{Adv}) \quad (3.6)$$

(where m is the droplet mass, g is the acceleration of gravity, k is a constant, w is the droplet contact diameter, ϑ_{rec} and ϑ_{adv} are the receding and advancing contact angles respectively and γ_{LV} is the liquid vapor surface tension) can be used to predict α , the minimum tilt angle required for the droplet to move. This equation predicts that by decreasing contact angle hysteresis, i.e. the difference between the advancing and receding contact angles, the inclination angle required to induce drop motions is decreased. In fact, if there is no hysteresis, then a droplet of liquid should

move spontaneously on a horizontal surface. However, if the hysteresis is high enough, then a droplet will remain pinned on a vertical surface even though the static contact angle is greater than 160° . Contact angle hysteresis on hydrophobic rough surfaces can vary dramatically depending on how the liquid wets the surface. Wenzel described a wetting regime where the liquid penetrates between the surface asperities on a rough surface. It was later shown that such surfaces often have high contact angle hysteresis: the three-phase contact line remains pinned because there are high-energy barriers between metastable states that the contact line can adopt. A liquid droplet can also rest on top of surface features as described by Cassie and Baxter. In this wetting regime the hysteresis is generally lower because the droplet is sitting on a composite surface of air and solid and the energy barriers limiting the discontinuous contact line movement are small. Therefore, according to Eq. (3.6), a droplet would move more easily (at a lower tilt angle) if sitting in the Cassie-Baxter wetting regime than in the Wenzel one.

Recently, it has been shown that a transition from the Cassie- Baxter to Wenzel wetting regime can be observed on some superhydrophobic surfaces. Condensation on superhydrophobic surfaces has also been shown to decrease drop mobility and increase hysteresis. Surfaces with low hysteresis but minimal roughness (i.e., not superhydrophobic) would be more appropriate choices for applications where liquid mobility needs to be maintained during condensation (Wier and McCarthy, 2006).

3.2.3 How to get superhydrophobicity

As previously introduced, superhydrophobicity derives from a combination of two factors:

- Micro-/nano- scale roughness
- Low surface energy (hydrophobic) coating

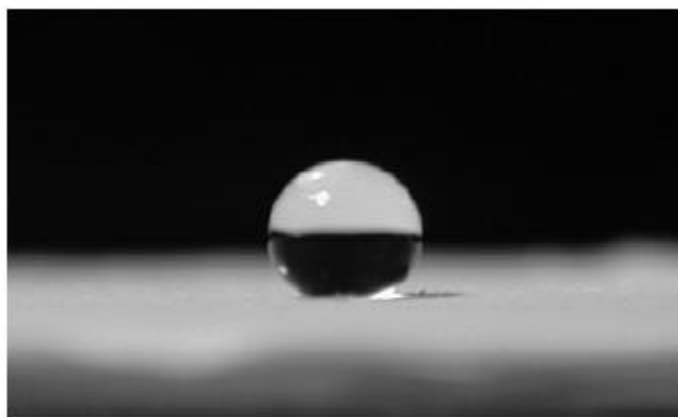


Figure 3.2.3.1. Water drop over a superhydrophobic surface.

A combination of these two factors is necessary, because of the practical difficulty of achieving flat surfaces with large hydrophobicity (Bico *et al.*, 2002).

These two characteristics allow water droplets to roll easily and the phenomenon is called “Lotus effect” which involves two important properties that are typical for many water-repellent plant leaves: the superhydrophobicity and self-cleaning. A surface with the water contact angle θ greater than 150° and with low contact angle hysteresis is called superhydrophobic. Scanning Electron Microscope (SEM) study on the lotus leaf reveals that its surface is covered by “bumps” more exactly called *papillae* (papillose epidermal cells), which, in turn, are covered by an additional layer of epicuticular wax. The wax is hydrophobic with water contact angle of about 95° – 110° , whereas the experimental values of the static water contact angle with the lotus leaf were reported as about 160° . The increase of the contact angle is a result of the surface roughness due to the papillae (Figure 3.2.3.2, left). On the right side of Figure 3.2.3.2 is reported the picture of a water droplet on a lotus leaf. The bottom of the drop (blue-greyish colors) reflects sky, an indication of the additional air layer underneath the drop, causing total reflection of the light at the air-water interface. Thanks to this air layer, the drop is highly mobile on the surface.

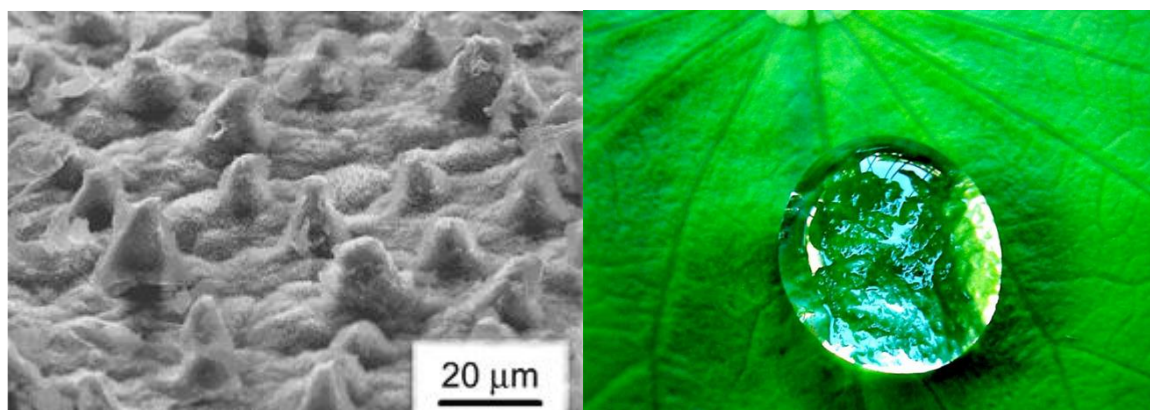


Figure 3.2.3.2. (Left) SEM image of the lotus leaf surface showing papillae. (Right) Water droplet on a lotus leaf.

There are many other examples in nature of superhydrophobic leaves. Neinhuis and Barthlott (1997) reported the study of about 200 plants, which have all three common features:

- They are coated by an epicuticular film of wax that makes them hydrophobic
- They have a textured surface made by bumps at a typical scale of $10\ \mu\text{m}$
- They have also a secondary texture, much smaller in size than the previous one (around $1\ \mu\text{m}$) often superimposed on the first one

Also many animals, like for examples water striders, ducks and butterflies have optimal water repellency characteristics, and their structure confirms that the hydrophobicity of a solid is enhanced by textures (Quéré, 2005). In the recent years, many researchers have become interested in superhydrophobic surfaces because a variety of experimental techniques allow the preparation of samples with controlled roughness. A review of these processes will be presented in Paragraph 3.3.

3.2.4 Dropwise condensation phenomena

The experimental evidence of Umur and Griffith (1965) that the surface area between condensate drops during vapor condensation remains dry, with no indication of a microfilm, opened a new perspective on this phenomenon. Like in the heterogeneous nucleation of vapor bubbles on superheated surfaces in boiling, a cyclic process of droplet nucleation at active sites occurs during DWC. The nucleated droplets grow by mass addition due to condensation, coalescence due to surface tension effects and finally are removed by body force or shear drag. This nucleation model, since its early proposal, have been further developed and confirmed by other authors (Rose, 1981), and is widely accepted.

It has been reported that dropwise condensation on a hydrophobic surface enhances heat transfer by an order of magnitude compared with filmwise condensation (Schmidt *et al.*, 1930). Dropwise condensation occurs on surfaces which are not strongly wetted by the liquid and so the drops do not spread out over the surface. They grow until they became so large that they run off the surface by gravity or are blow off by the flowing vapor. Surfaces with strong hydrophobicity are believed to allow effective dropwise condensation, due to the promotion of the drop movement and departure and consequently accelerating surface renewal, allowing small drops to form.

As already said, the main mechanism responsible for the heat transfer enhancement in dropwise condensation heat transfer is related to droplet mobility. Then the question that arises is how to evaluate if a surface is suitable for promoting the condensation in the dropwise mode.

Contact angle has sometimes been used to predetermine the condensation mode on a specific solid surface. However, the contact angle measured at room temperature and in equilibrium with an air environment has been proven not to be useful for determining the wettability of systems where mass transfer takes place. For example, the contact angle of water on a polytetrafluoroethylene (PTFE) surface is 88° under the condensation condition at atmospheric pressure but 108° at room temperature with an air environment (Ponter, 1992). Condensation on superhydrophobic surfaces has been shown to decrease drop mobility and increase hysteresis.

Quere *et al.* (2003) reported experiments of water condensation where the contact angle hysteresis was nearly 110° , compared to about 5° on the dry surface. Also some tests of condensation on a lotus leaf showed an increase in the contact angle hysteresis (Narhe and Beyesen, 2004). Wier and McCarthy (2006) observed the condensation of water on ultrahydrophobic surfaces containing hydrophobized silicon pillars. They found that water condenses both between and on the top of the surface features: a macroscopic water drop coalesces with the water and undergoes a wetting transition from sitting on the top of the posts (Cassie-Baxter State) to wetting between them (Wenzel State). That causes an increase in contact angle hysteresis and a decrease in macroscopic water drop mobility. They concluded that surfaces with low hysteresis but minimal roughness would be more appropriate choices for applications where liquid droplet mobility needs to be maintained during condensation.

Out of Wenzel and Cassie-Baxter states the one with lowest energy should prevail. In practice, however, the interface is not always in the energetically most favorable state. Very often a composite state is observed where a noncomposite state would be favorable. This state is metastable, and applying pressure to the drop causes a transition of the interface to a noncomposite state (Quere *et al.*, 2003). It is therefore possible that both states coexist on the same substrate for different drops. A pressure applied to the drop of liquid can cause an irreversible transition to the Wenzel's regime, but also the condensation has been shown to decrease drop mobility and increase hysteresis. For moderate roughness and hydrophobicity Wenzel angles are expected to be smaller than Cassie ones. But the most important fact is the increase of the contact angle hysteresis. This can be understood because, in Wenzel state, the droplet is in contact with water trapped inside the texture. The contact angle should be given by an average between ϑ and 0, yielding a very low value for the receding angle. Thus, rather than a small difference in contact angles, the main difference between both superhydrophobic states lies in the adhesion properties: a Wenzel drop will adhere efficiently to its substrate, in spite of high contact angle.

Rausch *et al.* (2010) suggested a condensation model based on droplet nucleation and growth on elevated precipitates, resulting in short-term steam entrapment after droplet coalescence. According to the wetting theory, this transition state corresponds with the contact angle model suggested by Johnson and Dettre (1964) which combines the approaches by Wenzel and Cassie and Baxter: gas or steam entrapment enhances the macroscopic contact angle and thus increases the tendency for stable dropwise condensation. Therefore, the wettability parameters determined at dry conditions are not capable of predicting condensation forms on surfaces suitable for the suggested condensation mechanism.

Lan *et al.* (2010) considered the effect of surface energy and nanostructures on dropwise condensation. Self-assembled monolayers coatings of *n*-octadecyl mercaptan were prepared on a mirror-polished (SAM-2) and a nanostructured (SAM-1) copper substrates to promote the DWC. According to modified Cassie–Baxter model, the SAM-1 surface would show a better performance of superhydrophobicity and a larger apparent contact angle in the air-steam environment. However the DWC heat transfer coefficients for SAM-1 surface are lower than those of SAM-2, because DWC takes place on a composite surface, comprising SAM-1 surface and condensate.

According to Ma *et al.* (2010), in a pure steam condensation process, usually the condensate filled the cavities of roughness induced coated surface without air. During the condensation of pure steam, the micro-nanostructures, which are important for superhydrophobic surfaces, would not adsorb and trap air to show superhydrophobicity. The condensate droplets are in a new condensate wetting mode with condensate stagnating in the cavities of the micro-nanostructures, resulting in an increase in the thermal resistance and a decrease in heat transfer performance. This is in contrast with the model of Rausch *et al.* (2010) who started from the hypothesis that the initial droplet formation takes place on elevated sites, resulting in short-term steam entrapment after droplet coalescence.

Chen *et al.* (2007) have recently reported continuous dropwise condensation on a superhydrophobic surface with short carbon nanotubes deposited on micromachined posts, a two-tier texture mimicking lotus leaves. Superhydrophobicity is retained during and after condensation and rapid drop removal is enabled with a hexadecanethiol coating.

Much work is needed to quantitatively understand the dynamic condensation process down to the nanoscale. Boreyko and Chen (2009) reported continuous dropwise condensation spontaneously occurring on a superhydrophobic surface without any external force. The spontaneous drop removal results from the surface energy released upon drop coalescence, which leads to surprising out-of-plane jumping motion of the coalesced drops at a speed as high as 1 m s^{-1} . They prepared a superhydrophobic substrate composed of two tier roughness with carbon nanotubes deposited on silicon micropillars and coated with hexadecanethiol. As a control case they also prepared a smooth silicon substrate coated with hexadecanethiol. During the condensation tests, they saw conventional dropwise condensation with a continuous grow of the droplets on the smooth hydrophobic sample, while, in the superhydrophobic sample, the drops were autonomously removed. This is the first example of sustained Cassie state reported during condensation.

Superhydrophobic surfaces have been considered suitable DWC promoters because such surfaces are believed to bead up water drops and to let them to roll off easily. But what is the best

configuration for promoting stable DWC? Patankar (2010) presented an analysis of the thermodynamic of phase change and concluded that the best surface should be made by long slender pillars with hydrophobic sides and hydrophilic tops. In this way, the hydrophilicity in the top of the pillars promotes drops nucleation there, avoiding the condensation in the grooves or cavities that can lead to drops in the Wenzel state which do not slide off easily.

Varanasi *et al.* (2009) prepared a textured surface consisting of an array of hydrophobic posts with hydrophilic tops. They demonstrated that nucleation and subsequent growth of droplets occurs preferentially on the hydrophilic post tops of the hybrid surface when compared to the random nucleation of droplets on the identically textured superhydrophobic surface with uniform wettability. The condensation experiments of Varanasi *et al.* (2009) have been done on a horizontal surface: no external forces were present for removing the condensed droplets from the surface.

These works are a guideline in the realization of new surfaces for promoting stable dropwise condensation. However, if such principles have to be transferred in the realization of superhydrophobic surfaces for the promoting of dropwise condensation in the heat exchangers industry, some easy to do and costless treatments should be found.

3.3 State of the art about superhydrophobic aluminum surfaces

In this Section a short review of the most relevant works about the techniques that can be used to obtain superhydrophobicity over aluminum surfaces is presented. As previously underlined, to get superhydrophobicity the substrate has to be treated to get the proper superficial roughness as well as coated to lower surface free energy.

Sarkar *et al.* (2008) demonstrated superhydrophobicity on micropatterned aluminum surfaces obtained by chemical etching with hydrochloric acid (HCl). After the etching treatment the surfaces were coated with an ultrathin rf-sputtered Teflon film using Ar plasma in an inductively coupled plasma reactor. The optimum etching time to obtain high superhydrophobic properties on the surface was 2.5 minutes (contact angle up to 164° and contact angle hysteresis down to 2.5°). More recently, Sarkar *et al.* (2010) investigated wetting characteristics of micro-nanorough substrates of aluminum chemically etched using dilute HCl, comparing them by depositing hydrocarbon and fluorinated-hydrocarbon coatings via plasma enhanced chemical vapor deposition. The authors found a contact angle equal to 135° for the hydrocarbon coated surface, and equal to 165° for the fluorinated-hydrocarbon coated one.

A particular treatment to vary the surface wettability is the one proposed by Saleema *et al.* (2010), where the creation of superhydrophobic properties on aluminum alloy surfaces treated with a simple one-step process is reported. This process incorporates low surface energy compounds in the solution used for the creation of surface micro-roughness. Clean Al coupons were superhydrophobized by dipping them in beakers containing a mixture of sodium hydroxide (NaOH) and fluoroalkyl-silane (FAS-17) at different FAS-17 to NaOH ratios. The authors verified that the formation of a rough microstructured surface, in combination with a modified surface chemistry, leads to the creation of superhydrophobic properties. In particular with 1M NaOH and 400 mM FAS-17 the authors obtained a contact angle equal to 162° and a contact angle hysteresis equal to 4° . An alternative approach is the one proposed by Chen *et al.* (2010), who developed a method for fabricating superhydrophobic aluminum surfaces by chemical etching without hydrophobic treatment. Washed aluminum substrates were treated with 1 mol L^{-1} NaOH aqueous solution for 10 min at room temperature, and immersed in a mixed acid solution of hydrochloric (36 wt%) and acetic acids (CH_3COOH - 99.6 wt%) at the desired ratio of volumes for the desired time. The authors found that water contact angle increases with increasing etching time, until a critical value equal to 5 min, and with increasing acetic acid concentration, until a critical value equal to 1mL.

Finally, an innovative method to render aluminum substrates superhydrophobic was proposed by Jafari and Farzaneh (2011). The authors treated the samples by consecutive immersion in boiling water, to roughen them, and Teflon sputtering, to create the hydrophobic coat. To achieve different roughness levels, the duration of the boiling process was varied. They demonstrated that high superhydrophobic properties (contact angle equal to 164° and contact angle hysteresis equal to 4°) can be obtained after 5 minutes immersion in boiling water.

3.4 State of the art about dropwise condensation analysis

Dropwise condensation on rough superhydrophobic surfaces has been reported in the literature (Dorrer and Rhuhe 2007; Narthe and Beysens 2004; Varanasi *et al.* 2009). Recently Lan *et al.* (2010) reported heat transfer measurements on a superhydrophobic nanostructured copper sample and compare them to a mirror polished hydrophobic sample. They found that the nanostructured substrate do not improve the dropwise condensation heat transfer performance as expected by the higher contact angle. Dietz *et al.* (2010) reported a study of the dynamic condensation on a superhydrophobic sample. Droplet departure frequency was investigated using environmental

Scanning Electron Microscopy on a tilted (30°) superhydrophobic surface formed by cupric hydroxide nanostructures. Comparing these results with a nonstructured hydrophobic surface, they found that the droplets tend to depart from the surface at reduced diameter. Since droplets with diameter less than 10 μm provide the most significant contribution to the heat transfer during dropwise condensation and the formation of new droplets occurs only once large drops have departed from the surface, they concluded that the larger surface renewal frequency of the superhydrophobic surface leads to an increase in the heat transfer coefficient. They estimate an increase by a factor two but no measurements have been done. Berndt *et al.* (2008) presented an experimental and theoretical study of dropwise condensation of vapor in a three channels plate heat exchanger with the plates coated with hydrophobic Ni-P-PFA. They showed an increase in heat flux of up to 20%, and an enhancement of about 100% in term of heat transfer coefficients with respect to the filmwise mode. However, this enhancement was limited at low subcooling ($<6^{\circ}\text{C}$) since for higher value they observe a transit to the filmwise regime. High temperature saturated vapor flow condensation tests on superhydrophobic nanotextured copper surfaces presented by Torresin *et al.* (2013) show that condensing drops form and penetrate into the surface texture and attain a Wenzel state, but the vapor shear-induced droplets roll-off compensates for the reduced drops mobility and enhances the overall thermal transport. These results underlined the potential of nanostructured surfaces for application in flow condensation conditions.

When designing and fabricating an experimental apparatus for analysis of condensation phenomena over nano-engineered surfaces one has to face some challenges. First of all, the heat flux must be determined indirectly, by looking at the coolant side or at the wall. Beside the heat flux, an accurate evaluation of the heat transfer coefficient associated with the two-phase phenomena requires an accurate evaluation of the surface temperature. Experimental techniques that do not require the measurement of the wall temperature should not be used here, being the main thermal resistance on the coolant side. Estimating through correlations the heat transfer coefficient on the coolant side will thus lead to high experimental uncertainty in this case. As an additional problem, direct measurement of surface temperature over a nano-engineered substrate, by soldering thermocouples over it, is not feasible, since this would locally modify surface wetting properties, leading to a modification of the condensation mode.

For the experimental analysis of heat transfer during condensation of pure or non-pure steam the majority of the experimental setups that it is possible to find in the open literature are composed by three main parts: an evaporator, a condensing chamber and a cooling system (Torresin *et al.*, 2013; Lan *et al.*, 2010; Ma *et al.*, 2008; Ma *et al.*, 2012; Rausch *et al.*, 2008; Kananah *et al.*, 2006;

Kananeh *et al.*, 2011; Vemuri *et al.*, 2006). Auxiliary systems can be introduced for complete vapor condensation, for condensate collection, for process visualization and for non-condensable gas introduction.

At the best of the authors knowledge, two main techniques for heat transfer measurement during condensation on engineered surfaces can be found in the literature: one based on Fourier's law with an evaluation of temperature difference between saturation and surface through the temperature profile inside sample over which condensation occurs (Torresin *et al.*, 2013; Lan *et al.*, 2010; Ma *et al.*, 2008; Ma *et al.*, 2012), one based on an estimation of convective heat transfer coefficient during condensation outside tubes (Kananeh *et al.*, 2006; Kananeh *et al.*, 2011; Vemuri *et al.*, 2006).

When evaluating surface temperature through Fourier's law cylindrical condensing blocks (as the one shown in Figure 3.4.1), well insulated to ensure one-dimensional steady state conduction, are typically used. These are fitted with thermocouples to obtain the temperature gradient, thus to extrapolate the temperature of the condensation surface. To match extremely high condensation heat transfer capacity heat transfer surface at the coolant side is normally enhanced with several fins. To get a proper extrapolation of the surface temperature several thermocouples are used, thus the sample needs to have a considerable height. Considering the extremely high heat fluxes expected during condensation this technique leads to a significant temperature difference between the condensation side and the coolant side of the cylindrical sample, thus it can be used only with metals having very high thermal conductivity, as copper and aluminum, but it cannot be applied with metals having lower thermal conductivity, as steel, without entering with the cooling fluid at extremely low temperatures.

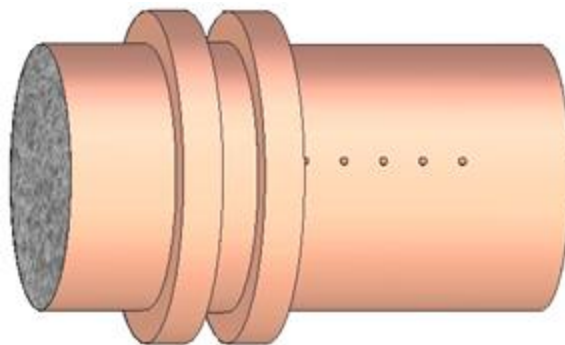


Figure 3.4.1. Cylindrical sample used for DWC analysis, sketch from Torresin *et al.* (2013).

When performing steam condensation outside tubes (Figure 3.4.2) the main thermal resistance is located on the internal side of the wall, because of much higher condensation heat transfer

coefficient than convective one. This is even empathized if condensing in dropwise mode over engineered-treated surfaces. Because of this, experimental techniques based on the estimation of convective heat transfer coefficient inside a tube to get the one on the external condensation side are not enough accurate. Since the main part of the overall heat resistance is located on the internal side, even a small error in the estimation of the convective heat transfer coefficient will lead to high uncertainty on the estimation of the condensation one.

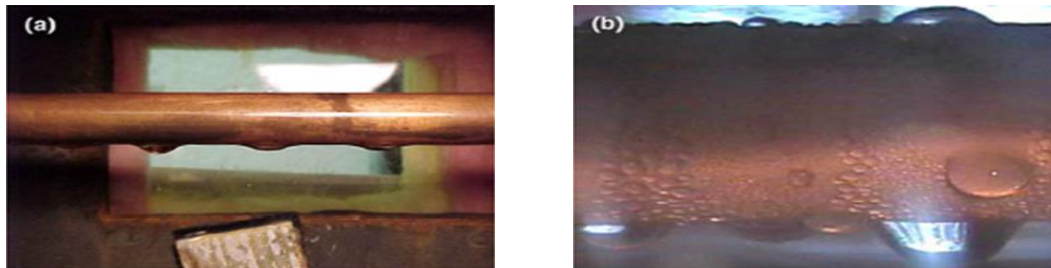


Figure 3.4.2. Condensation outside tube, photo from Vemuri et al. (2006).

Because of all these considerations, in this Thesis a new experimental apparatus for investigation of pure steam condensation over nano-engineered surfaces is presented. This apparatus aims to be used for analysis of condensation over substrates of different material (also with low thermal conductivity) providing a precise evaluation of the heat transfer properties.

3.5 Experimental setup

In this Paragraph, the experimental apparatus built at the Two-Phase Heat Transfer Laboratory of the Department of Industrial Engineering of Padova University for analysis of condensation over plain surfaces is presented. First, the two-phase flow loop and the test section are described in detail. Then, the data reduction technique is presented, as well as the instrument calibration and the uncertainty analysis. Finally, the calibration of the whole apparatus is discussed.

3.5.1 Two-phase flow loop description

A schematic diagram of the experimental flow loop presented in this Thesis is shown in Figure 3.5.1.1.

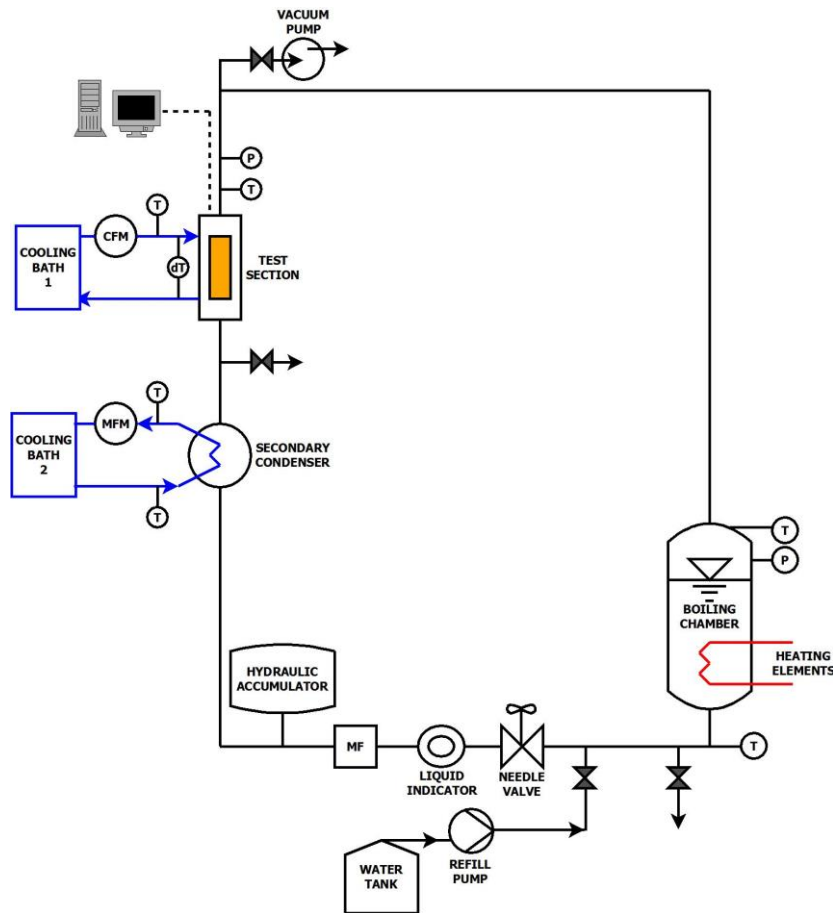


Figure 3.5.1.1. Schematic of the experimental thermosyphon flow loop for condensation tests. P = Pressure transducer, T = Thermocouple, dT = Thermopile, CFM = Coriolis mass Flow Meter, MFM = Magnetic mass Flow Meter, MF = Mechanical Filter.

The system consists of four main components: the boiling chamber, the test section, the cooling water loop and the post-condenser. Steam is generated in a cylindrical stainless steel boiling chamber which is connected to the test section by a stainless steel vapor line. The water vaporization is promoted inside the chamber by means of four electrical heaters with a total power of 4 kW. The electrical power supplied to the heaters is measured using a power analyzer NORMA 4000. The pipe connections between the boiler and the test section are well insulated and heated by means of a resistance wire installed around the pipe, to avoid formation of condensate before the entrance of the test section (wall temperature is also checked through a T-type thermocouple).

The steam enters the test section in saturated conditions with a vapor quality equal to one. In the test section (presented in Section 3.5.2) the steam is partially condensed over the test aluminum surface and the latent heat is removed by the cold water coming from a thermostatic bath. The coolant inlet temperature is measured by a T-type thermocouple, while coolant temperature

difference between the inlet and the outlet is measured by means of a three-junction copper-constantan thermopile. To assure precise evaluation of heat flux, the coolant mass flow rate is measured by means of a Coriolis effect mass flow meter. The vapor pressure and temperature are gauged at the inlet of the measuring section by means of a differential pressure transducer (coupled with an absolute one for ambient pressure evaluation) and a T-type thermocouple, respectively.

Downstream the test section, the two-phase mixture passes through a secondary water condenser where the condensation is completed and the liquid subcooled. The post-condenser consists of a cylinder which contains a coiled tube: the vapor condenses on the external surface of the tube, while the cooling water flows inside the pipe. The subcooled liquid returns to the boiler driven by the difference between liquid and vapor density and it completes the loop. The water temperature is measured at the inlet and the outlet of the post-condenser by means of T-type thermocouples while water mass flow rate is measured using a magnetic flow meter.

To regulate the system pressure, a hydraulic accumulator is installed in the liquid line downstream the post-condenser. A precise needle valve, placed before the boiling chamber, is used to regulate the liquid flow and it allows to achieve stable conditions during the tests. In between the hydraulic accumulator and the needle valve a mechanical filter and a liquid indicator are installed. Before entering the boiling chamber the temperature of the subcooled liquid is measured by means of a T-type thermocouple.

All the components of the test rig, with the exception of the test section, are made by stainless steel in order to avoid contamination of the fluid. Finally, after the construction of the test rig, the boiling chamber and the stainless steel lines were well insulated in order to avoid heat losses to the ambient.

Since even a few concentration of non-condensable gases (NCG) in the vapor could lead to huge decrease in the thermal performance of the condensation process, several actions have been undertaken to avoid it. First of all, it has been decided to operate always in overpressure conditions with respect to the ambient in order to prevent air infiltration inside the test loop. Furthermore, before each test run, the whole system is vacuumed; then the test rig is charged with deionized (DI) water coming from the supplying tank and pumped inside the test rig by a centrifugal pump. When the pressure inside the setup becomes higher than the ambient one, water is released from the top valve while the refill pump is continuing to run, in order to get rid of any non-condensable gases still inside the setup. After one minute of free water discharging, the top valve is closed and then also the filling line. Subsequently the boiler is started: when the water reaches the saturation temperature and boils for several minutes, the vapor is released

from the top of the system as well as from a discharge valve located in the upper part of the post-condenser. This procedure is repeated several times in order to get rid also of the gases dissolved in the water. In all these operations particular care is observed in order to be always in overpressure inside the test rig. If non condensable gases were present inside the setup, a disagreement between the measured saturation pressure and the one calculated from the measured temperature should be observed. In particular, if NCG are present, the measured saturation pressure should be greater than the one calculated from the measured saturation temperature. On the contrary, in all the data presented below, the saturation pressure calculated from the temperature is higher than the measured one and the observed discrepancy between the two temperatures is always inside the experimental uncertainty of the pressure transducer and the thermocouple.

For testing a sample over several days, the hydraulic accumulator is used to maintain the system in overpressure overnight, thereby avoiding the need for DI water refilling every day.

Since it has been decided not to install a circulating pump, several considerations have been made in order to assure the natural convection of the fluid. First of all, the dimensions of the evaporator have been properly defined. This is very important because the electrical heaters must always be wet by the water. To guarantee the liquid return in the boiler, a liquid head is necessary and the post-condenser must be placed at a higher level than the boiling chamber. The total pressure drop due to the sum of the frictional pressure losses in the vapor line and in the test section, plus the local pressure drops due to abrupt geometry changes have been calculated. With a maximum mass flow rate equal to 5.57 kg h^{-1} a value of 4340 Pa has been obtained, corresponding to a required height difference between the liquid level in the boiling chamber and the one in the post-condenser equal to about 0.44 m.

Table 3.5.1.1 reports the pressure drop calculations for different vapor mass flow rates. These are evaluated from the power given at the boiling chamber, as it will be shown in Section 3.5.3.

$Q_{\text{BoilingChamber}}$ [kW]	m_{steam} [kg h ⁻¹]	u_{steam} [m s ⁻¹]	Δp_{TOT} [Pa]	Δp_{TOT} [mmH ₂ O]
0.5	0.70	1.6	72.4	7.4
1	1.39	3.1	283.3	28.9
1.5	2.09	4.7	628.6	64.1
2	2.78	6.3	1107.1	113.0
2.5	3.48	7.8	1718.0	175.3
3	4.18	9.4	2460.8	251.1
3.5	4.87	10.9	3334.8	340.3
4	5.57	12.5	4339.8	442.8

Table 3.5.1.1. Pressure losses in the vapor line.

If p_{SAT} is the saturation pressure in the system and Δp_{TOT} the sum of the local and distributed pressure losses from the boiling chamber exit to the post-condenser inlet, the pressure at the inlet of the post-condenser is:

$$p_{\text{post-cond}} = p_{\text{sat}} - \Delta p_{\text{TOT}} \quad (3.7)$$

In order to assure the return of the liquid in the boiling chamber, the height ($H_{\text{post-cond}}$) of the condensate in the post-condenser, with respect to the liquid level in the boiling chamber, should be:

$$H_{\text{post-cond}} \geq \frac{\Delta p_{\text{TOT}}}{\rho_L \cdot g} \quad (3.8)$$

where ρ_L is the liquid density and g is the acceleration gravity term.

Even in the case of the higher mass velocity (i.e. 5.57 kg h⁻¹), when the pressure losses are maximum, the condensate level inside the post condenser has to provide the necessary overpressure to assure its return in the boiling chamber. For this reason, working from the side of safety, it has been decided to position the bottom surface of the post-condenser 50 cm above the top surface of the boiling chamber.

Once it has been decided the position and dimensions of the subcooler, further considerations have been made in order to choose the K_v of the needle valve to install at the inlet of the boiling chamber. In fact, since the mass flow rates are very low, a local pressure drop which could be finely regulated is necessary to stabilize the flow and guarantee stationary conditions. Calculations have been made considering that the maximum pressure drop admitted across the needle valve is due to the difference between the sum of the Δp_{TOT} and the pressure gain of the condensate column in the subcooler.

Finally, it has to be pointed out that in all the calculations that will be reported later on, the heat losses towards the ambient in the tube connecting the boiling chamber to the test section are neglected. In fact, a simple estimation of heat losses using the correlation for natural convection, yields a heat loss of about 2%, thus justifying our approach to neglect the heat losses of the setup. Figure 3.5.1.2 shows a photo of the whole experimental apparatus.

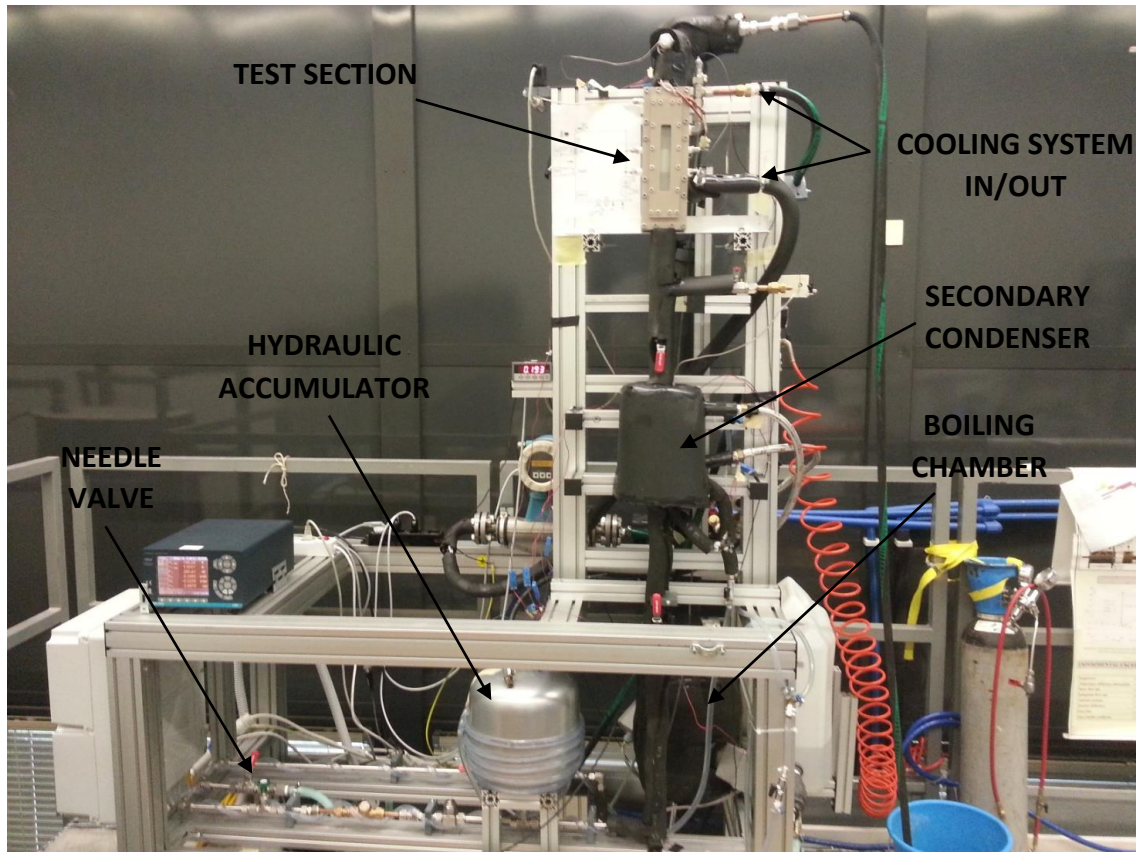


Figure 3.5.1.2. Photo of the whole apparatus. Main components are reported.

3.5.2 Description of the test section

The test section is designed for the measurement of the heat transfer coefficient over a metallic surface and for the simultaneous visualization of the condensation process. It consists of several pieces realized both in PEEK and glass. A rectangular channel 160 mm long (cross section 30 mm x 5 mm) was grooved into a PEEK block and the vapor, coming from the boiling chamber, flows inside it. The PEEK channel is fitted with a rectangular aluminum plate (vertically oriented), over which condensation occurs. One side of the channel is covered by a double glass (with an air chamber for thermal insulation) to allow the visualization of the process. In addition, the frontal glass is heated up by means of an electrical heater to avoid vapor condensation over it. The other side of the channel, opposite to the glass, is machined for accommodating the metallic substrate.

The metallic specimen has 10 mm thickness and the condensation surface dimensions are 50 mm x 20 mm.

The specimen is fitted with four T-type thermocouples located at the inlet and at the outlet of the condensing surface, inside four 0.7 mm holes obtained by electroerosion 1 mm and 2.75 mm below the surface of the sample.

Figure 3.5.2.1 reports two photos of the metallic substrate.

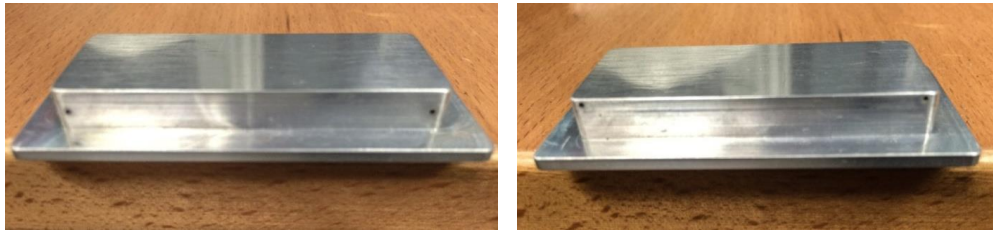


Figure 3.5.2.1. Photos of the metallic specimen from two opposite sides.

The sample is located in between the two main parts of the test section: one contains the steam channel (in contact with the frontal face of the specimen) and the other contains the cooling system (in contact with the back face of the specimen).

Figure 3.5.2.2 shows a scheme of the position of the sample inside the test section.

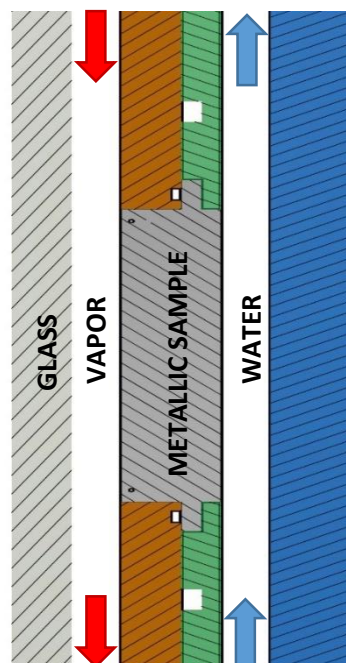


Figure 3.5.2.2. Scheme of the position of the metallic sample inside the test section. Red and blue arrows refer to vapor and cooling water flow directions, respectively.

The cooling system consists of a 20 mm x 5 mm rectangular PEEK channel and it is used to remove the heat from the condensation process. The cooling water flows in countercurrent with respect to the steam direction inside the test section.

In between the vapor block and the cooling system block, two frames are located for thermocouples and gaskets positioning. A sketch of the whole test section can be seen in Figure 3.5.2.3. Finally, Figure 3.5.2.4 shows a photo of the components of the test section before and after setting them up.

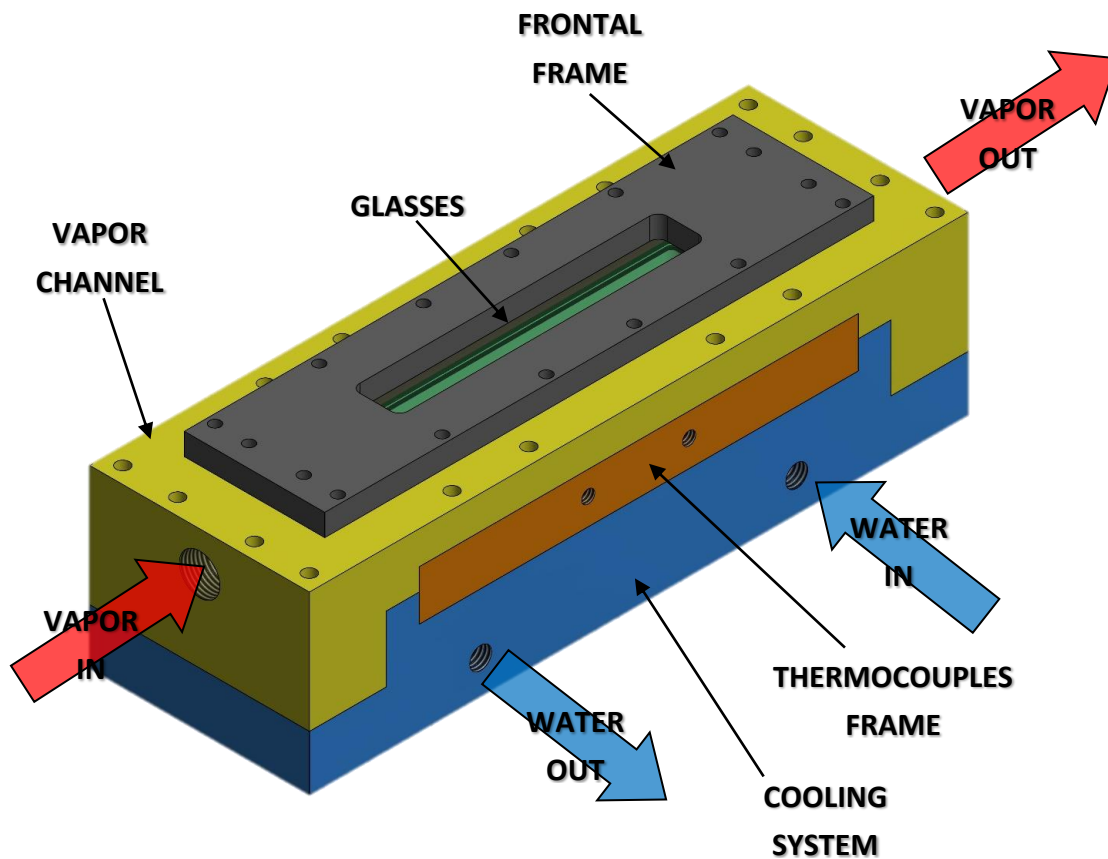


Figure 3.5.2.3. Sketch of the whole test section.

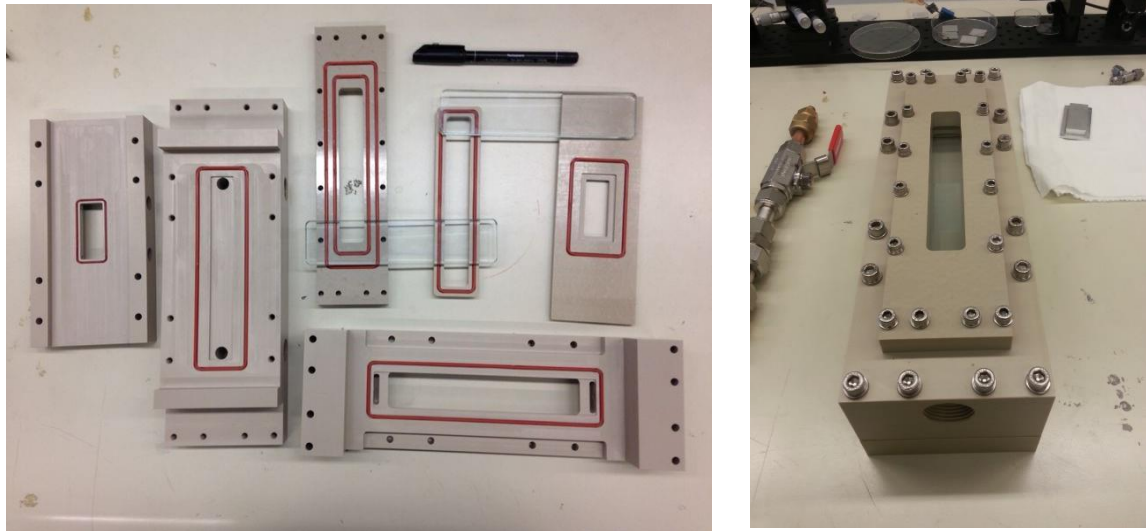


Figure 3.5.2.4. Photo of the components of the test section before (left) and after (right) assembling.

CFD simulations have been carried out to optimize the geometry of the cooling channel, with the goal to achieve a high and uniform heat transfer coefficient on the cooling side that guarantees an almost one-dimensional temperature profile inside the specimen. Positions of the inlet and outlet manifolds, as well as the length of the rectangular cooling channel, were determined in order to obtain a uniform velocity profile of the water in contact with the back part of the cooling system. The geometry (symmetrical) that gave the best results is reported in Figure 3.5.2.5.

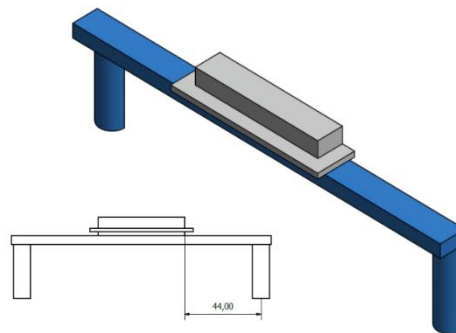


Figure 3.5.2.5. Simulated geometry.

Simulations were performed using ANSYS/FLUENT 13, fixing water inlet temperature and velocity as well as steam temperature and condensation heat transfer coefficient. Figure 3.5.2.6 reports the results of one of the simulations, which show that with the actual geometry the temperature profile along the specimen is uniform (thus the heat flux) as well as the water velocity profile of the water in the channel.

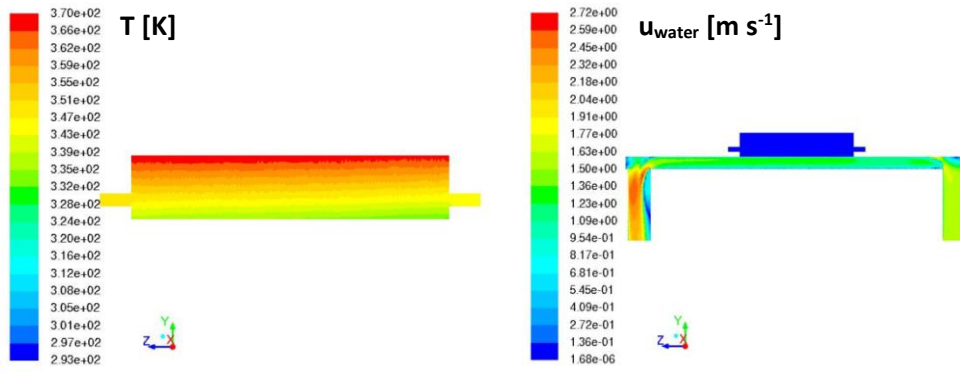


Figure 3.5.2.6. Results of one of the CFD simulations. Temperature profile along the specimen (left) and water velocity profile inside the cooling system channel (right).

3.5.3 Data reduction

Since the coolant mass flow rate $\dot{m}_{coolant}$ is directly measured, as well as the inlet coolant temperature $T_{IN-coolant}$ and the coolant temperature difference $\Delta T_{coolant}$, the heat flow rate extracted by the cooling water from the condensing vapor can be obtained as

$$\dot{Q}_{cooling} = \dot{m}_{coolant} c_{p,coolant} \Delta T_{coolant} \quad (3.9)$$

where $c_{p,coolant}$ is the specific heat capacity of the coolant evaluated at the mean temperature between the inlet and the outlet.

Hence, the condensation heat flux q is given by

$$q = \frac{\dot{Q}_{cooling}}{A} \quad (3.10)$$

where A is the heat transfer area.

Since the wall temperatures are measured with thermocouples 1 mm and 2.75 mm below the surface of the sample, wall temperatures can be obtained from the measured values with the hypothesis of one-dimensional temperature distribution extrapolation. Thus, at the inlet and outlet of the aluminum sample the superficial temperatures are

$$T_{SUP,IN} = T'_{IN} + (T'_{IN} - T''_{IN}) \frac{z_1}{z_2 - z_1} \quad (3.11)$$

$$T_{SUP,OUT} = T'_{OUT} + (T'_{OUT} - T''_{OUT}) \frac{z_1}{z_2 - z_1} \quad (3.12)$$

where T' are the inlet and outlet temperatures measured in the aluminum sample at $z_1 = 1$ mm and T'' are the inlet and outlet temperatures measured in the aluminum sample at $z_2 = 2.75$ mm.

As an example, Figure 3.5.3.1 reports inlet and outlet temperatures profiles for a test case, with a saturation temperature of the steam equal to 106.6°C.

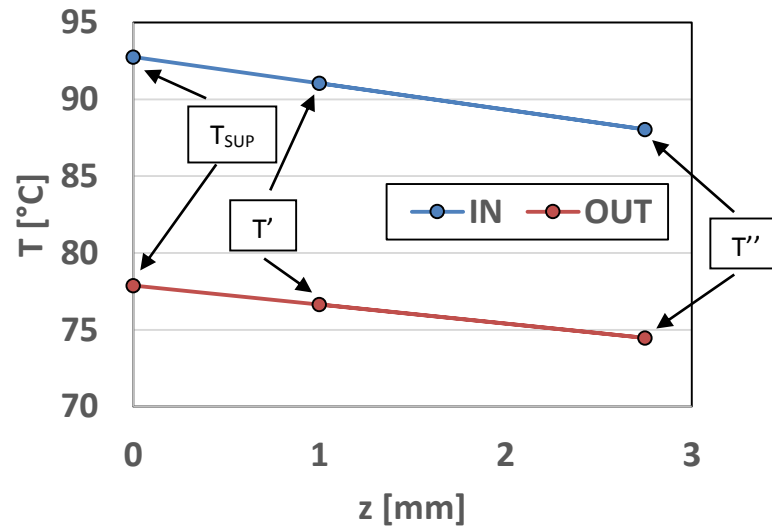


Figure 3.5.3.1. Temperature profiles at the inlet and outlet of the metallic specimen for a case test ($T_{sat} = 106.6^{\circ}\text{C}$).

Once the superficial temperatures are known, the condensation heat transfer coefficient is evaluated as

$$\text{HTC} = \frac{q}{\Delta T_{ml}} \quad (3.13)$$

where ΔT_{ml} is the mean logarithmic temperature difference between the surface and the steam. The steam temperature is directly measured by a T-type thermocouple located at the inlet of the test section and checked using vapor pressure to evaluate non-condensable gases presence.

The enthalpy of the subcooled liquid at the inlet of the boiling chamber as well as the one of the saturated steam are evaluated by means of NIST Refprop Version 9.1 (see Lemmon *et al.*, 2013), from temperature and pressure measurements.

The steam mass flow rate $\dot{m}_{steam}$ is obtained from Eq. (3.14), where Q_{BC} is the heat flow rate in the boiling chamber (measured from the electrical power supplied to the heaters), besides h_v and $h_{IN,BC}$ are the enthalpies of the saturated steam and of the subcooled liquid at the entrance of the boiling chamber, respectively.

$$\dot{m}_{steam} = \frac{Q_{BC}}{h_v - h_{IN,BC}} \quad (3.14)$$

Once the mass flow rate is known, it is easy to calculate the mass velocity of the vapor flowing inside the test section as

$$\dot{G}_{steam} = \frac{\dot{m}_{steam}}{S} \quad (3.15)$$

Where S is the cross section of the channel.

By acting on the power supplied to the boiling chamber, is it possible to regulate the mass flow rate of the fluid inside the system, thus to perform tests at different vapor mass velocity.

3.5.4 Instruments calibration

All the instruments installed in the test apparatus were calibrated prior to perform any heat transfer measurement.

Temperature transducers (thermocouples and thermopile) were calibrated against Heart Scientific Super Thermometer II, coupled with two four wires Standard AS115 Thermistors in the range 10°C – 60°C and with a PT100 Thermistor in the range 60°C – 100°C. The measuring chain (Superthermometer + Thermistor) has a global accuracy equal to ± 0.002 K when using the AS115 probes and equal to ± 0.01 K when using the PT100 probe.

Differential pressure transducer was calibrated against a relative Druck DPI 605 calibrator, while the absolute pressure transducer was checked coupling the Druck DPI 605 calibrator with a Fortin-type barometer.

Comparing the data read by the instruments and the calibrators, proper interpolating functions have been found and inserted in the data acquisition program. After the calibration procedure, the discrepancies between the values acquired by the transducers and the calibrators were reduced to ± 0.03 K for the thermopile, to ± 0.05 K for the thermocouples and to ± 1.5 mbar for the pressure transducers.

Figure 3.5.4.1 reports a photo of the setup used for temperature sensors calibration.

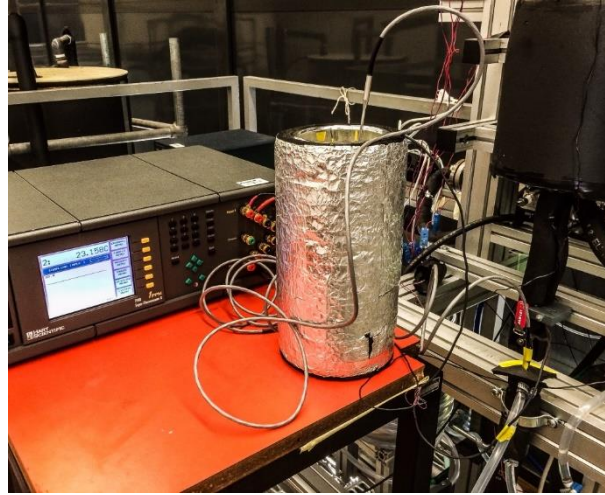


Figure 3.5.4.1. Photo of the temperature calibration system (Superthermometer + Thermistors).

3.5.5 Uncertainty analysis

The experimental uncertainties are calculated following the general rules reported in ISO Guide to the Expression of Uncertainty in Measurement (1995). For each parameter, the combined uncertainty u_c is calculated considering “Type A” and “Type B” components.

Type A uncertainty is equal to the standard deviation of the measured value while Type B uncertainty is related to the instrument properties. Table 3.5.5.1 reports the Type B uncertainty of the most relevant measured parameters. The uncertainty in the thermocouples position is assumed to be zero.

Temperature	$\pm 0.05 \text{ K}$
Cooling water flow rate	$\pm 0.15\% \pm \left[\frac{0.1}{\dot{m}} 100 \right] \% \text{ of the flow rate } \dot{m}$
Differential pressure	$\pm 0.04\%$
Absolute pressure	$\pm 0.1\%$
Electrical power	$\pm 0.1\%$

Table 3.5.5.1. Type B experimental uncertainty of the measured parameters.

The global uncertainty of the measured data x_i is evaluated as

$$u_{TOT}(x_i) = \sqrt{u_A^2(x_i) + u_B^2(x_i)} \quad (3.16)$$

Where $u_A(x_i)$ and $u_B(x_i)$ are respectively the Type A and Type B uncertainties of the i-th parameter x .

The combined standard uncertainty of the derived data $y = f(x_1, x_2, \dots, x_n)$ is evaluated as

$$u_c(y) = \sqrt{\sum_{i=1}^n \left(\frac{df}{dx_i}\right)^2 u_{TOT}^2(x_i)} \quad (3.17)$$

The expanded uncertainty $u_M(y)$ is finally obtained by multiplying $u_c(y)$ by a coverage factor $k = 2$. In this analysis the uncertainties of the thermodynamic properties evaluated using NIST Refprop Version 9.1 (Lemmon *et al.*, 2013) are neglected.

The results of the uncertainty analysis lead to an expanded uncertainty of the mean logarithmic temperature difference between the steam and the wall equal to about $\pm 1.1\%$, an expanded uncertainty of the heat flux removed in the measuring section equal to about $\pm 5.2\%$ and an expanded uncertainty of the heat transfer coefficient equal to about $\pm 7.4\%$.

3.5.6 Calibration tests

Preliminary tests were performed during filmwise condensation of pure steam at different vapor velocities over a conventional (untreated) metallic sample: the temperature difference between the saturated steam and the condensation surface was varied by acting on the cooling system, thus on the removed heat flux. The data set was preliminarily used to validate the system performances, by checking the thermal balances of the whole system and of the test section alone. A magnetic mass flow meter and two T-type thermocouples allow the measurement of the heat flow rate extracted by the water in the secondary condenser as

$$Q_{post-cond} = \dot{m}_{water} c_{p,water} (T_{OUT-water} - T_{IN-water}) \quad (3.18)$$

Thus, knowing the heat power provided by the electrical heaters to the boiling water in the chamber Q_{BC} (measured with the NORMA 4000), it is possible to check the heat balance of the whole system, expressed by Eq. (3.19).

$$Q_{BC} = Q_{cooling} + Q_{post-cond} \quad (3.19)$$

Figure 3.5.6.1 shows the comparison between the heat supplied to the system (at the boiling chamber) and the one extracted from it (by the cooling system in the test section and at the post-condenser). Lines relative to $\pm 10\%$ uncertainty band and $\pm 15\%$ uncertainty band are also reported.

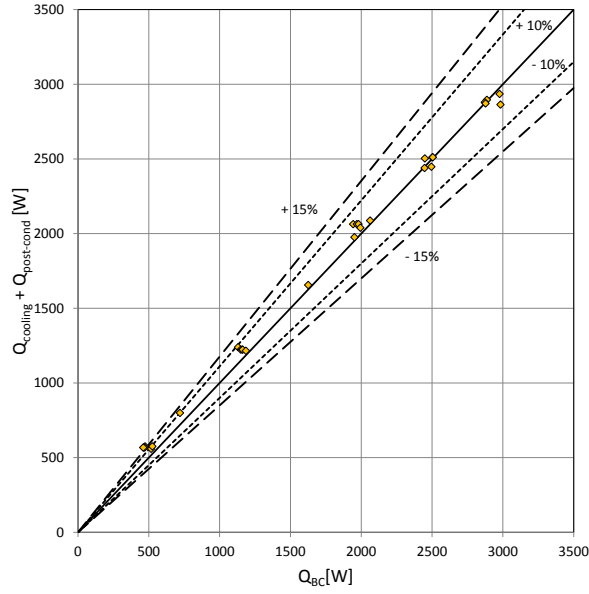


Figure 3.5.6.1. Thermal balance of the experimental apparatus. Comparison between the heat given to the system (at the boiling chamber) and the one extracted from it (at the test section and the post condenser).

The Figure shows that the global thermal balance of the system is satisfactory respected. All the data are inside the $\pm 10\%$ band (the majority of them presents an uncertainty band lower than $\pm 5\%$), except those at the lower Q_{BC} , which are inside the $\pm 15\%$ band. For these data the relative uncertainty of the measures is higher than the others, but still the global balance of the system is satisfactory.

Moreover, since the four thermocouples embedded in the metallic specimen are located at two different z positions, it is possible to evaluate two local heat fluxes, one at the inlet and one at the outlet of the metallic specimen, according to Fourier's law and assuming a one-dimensional heat flux along the specimen (this assumption is justified considering the much lower temperature gradient in the longitudinal direction of the specimen compared to the gradient in the direction orthogonal to the sample surface).

$$\begin{cases} q_{IN} = k \left| \frac{dT_{IN}}{dz} \right| \\ q_{OUT} = k \left| \frac{dT_{OUT}}{dz} \right| \end{cases} \quad (3.20)$$

The average heat extracted along the sample can thus be calculated as

$$Q_{AVE} = \frac{1}{2} (q_{IN} + q_{OUT}) A \quad (3.21)$$

By comparing the results obtained with Eq. (3.21) against the heat flow rate evaluated through Eq. (3.9) it is possible to verify the thermal balance of the test section alone, as shown in Figure 3.5.6.2. All the data presented in the Figure falls within the $\pm 10\%$ band.

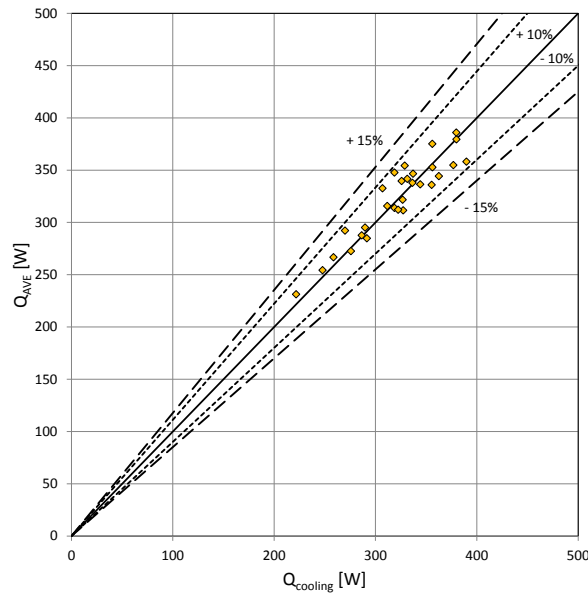


Figure 3.5.6.2. Thermal balance in the test section. Comparison between the heat measured at the cooling side and the one evaluated by looking at the temperatures profiles inside the specimen.

From Figures 3.5.6.1 and 3.5.6.2 it is possible to infer that the systems works properly and that the measuring technique is accurate.

3.6 Condensation over standard aluminum surfaces

Filmwise condensation tests over standard (untreated) aluminum specimens are hereinafter presented. These tests were performed with the aim to study the effect of the shear stress on the filmwise condensation phenomena, as well as to perform an additional validation of the setup before working with nano-engineered samples. Because of this, experimental data were also compared against heat transfer coefficients predicted by the literature, as shown in Section 3.6.2.

3.6.1 Experimental analysis

Condensation heat transfer data were acquired at about 106°C saturation temperature, which correspond to a saturation pressure ≈ 1.25 bar. Measurement tests were performed changing the

inlet water temperature and the mass flow rate at the cooling system, resulting in a different mean logarithmic temperature difference between the steam and the wall. Tests were performed varying the vapor mass velocity by acting on the boiling chamber power.

Figure 3.6.1.1 reports the measured heat flux and the corresponding condensation heat transfer coefficient as a function of the wall subcooling degree, at different mass velocity G_{steam} .

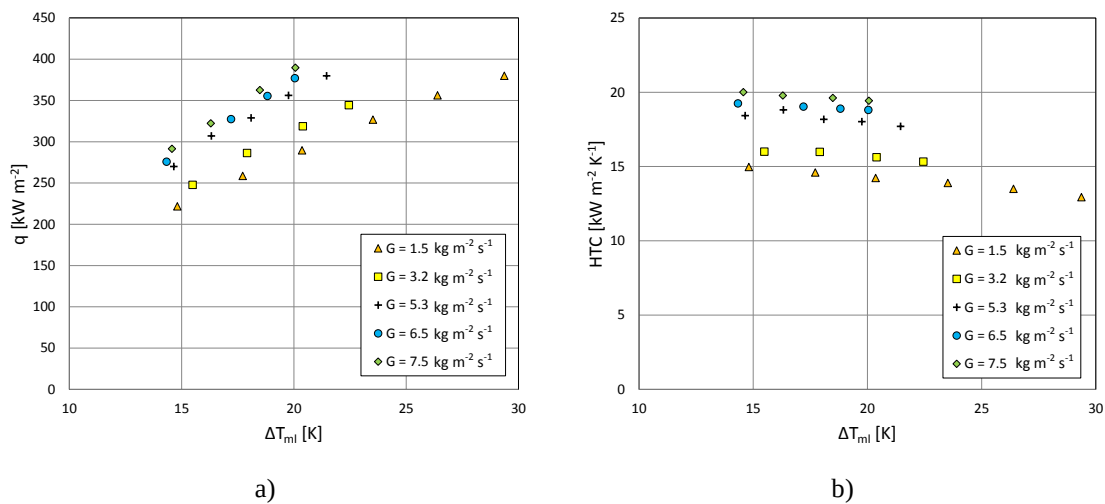


Figure 3.6.1.1. Heat flux (a) and heat transfer coefficient (b) versus mean logarithmic temperature difference between the saturated vapor and the surface during condensation over the conventional sample. Data refer to different steam mass velocities G [kg m⁻² s⁻¹].

The Figure shows that the heat flux increases when increasing the wall subcooling degree and the steam mass flow rate. On the other side, the condensation heat transfer coefficient exhibits a different behavior: it remains almost insensitive to the increasing of the mean logarithmic temperature difference ΔT_{ml} (especially at high vapor velocity), while it is strongly affected by variations in the steam mass flow rate. The vapor velocity has an important influence on the liquid film thickness and consequently it affects the condensation heat transfer coefficient. In fact, when the mass flux G_{steam} increases, the velocity of the liquid film increases, leading to a reduction of the thickness of the liquid film near the wall and thus to a reduction of the thermal resistance associated with the condensate layer.

This behavior is underlined also in Figure 3.6.1.2, where the removed heat flux and the condensation heat transfer coefficient are reported as a function of G_{steam} , maintaining constant the coolant water inlet conditions ($T_{IN-coolant} = 15^\circ\text{C}$ and $m_{coolant} = 320 \text{ kg h}^{-1}$).

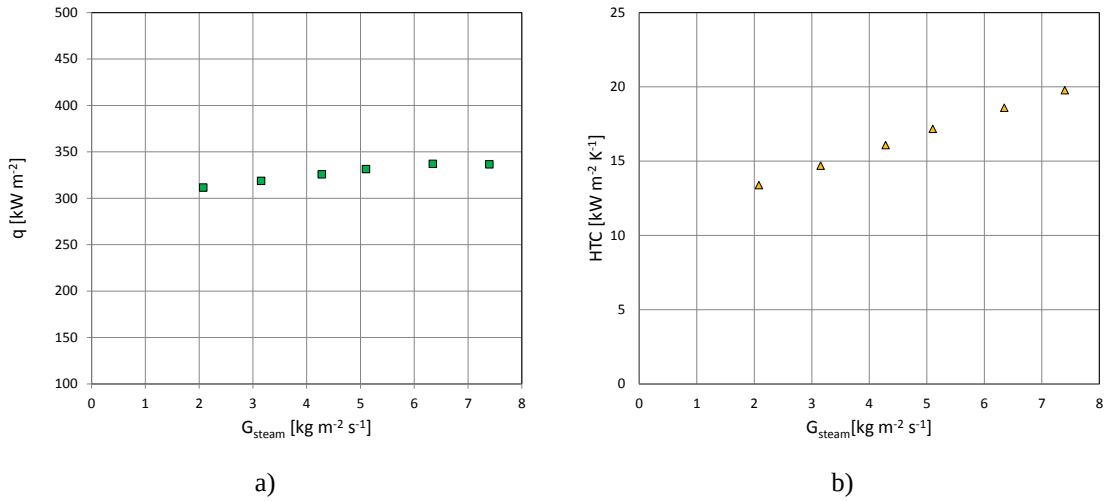


Figure 3.6.1.2. Heat flux (a) and heat transfer coefficient (b) data at different steam specific mass flow rate, maintaining $T_{IN-coolant} = 15^\circ\text{C}$ and $m_{coolant} = 320 \text{ kg h}^{-1}$, during condensation over the untreated sample.

From the experimental data it is possible to see that, increasing the steam specific mass flow rate, the condensation heat transfer coefficient increases almost linearly while the extracted heat remains almost constant. This is because the two-phase heat transfer coefficient is much higher than the convective one on the cooling side, thus the main thermal resistance, which dominates the heat exchange, is located on the back side of the sample. On the other hand the heat transfer coefficient on the hot side is strongly dependent on the thickness of the liquid film at the wall, which reduces when the vapor mass velocity increases.

3.6.2 Theoretical analysis

The experimental data are compared against values calculated using correlations available in the open literature. In the present experimental setup, both gravitational effect and shear stress effect influence the filmwise condensation process, thus both have to be considered in the calculations. The comparison between experimental and calculated data is useful both for theoretically analyze the two-phase process as well as for performing a final validation of the experimental technique.

The heat transfer coefficient for gravity-controlled condensation over vertical plain surfaces can be predicted using the following equation:

$$HTC_{Nu} = 1.15 \cdot \left[0.026 \left(\frac{H_{lv} \mu_l}{\lambda_l (T_{SAT} - T_{wall})} \right)^{\frac{1}{2}} + 0.79 \right] \cdot 0.943 \cdot \left[\frac{\rho_l (\rho_l - \rho_v) g H_{lv} \lambda_l^3}{\mu_l (T_{SAT} - T_{wall}) L} \right]^{\frac{1}{4}} \quad (3.22)$$

where ρ_l is the condensate density, ρ_v is the density of the steam, g is the acceleration of gravity, λ_l is the thermal conductivity of the condensate, μ_l is the dynamic viscosity of the condensate, T_{SAT}

and T_{wall} are the saturation and surface temperature respectively, and L is the condensing surface length.

The second part of Eq. (3.22) derives from the Nusselt (1916) film condensation theory. Since waves formation was observed on the condensate film during the filmwise condensation experiments, the heat transfer enhancement due to this phenomenon has to be taken into account introducing the correction factor 1.15 (Baehr and Stephan, 2004). Inertia effects, which are neglected in Nusselt's theory, are included by the first term in Eq. (3.22) as suggested in Depew and Reisbig (1964). The variation of the physical properties of the condensate with temperature has also to be considered. Here, the viscosity shows the dominant effect and this fact can be compensated by calculating all physical properties of the condensate at the mean temperature:

$$T_m = 0.75 \cdot T_{wall} + 0.25 \cdot T_{SAT} \quad (3.23)$$

while only the latent heat H_{lv} and the density of the saturated steam ρ_v are calculated at T_{SAT} (Hewitt *et al.*, 1964).

When condensation occurs in the presence of vapor velocity, also the effect of the interfacial shear stress has to be considered. For modeling of shear-driven condensation several correlations can be found in the literature. In the present work, the new database is compared against the shear stress correlation proposed in Hewitt *et al.* (1964) for the shear-controlled condensation process with laminar flow in the condensate film (the maximum Reynolds number of the condensate film is about 130 for the present data). When the shear stress force became dominant in comparison to gravitational one, the two-phase heat transfer coefficient can be calculated as

$$HTC_{SS} = \sqrt{\frac{\lambda_l^2 \tau_i \rho_l}{2 \Gamma_L \mu_l}} \quad (3.24)$$

where Γ_L is the condensate rate leaving the surface at a distance L from the top and τ_i is the shear stress at the liquid-vapor interface. The two quantities can be evaluated as follows (Hewitt *et al.*, 1964)

$$\Gamma_L = \frac{HTC_{Nu} (T_{SAT} - T_{wall}) L}{H_{lv}} \quad (3.25)$$

$$\tau_i = -\frac{D_h}{4} \left(\frac{dp}{dz} \right)_F \quad (3.26)$$

where D_h is the hydraulic diameter of the channel and $(dp/dz)_F$ is the two-phase frictional pressure gradient. For the evaluation of the frictional pressure losses, the model by Friedel (1979) has been used.

When condensation occurs under combined gravity and shear control, as in the present case, the global heat transfer coefficient can be calculated as in Eq. (3.27)

$$HTC_{CALC} = \sqrt{HTC_{Nu}^2 + HTC_{SS}^2} \quad (3.27)$$

Figure 3.6.2.1 reports the comparison between the present experimental data and the heat transfer coefficients evaluated with Eq. (3.27).

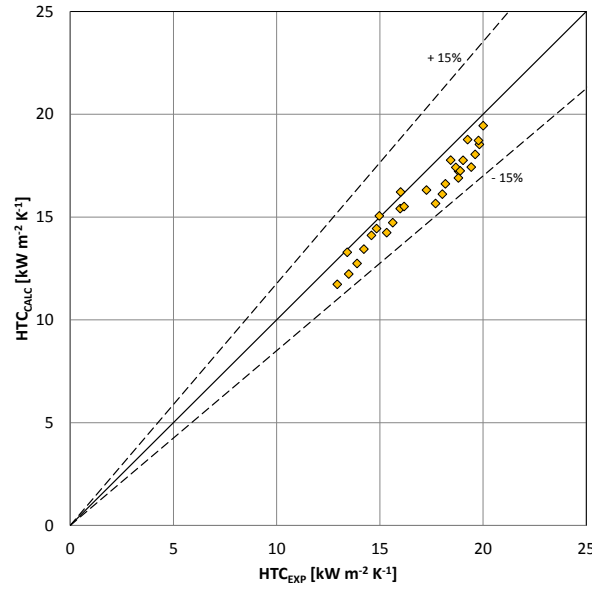


Figure 3.6.2.1. Comparison between experimental and calculated heat transfer coefficient.

The calculated and experimental data fall within $\pm 15\%$ band (reported in Figure 3.6.2.1). The data are predicted by Eq. (3.27) with an average deviation $\bar{e}$ equal to -6%. The standard deviation of the predicted values σ_N is equal to 3.4%. The satisfactory agreement between calculated and experimental data gives a final validation of the performances of the setup and of the measuring technique.

Figure 3.6.2.2 shows the effect of the vapor velocity on the prediction obtained with Eq. (3.27), as well as on the gravity and shear stress components of the filmwise condensation heat transfer coefficient. In Figure 3.6.2.2 a, the “Gravity Component” curve refers to HTC_{Nu} , the “Shear Stress Component” curve to HTC_{SS} , the “CALCULATED” curve to HTC_{CALC} and the “EXPERIMENTAL” curve to the experimental data.

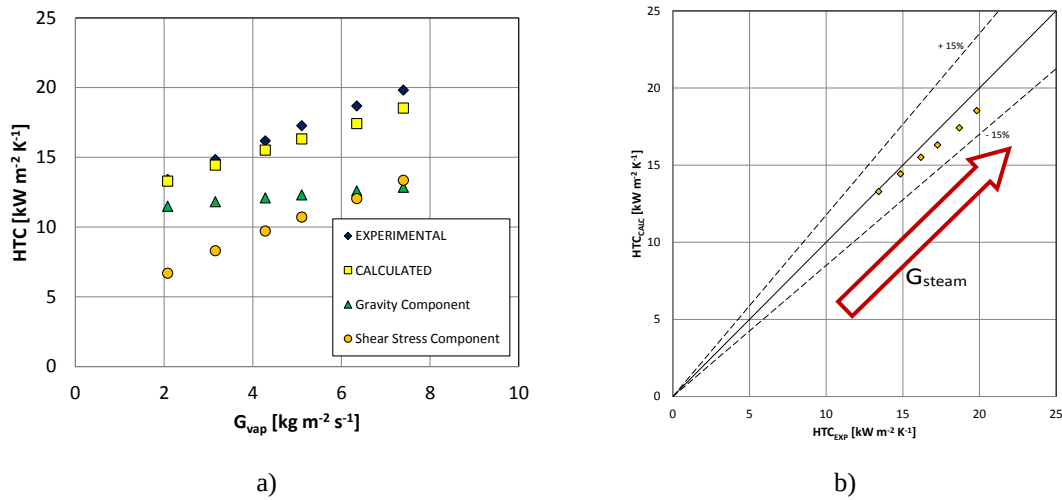


Figure 3.6.2.2. Experimental and calculated heat transfer coefficient as a function of the steam specific mass velocity (a) and comparison between experimental and calculated data (b). Tests at $T_{\text{IN-coolant}} = 15^\circ\text{C}$ and $m_{\text{coolant}} = 320 \text{ kg h}^{-1}$.

The comparison shows the strong influence of the shear stress component on the heat transfer coefficient, that becomes more important at high values of vapor velocity. For low steam specific mass flow rates (i.e. $2 \text{ kg m}^{-2} \text{s}^{-1}$), the velocity component is lower than the gravity one, but still it enhances the global heat transfer coefficient from $HTC_{\text{Nu}} \approx 11.5 \text{ kW m}^{-2} \text{K}^{-1}$ to $HTC_{\text{CALC}} \approx 13.3 \text{ kW m}^{-2} \text{K}^{-1}$. When G_{steam} increases, the specific weight of HTC_{SS} becomes more relevant (since it increases while HTC_{Nu} stays almost constant) and for vapor specific mass fluxes higher than $\approx 6.5 \text{ kg m}^{-2} \text{s}^{-1}$ the shear stress component is predominant. It should be noticed that, since the shear stress component is more difficult to predict than the gravitational one, the absolute percentage deviation between the calculated and the experimental data $|e_i|$ increases with G_{steam} , from 1 % at $1.5 \text{ kg m}^{-2} \text{s}^{-1}$ to 6.5 % at $7.5 \text{ kg m}^{-2} \text{s}^{-1}$.

3.7 Modification of wetting properties in aluminum surfaces

The effect of the wetting properties of the metallic surface on the heat transfer coefficient during pure steam condensation were investigated by modifying the wetting characteristics of the aluminum substrates through proper chemical processes, in order to obtain superhydrophilic, hydrophobic and superhydrophobic samples. In this Section the experimental techniques used to change surface wettability and the wetting properties characterization methods are described.

3.7.1 Materials

A high purity (AW 1050, minimum Al quantity 99.50%) aluminum plate was used for specimen fabrication.

Iron(III) Chloride (reagent grade 97%), Hydrogen Peroxide (30% w/w solution), Hexane (anhydrous, 95%), Tetrahydrofuran ($\geq 99.9\%$) and 1H,1H,2H,2H-Perfluorooctyltriethoxysilane (98%) were purchased by Sigma Aldrich.

Sodium Hydroxide pellets and Polydimethylsiloxane (Sylgrad 184) were obtained by Merck and Dow Corning, respectively.

3.7.2 Superhydrophilic sample preparation

The superhydrophilic sample is obtained mainly by chemical etching through wet immersion processes into different corrosive solutions.

Prior to any treatment, the sample is immersed for 15' into Isopropanol (IPA), continuously stirring the solution through a sonication probe, and dried into a nitrogen stream for cleaning purposes. After cleaning, the sample is immersed in a NaOH (1% w/w) aqueous solution for 15', to get rid of the superficial aluminum oxide. When extracting the sample from the solution this is dipped in IPA and deionized (DI) water to stop the reaction, before drying with N_2 .

For getting the proper superficial roughness, a $FeCl_3$ – water (1 mol L^{-1}) corrosive solution is used. The specimen is immersed into the etching mixture for 7.5', and subsequently dipped into IPA and DI water before drying with nitrogen.

Finally, for controlled re-oxidation of the surface, the metallic specimen is immersed for 30' into a H_2O_2 (30% w/w) aqueous solution, cleaned with DI water and dried in N_2 .

3.7.3 Hydrophobic sample preparation

The hydrophobic sample is prepared through a functionalization process which reduces the specimen surface free energy, forming a Self Assembled Monolayer (SAM) over it.

The process consists in spin-coating a n-Hexane – Perfluorooctyltriethoxysilane (FOTS) mixture (5% by volume) onto the sample at 800 rpm for 30 seconds. After the spin coating process, the sample is baked at 150°C for 30 minutes for final solvent evaporation and SAM stabilization.

3.7.4 Superhydrophobic sample preparation

The superhydrophobic sample is prepared combining proper superficial roughness with low surface free energy.

Proper morphology is obtained through the cleaning, de-oxidation, etching and re-oxidation steps used for getting the superhydrophilic properties, as presented in Section 3.7.2. After that, surface

free energy is reduced forming a SAM over the substrate, with the same technique shown in Paragraph 3.7.3.

After the second (functionalization) step the sample is superhydrophobic, presenting very high advancing contact angle and very low contact angle hysteresis. However, it has been decided to introduce a third step, with the goal to enhance the resistance of the superhydrophobic properties in the presence of condensing vapor.

As shown in Zhou *et al.* (2012), Polydimethylsiloxane (PDMS) can be used in combination with fluorinated silane reagents and functionalized substrates to obtain durable superhydrophobic coatings. Inspired by that, the superhydrophobic aluminum sample has been further treated with an additional step, which consists in dip coating the substrate in a PDMS – FOTS – THF (Tetrahydrofuran) solution. The solution is prepared by combining 50 mg of PDMS (+10% curing agent) and 100 μL of FOTS for each 10 mL of THF. The sample is dip coated inside the solution at $100\text{ }\mu\text{m s}^{-1}$ before curing at 135°C for 30 minutes.

At the end of the last step the sample presents durable superhydrophobic characteristics.

3.7.5 Characterization of the surface

Surface wetting properties and morphological characteristics were analyzed by means of contact angles analysis as well as by Scanning Electron Microscope and Atomic Force Microscopy visualization.

A setup for contact angles measurement has been set up at the Two-Phase Heat Transfer Laboratory of the Department of Industrial Engineering of Padova University. This consists of a compact CMOS camera (Thorlabs GmbH ® DCC1545M) combined with a Thorlabs GmbH ® MVL7000 zoom lens and a LED light. Typical spatial and time resolution of the system are $5\text{ }\mu\text{m/pixel}$ and 30 fps respectively.

Contact angles are measured using the standard sessile drop method, recording a water drop (backlight illuminated by the LED) expanding and contracting quasi-statically over the horizontally oriented surface of interest. Advancing (θ_{adv}) and receding (θ_{rec}) contact angles are evaluated by post-processing the videos fitting with a circle the drop profile near the contact point. Figure 3.7.5.1 shows an example of contact angle measurement.

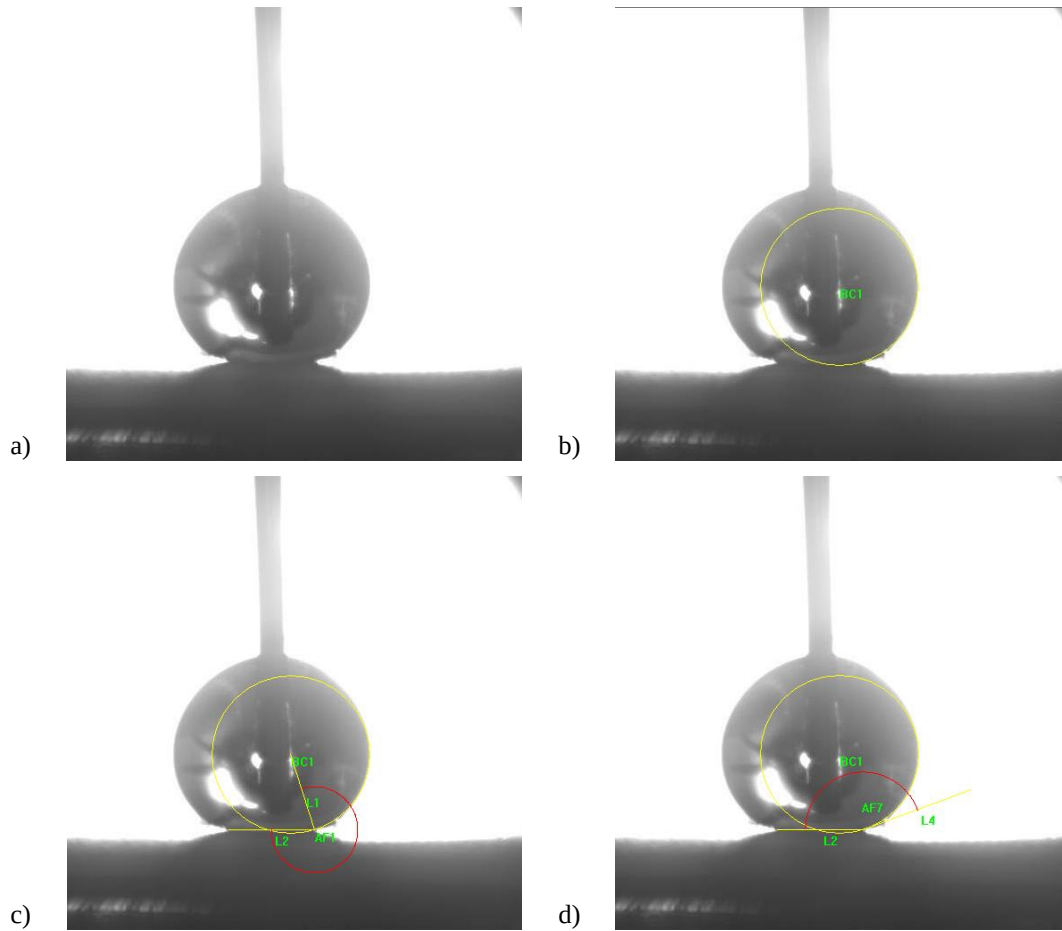


Figure 3.7.5.1. Example of contact angle characterization. As a beginning, a frame of the moving drop is extracted from the video (a). Then the circle fitting the droplet profile near the contact point is created (b). Subsequently, the angle between the circle radius and the contact line is measured via software (c). Finally, the contact angle is geometrically evaluated (d).

Contact angle hysteresis ($\Delta\theta$) is calculated as the difference between θ_{adv} and θ_{rec} . Values of the contact angles reported in this Thesis are the average between at least five measurements for each sample in each condition, and the corresponding standard deviation is reported as experimental uncertainty.

Prior to any treatment, the as-received aluminum sample presents an advancing contact angle equal to $84.8^\circ \pm 4.4^\circ$, a receding contact angle equal to $17.4^\circ \pm 4.6^\circ$ and a contact angle hysteresis equal to $67.3^\circ \pm 4^\circ$.

After the etching process the roughened superhydrophilic sample presents $\theta_{adv} = 12.3^\circ \pm 2.5^\circ$, $\theta_{rec} = 5.2^\circ \pm 4.2^\circ$ and $\Delta\theta = 6.9^\circ \pm 3.6^\circ$.

The functionalized-only hydrophobic sample is characterized by an advancing contact angle equal to $143.5^\circ \pm 2.6^\circ$, a receding contact angle equal to $43^\circ \pm 4.8^\circ$ and $\Delta\theta = 100.5^\circ \pm 3.4^\circ$.

Finally, at the end of the three-steps process, the superhydrophobic treated aluminum surface exhibits $\theta_{adv}=152.4^{\circ} \pm 5.7^{\circ}$, $\theta_{rec}=148.3^{\circ} \pm 6.9^{\circ}$ and $\Delta\theta=4.1^{\circ} \pm 3.6^{\circ}$.

Figure 3.7.5.2 shows examples of the advancing and receding angles for the different specimens.

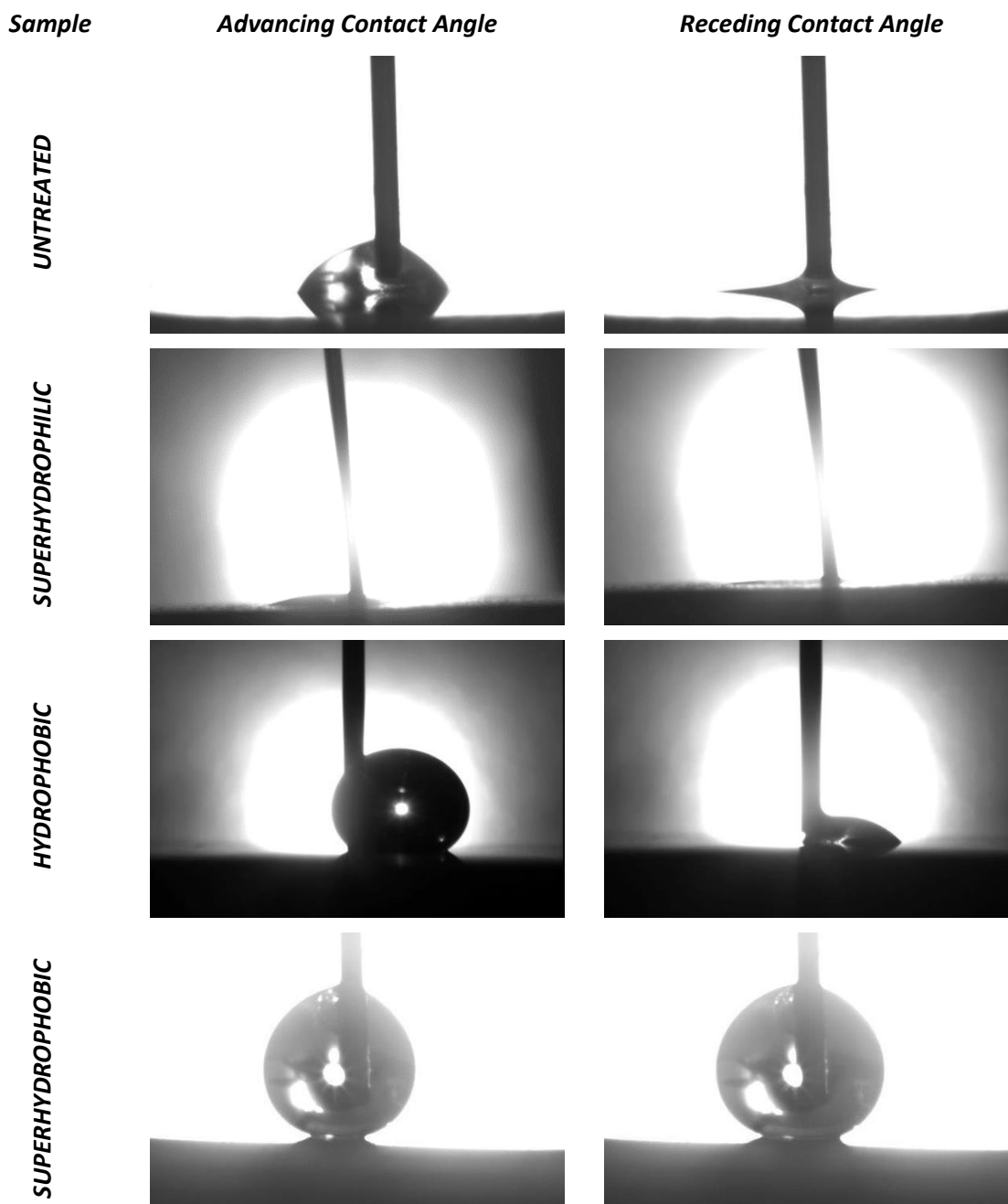


Figure 3.7.5.2. Examples of advancing and receding contact angles for the four specimens.

Surface morphology is visually characterized by digital field emission Scanning Electron Microscope (SEM) as well as by Atomic Force Microscopy (AFM).

SEM is mostly used for scanning the surface of a sample by means of an electron beam which is scanned over the surface of the specimen. This must be electrically conductive, in order to avoid

surface charging which causes instability and decreases the resolution. If the sample is an electric insulator, the surface is coated with a thin layer of gold or carbon. The interaction of the electron beam with the atoms produces signals that are detected by a detector and the varying intensity of the signal is reproduced as an image on the screen.

In the following Figures Scanning Electron Microscope images of the as-received and the roughened samples are shown. Since functionalization step creates a Self Assembled Monolayer, which has atomic-layer thickness, over the surface, the morphology of the hydrophobic sample is identical to the one of the quasi-smooth untreated specimen. Moreover, since the dip-coating process deposits an extremely thin (few nanometers) and uniform layer over the substrate, no appreciable differences on the morphology of the sample before and after the dip-coating step were seen. Because of this, both for SEM and AFM analysis, only the morphological characteristics of the as-received and the roughened sample are reported hereinafter.

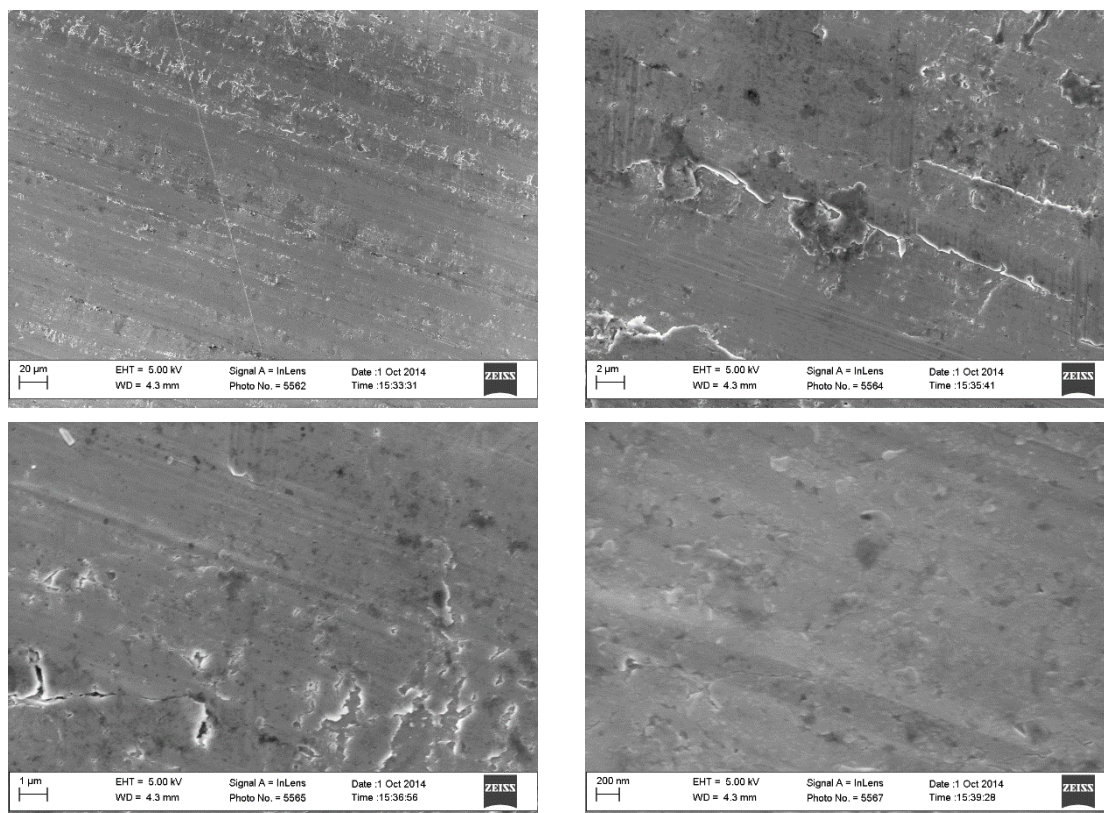


Figure 3.7.5.3. SEM images of the as-received sample, at different magnifications (1000x, 10000x, 20000x and 80000x).

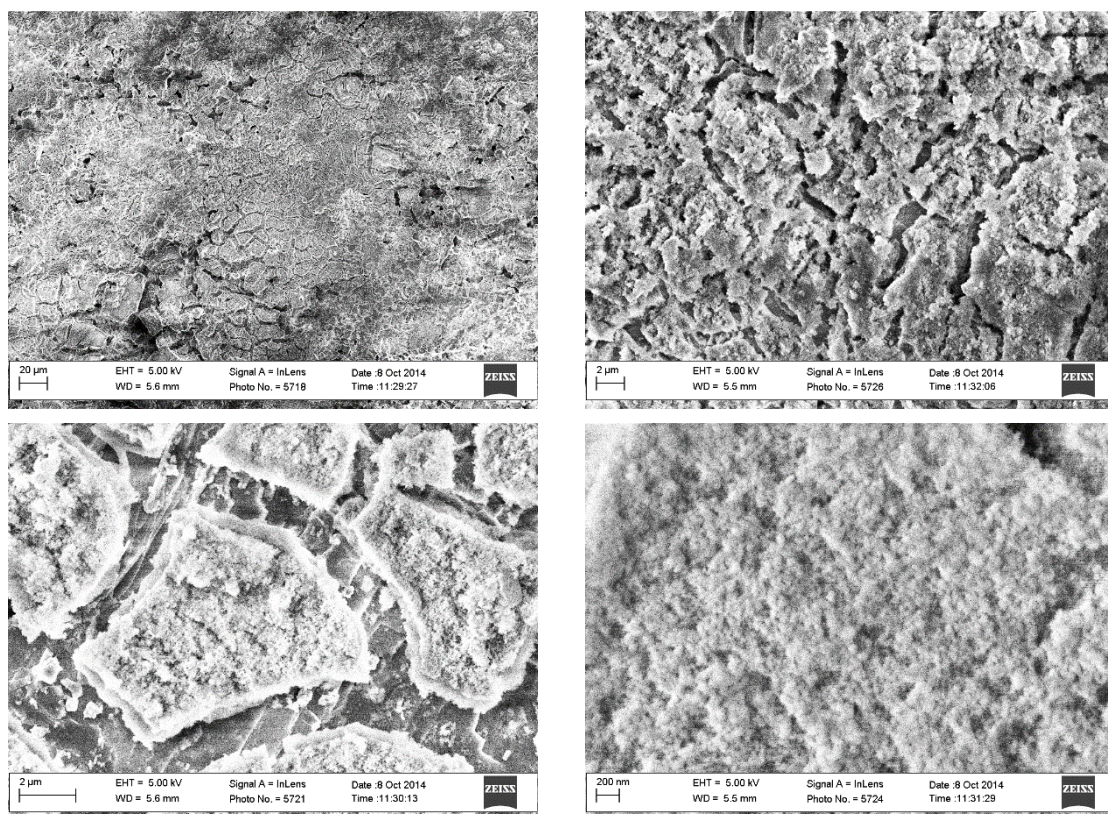


Figure 3.7.5.4. SEM images of the roughened sample, at different magnifications (1000x, 10000x, 20000x and 80000x).

From Figures 3.7.5.3 and 3.7.5.4 it is possible to see that the etching process imparts a drastic change in the surface morphology. The corrosion process is highly non-isotropic and it forms a porous superficial structure characterized by blocks having a polygonal shape. Moreover, looking at the high-magnification image, it is possible to see that the faces of the blocks present a second-level morphology, in the nano-scale magnitude, which gives to the walls of the polygons a “papier-mâché” appearance. This double-scale morphology is the reason of the extreme hydrophilicity of the only-roughened sample and of the extreme water repellency of the three-steps treated specimen.

For better showing the morphological base structure of the roughened sample, SEM images at different magnifications of the etched surface before the re-oxidation step (which forms the “papier-mâché” morphology by the deposition of an oxide layer over the structures, which partially covers them reducing the clarity of the SEM images) are reported in Figure 3.7.5.5. In the Figure the crystalline structure exposed on the aluminum surface by the FeCl_3 corrosion step is shown.

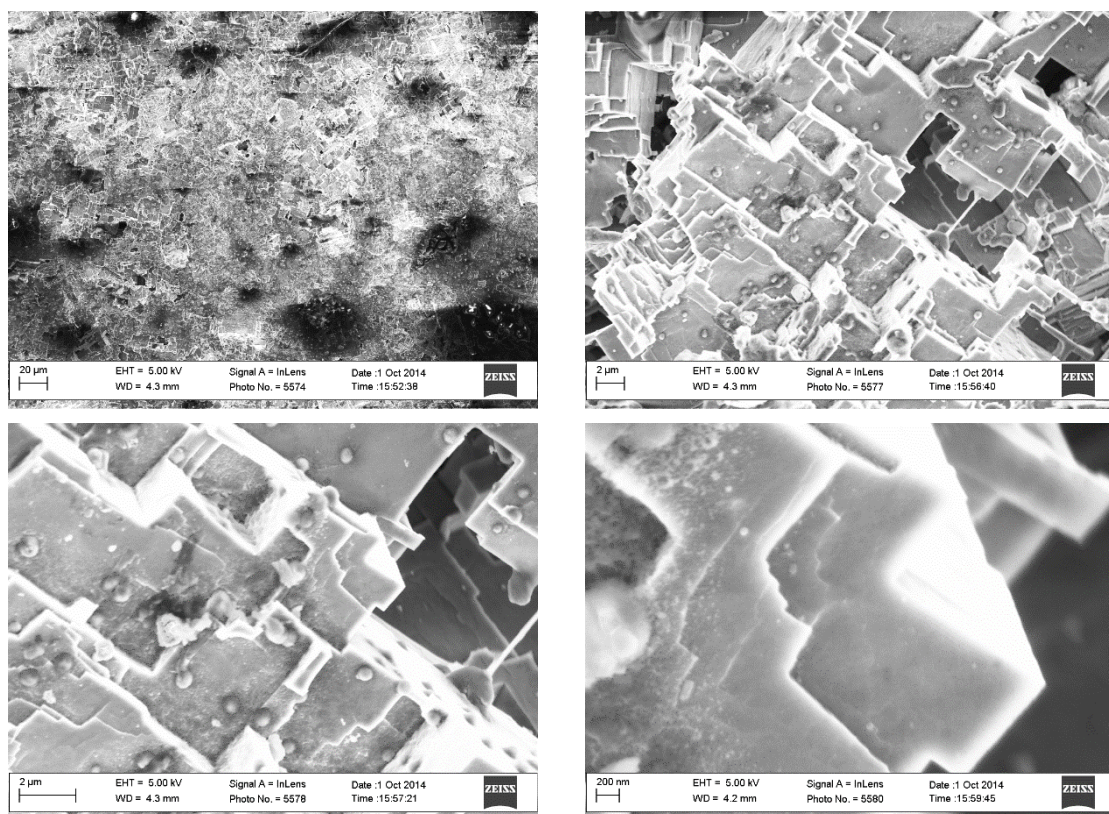


Figure 3.7.5.5. SEM images of the etched sample before the re-oxidation step, at different magnifications (1000x, 10000x, 20000x and 80000x).

AFM is capable of imaging surfaces with atomic resolution (0.1 – 0.2 nm) of both conductive and nonconductive materials. It measures the interactive force between the atoms of the probe and those of the specimen. A sharp tip is mounted on a microcantilever arm and scanned across the sample surface. The deflection of light off the back of the arm is used to monitor the force applied on the specimen. Typically, the forces applied to the specimen are of the order 10^{-9} N, which is so small that it ensures no specimen damage. Traditionally, the sample is mounted on a piezoelectric tube, that can move the sample in the z direction for maintaining a constant force, and the x and y directions for scanning the sample. In tapping mode AFM, the topographic features of the image and the spatial variation of the surface is mapped by phase imaging. This technique operates by detecting the phase shift associated with the probe's resonance and its proximal interaction with the sample.

In the following Figure 3D-reconstructions, obtained through the AFM analysis, of the morphology of the specimens before and after the roughening process are shown.



Figure 3.7.5.6. AFM images of the as-received (left) and the roughened (right) specimens.

From the AFM images it is possible to define the average line of the specimens profile, reported for the two samples in Figure 3.7.5.7.

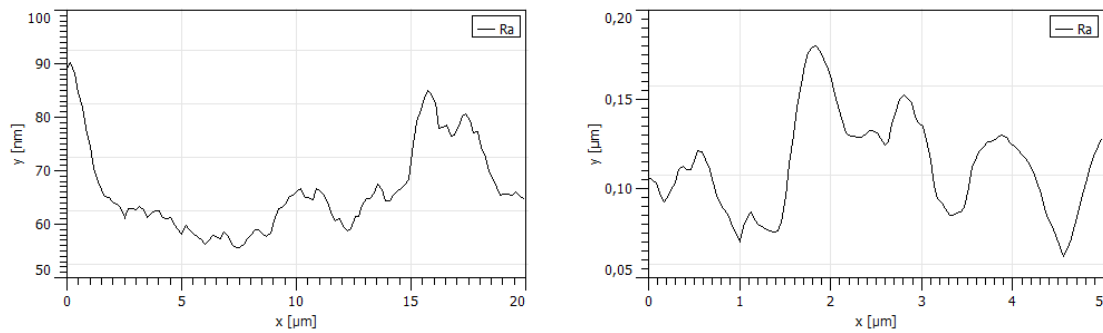


Figure 3.7.5.7. Profile lines for the as-received (left) and the roughened (right) specimens.

By looking at the sample profile it is finally possible to evaluate the arithmetical mean deviation of the assessed profile

$$R_a = \frac{1}{L} \int_0^L |y| dx \quad (3.28)$$

The as-received surface resulted to have an average roughness equal to 72.6 ± 25.4 nm while the nano-engineered one resulted to have $R_a = 282.9 \pm 47.3$ nm.

3.8 Condensation over engineered surfaces

In this Section, analysis of condensation mode and heat transfer coefficient over engineering treated surfaces exhibiting superhydrophilic, superhydrophobic and hydrophobic properties is reported.

3.8.1 FWC over superhydrophilic surfaces

Figure 3.8.1.1 reports the measured heat flux and the corresponding condensation heat transfer coefficient as a function of the wall subcooling, at different steam mass fluxes, for the superhydrophilic treated specimen.

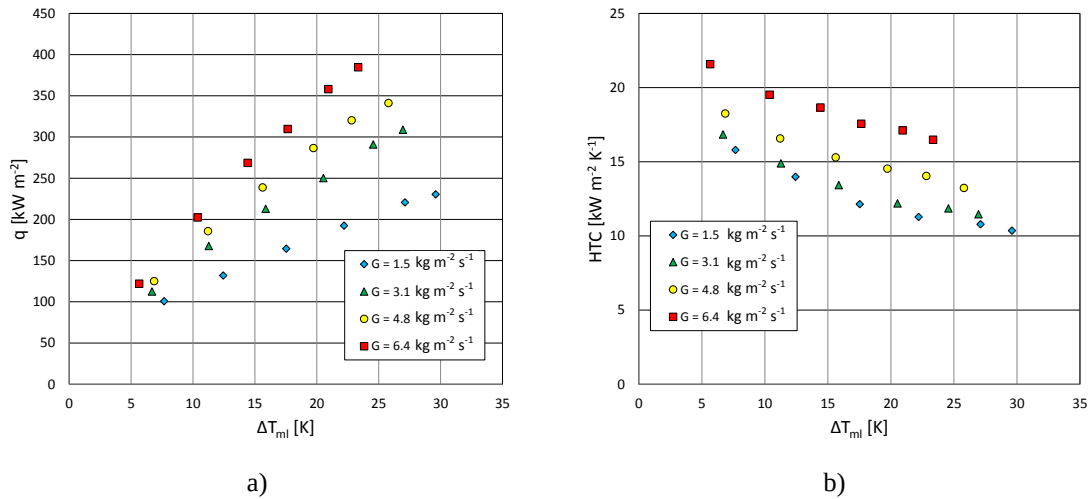


Figure 3.8.1.1. Heat flux (a) and heat transfer coefficient (b) versus mean logarithmic temperature difference between the saturated vapor and the surface during condensation over the superhydrophilic sample. Data refer to different steam mass velocities G [$\text{kg m}^{-2} \text{ s}^{-1}$].

The behavior of the superhydrophilic sample, in terms of trends with ΔT_{ml} and G_{steam} , is the same of the untreated one. Heat transfer coefficient enhances increasing steam mass velocity and decreasing saturation-to-wall temperature difference.

Figure 3.8.1.2 shows the ratio between the heat transfer coefficient obtained with the superhydrophilic and the untreated sample for two steam mass velocities ($1.5 \text{ kg m}^{-2} \text{ s}^{-1}$ and $6.5 \text{ kg m}^{-2} \text{ s}^{-1}$, respectively) as a function of the mean logarithmic temperature difference between the vapor and the wall.

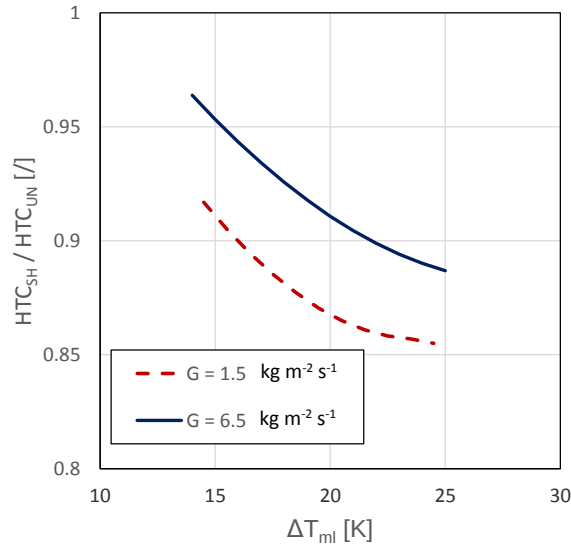


Figure 3.8.1.2. Ratio between the heat transfer coefficient on the superhydrophilic sample (HTC_{SH}) and the one on the untreated sample (HTC_{UN}) at $G_{steam} = 1.5 \text{ kg m}^{-2} \text{ s}^{-1}$ and at $G_{steam} = 6.5 \text{ kg m}^{-2} \text{ s}^{-1}$.

The comparison between the untreated and the superhydrophilic sample shows that the roughened surface presents lower heat transfer coefficient, thus lower heat flux, at the same G_{steam} and ΔT_{ml} . This is due to the different wettability of the two samples. On the superhydrophilic specimen, the condensate is strongly attached on the substrate, thus gravity influence and shear stress influence on the liquid film thickness are reduced in comparison to a quasi-smooth surface. This is especially true at low vapor mass velocities, when the gravity force is dominant. When increasing the steam velocity, the shear stress becomes more relevant and it is able to overcome the negative effect of the high adhesion forces.

From this analysis it is possible to infer that condensation over a roughened surface which presents high wettability is less efficient than the process on a quasi-smooth surface with water contact angle around 90° .

3.8.2 DWC over superhydrophobic surfaces

In this Paragraph analysis of pure steam condensation over the three-steps treated aluminum substrate is presented.

When performing heat transfer analysis over the superhydrophobic sample it turned out that this was able to promote dropwise condensation mode, but that the sample was able to sustain the two-phase enhanced process only for a few minutes, over which condensation mode moved back from dropwise to filmwise. The transition between the two processes was acquired and it is

presented in the following Figure, which shows the profiles of the four temperatures measured inside the specimen (named as in Sec. 3.5.3) as a function of the acquisition time.

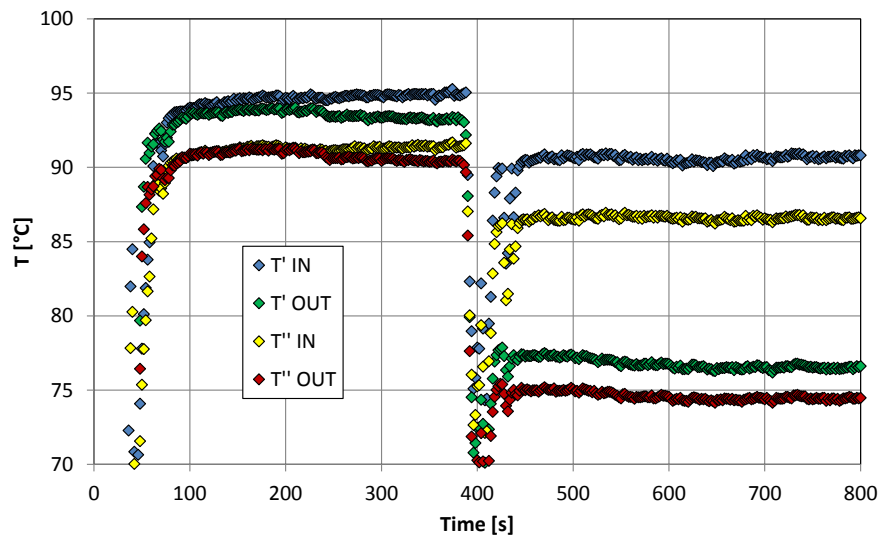


Figure 3.8.2.1. Temperature profiles inside the superhydrophobic specimen during the transition between dropwise and filmwise condensation mode. Saturation temperature is about 106°C, steam flows at about 6.5 kg m⁻² s⁻¹.

When condensing in dropwise mode specimen temperatures are very close to steam temperature and close to each other, for the same orthogonal position z . Looking at the Figure 3.8.2.1 it is possible to see a drastic reduction of the specimen temperatures, which is connected with the degradation of the condensation mode, at about 380 s. After a transition period of about 40 s the condensation mode stabilizes to filmwise, as it can be seen by the fact that the temperatures inside the specimen stabilize to lower values than those before the transition. Moreover, after 420 s the specimen presents a high temperature gradient between the inlet and the outlet zones. This temperature variation is related to the presence of the condensate film over the surface, which is thicker closer to the sample outlet, leading to a non-constant thermal resistance along the specimen which causes the temperature reduction.

This behavior is typical of filmwise condensation mode, as shown in Figure 3.8.2.2, where the superficial temperature difference between the inlet and the outlet of the sample as a function of the removed heat flux, for the heat transfer tests performed during FWC over the untreated surface (Sec. 3.6.1), is reported.

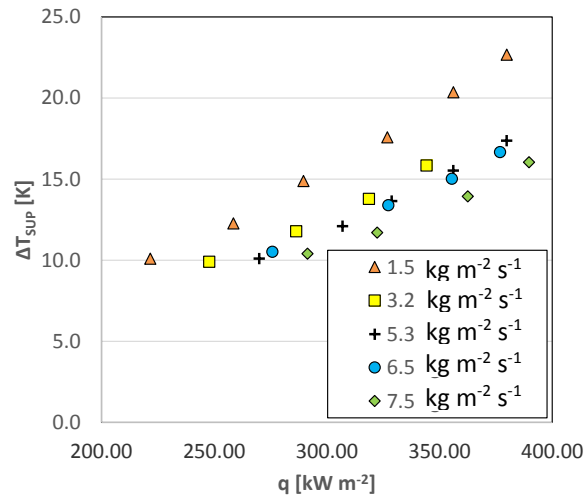


Figure 3.8.2.2. Superficial temperature difference between top and bottom as a function of the removed heat flux for the filmwise condensation tests over the untreated sample.

As it can be seen from the Figure, the presence of the liquid film leads to a high temperature gradient over the specimen surface, the higher the lower is the steam mass velocity and the higher is the exchanged heat flux, i.e. the more the thicker is the condensate film.

This is not the case when condensing in dropwise mode. Surface temperatures at the inlet and the outlet of the specimen are quite close to each other, leading to a smaller mean logarithmic temperature difference between the steam and the wall, thus to a higher heat transfer coefficient. This is shown in Figure 3.8.2.3, which reports the mean logarithmic temperature difference between the steam and the wall and the corresponding heat transfer coefficient during the transition between dropwise and filmwise condensation mode.

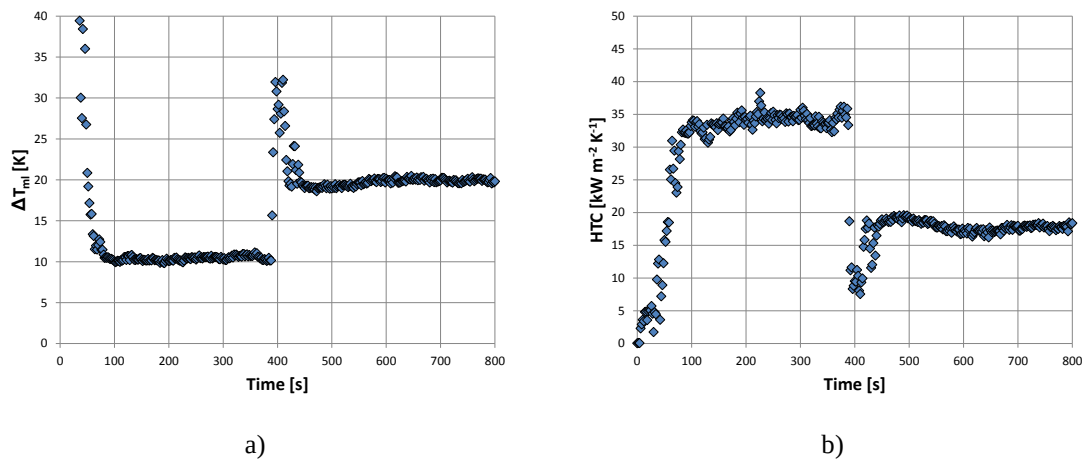


Figure 3.8.2.3. Mean logarithmic temperature difference between the steam and the wall (a) and heat transfer coefficient (b) as a function of the acquisition time during the transition between dropwise and filmwise condensation mode. Saturation temperature is about 106°C, steam flows at about 6.5 kg m⁻² s⁻¹.

The Figure shows that when condensing in dropwise mode the steam-to-wall temperature difference is much lower than the one during FWC, which reflects in a much higher heat transfer coefficient.

Removed heat flux remains almost constant between the two condensation regimes. This is because the main thermal resistance to the heat transport is located on the convective and conduction sides of the process, thus the heat flux is not substantially influenced by a variation of the two-phase heat transfer coefficient.

However, the superhydrophobic treated substrate was not able to sustain high-HTC dropwise condensation mode for more than a few minutes, over which a condensate film formed on the surface. This was not because of a degradation of the superhydrophobic properties of the sample, since when taken out from the setup it still presented high advancing contact angle and low contact angle hysteresis. The reason of the degradation of the condensation mode was probably related to the kind of micro-/nano- scale porous morphology created by the etching process.

When condensation of pure steam occurs over the engineered sample, the vapor located in between the nano-porosities condenses in contact with the colder walls, trapping condensate in between the surface micro-structures. With a superficial roughness like the one showed in Figure 3.7.5.4 the surface is not able to dispose the condensate in between the structures because of capillary forces, thus a droplet forming onto the surface is anchored onto a mixed solid-liquid layer, thus it presents high adhesion forces. Because of gravity and shear forces, at the beginning of the process droplets are removed from the surface, thus condensation occurs in dropwise mode. After a certain time, drops on the surface grow and coalescence because of hysteresis related to the high droplet adhesion, which slows the removal process, until they merge in a liquid film over the wall, causing the degradation of the two-phase process.

Because of this, despite its extremely high water repellency properties, the superhydrophobic surface presented in Section 3.7.4 is not adequate for stable dropwise condensation promotion. In addition to that, after dropwise to filmwise transition occurs, the superhydrophobic surface performs as a superhydrophilic one, since the condensate layer is strongly pinned on the roughened surface. Thus, after the degradation of the condensation mode, the surface performs worse than an untreated one.

This analysis shows that there is no direct connection between contact angles and condensation mode. When condensing in the absence of air or other NCG, a surface with extremely high contact angles but non-adequate morphology will induce filmwise condensation mode, despite its high

water repellency properties in open-atmosphere environments, because of droplets pinning on the rough substrate.

Even if DWC was sustained for a limited period of time, a visual analysis of the phenomena has been undertaken. Figure 3.8.2.4 captures dropwise condensation of steam over the superhydrophobic sample at $G_{steam} \approx 6.5 \text{ kg m}^{-2} \text{ s}^{-1}$ and $\Delta T_{ml} \approx 10 \text{ K}$, showing droplets nucleation, growing, coalescence, and departure cycle.

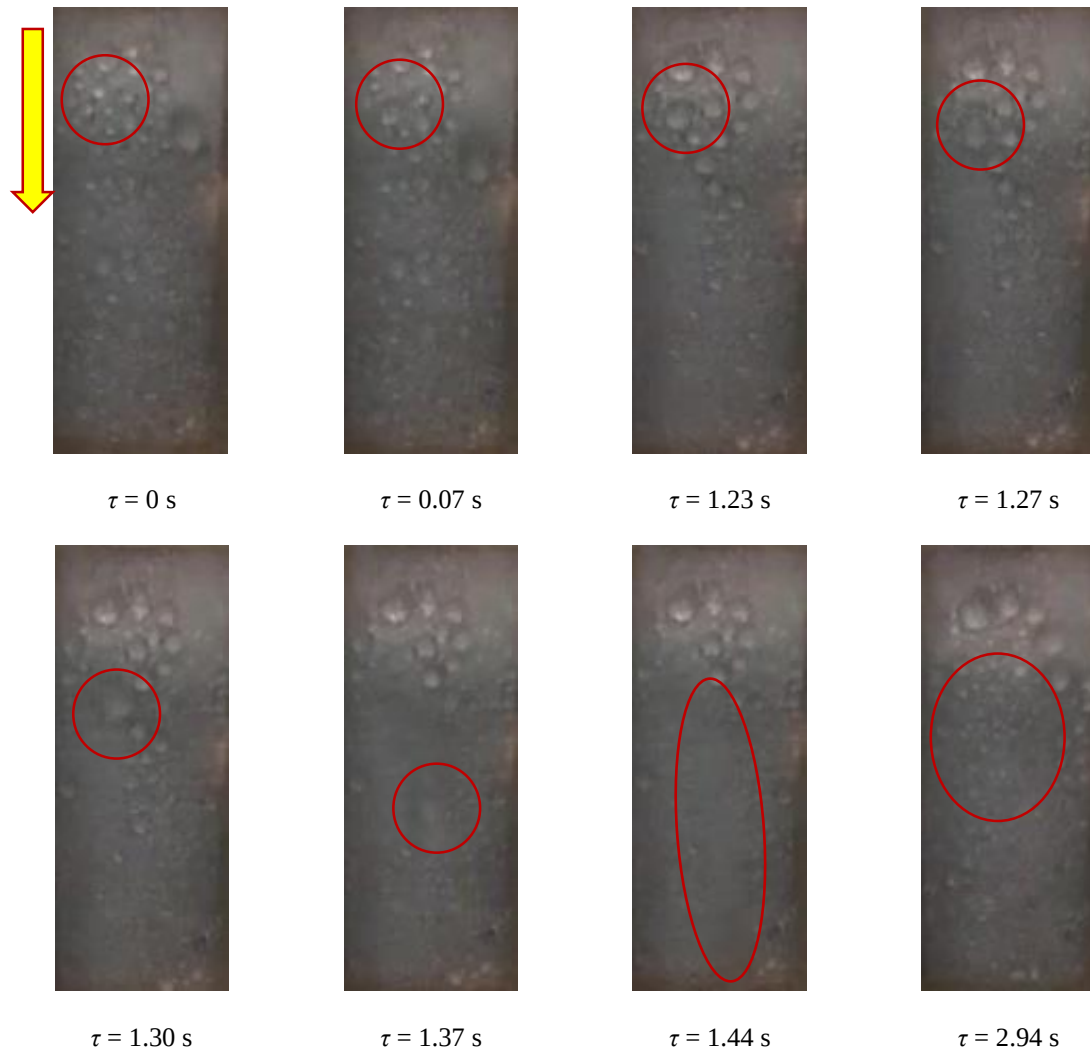


Figure 3.8.2.4. Droplets cycle on a superhydrophobic aluminum sample at about 10 K of surface subcooling and about $6.5 \text{ kg m}^{-2} \text{ s}^{-1}$ of vapor mass velocity. The arrow indicates the vapor flow direction.

The cycle starts with nucleation of small drops, which form and grow over the surface ($\tau = 0 \text{ s}$), and coalescence generating bigger droplets ($\tau = 0.07 \text{ s}$). New drops continue to grow, due to continuous vapor condensation, eventually absorbing other small droplets, until they reach a critical diameter ($\tau = 1.23 \text{ s}$), over which they move from the surface because of combined gravity

and shear stress forces ($\tau = 1.27$ s). The departing drops swept over the surface ($\tau = 1.27 - 1.44$ s), thereby wiping small droplets in their path. After the drop sweep, fresh drops start to nucleate ($\tau = 2.94$ s) and grow again, allowing the cyclic process to continue.

3.8.3 Enhanced FWC over hydrophobic surfaces

Heat flux and heat transfer coefficient data acquired during filmwise condensation over the hydrophobic treated aluminum substrate are reported in Figure 3.8.3.1. As for the other surfaces, tests were performed varying steam-to-wall mean logarithmic temperature difference and vapor specific mass flow rate.

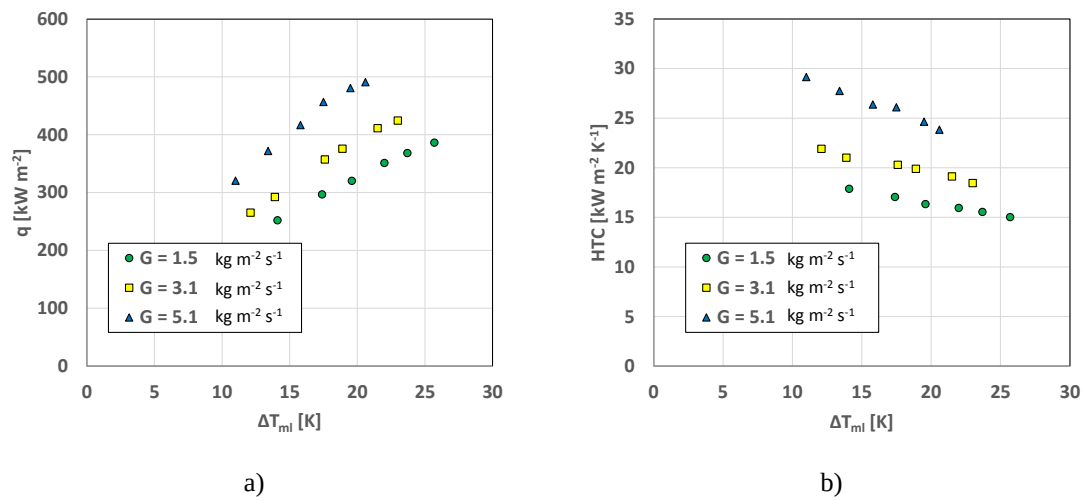


Figure 3.8.3.1. Heat flux (a) and heat transfer coefficient (b) versus mean logarithmic temperature difference between the saturated vapor and the surface during condensation over the hydrophobic sample. Data refer to different steam mass velocities G [$\text{kg m}^{-2} \text{ s}^{-1}$].

The trend is the one expected. Heat flux increases both when enhancing saturation-to-wall temperature difference and steam mass flow rate. The heat transfer coefficient reduces when enhancing ΔT_{ml} , while it is strongly enhanced by an augmentation of the steam velocity.

Figure 3.8.3.2 shows the comparison between the results obtained with the hydrophobic and the untreated samples. Figure a) shows the comparison between the heat transfer coefficient on the hydrophobic sample (HY) and the one on the untreated specimen (UN) as a function of the saturation to wall temperature difference, while Figure b) shows the ratio between the two. Both graphs refer to three different vapor mass velocities ($1.5 \text{ kg m}^{-2} \text{ s}^{-1}$, $3.1 \text{ kg m}^{-2} \text{ s}^{-1}$, $5.1 \text{ kg m}^{-2} \text{ s}^{-1}$).

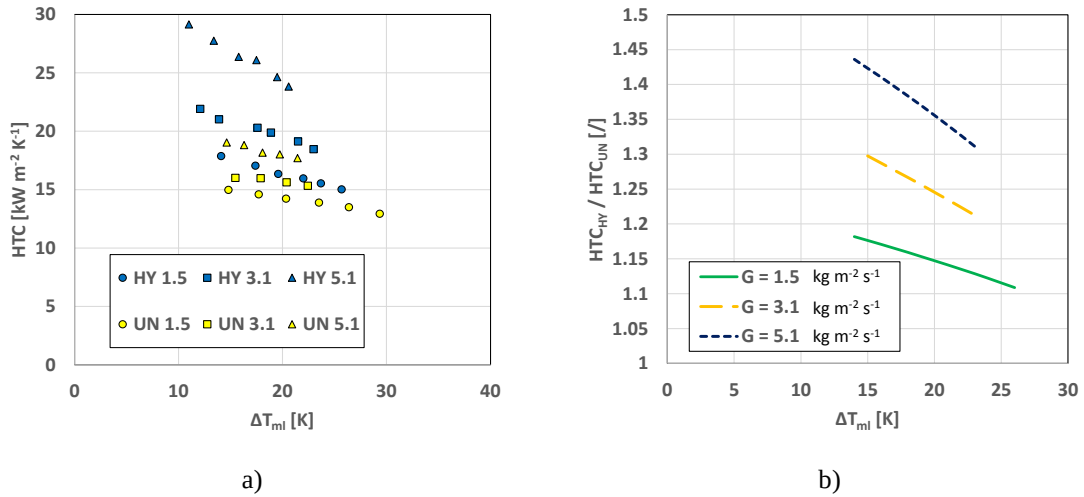


Figure 3.8.3.2. Comparison between the heat transfer coefficient on the hydrophobic (HY) sample and the one on the untreated (UN) specimen (a), and ratio between the two (b), as a function of the mean logarithmic temperature difference between the steam and the wall. Data refer to three steam mass velocities G ($1.5 \text{ kg m}^{-2} \text{ s}^{-1}$, $3.1 \text{ kg m}^{-2} \text{ s}^{-1}$ and $5.1 \text{ kg m}^{-2} \text{ s}^{-1}$).

The Figure shows that when condensation occurs over the hydrophobic treated specimen higher heat transfer coefficients are achieved, comparing to the case of a standard surface. The enhancement is higher the higher is the vapor specific mass flow rate and lower the higher the saturation to wall temperature difference. Moreover, it can be seen that the slope of the heat transfer coefficients ratio tendency lines varies when varying steam mass velocity. The higher the vapor mass flow rate the more the heat transfer coefficient enhancement is sensitive to a variation of the wall subcooling degree.

It can thus be inferred that the heat transfer coefficient enhancement is strongly dependent on the liquid film thickness. When the condensate layer increases the associated thermal resistance enhances and dominates the positive effect related to the hydrophobic properties of the surface. In the present conditions, heat transfer coefficient enhancement between the hydrophobic and the untreated surfaces stands between 10% and 20% at G_{steam} equal to $1.5 \text{ kg m}^{-2} \text{ s}^{-1}$, between 20% and 30% at G_{steam} equal to $3.1 \text{ kg m}^{-2} \text{ s}^{-1}$ and between 30% and 45% at G_{steam} equal to $5.1 \text{ kg m}^{-2} \text{ s}^{-1}$.

The question that arises from the experimental analysis herein presented is why on hydrophobic surfaces it is possible to get enhanced heat transfer coefficients, maintaining a filmwise condensation mode? A detailed analysis about this phenomenon is still in progress, but some evaluations have been made by looking at investigations presented in the literature for similar phenomena.

Enhancement of condensation heat transfer can be explained by considering slipping of the condensate layer over the treated surface, because of its water repellency properties. Slip driven condensation over hydrophobic surfaces has not been investigated in the open literature, to the best of the author's knowledge, but several works relate hydrophobic properties of the substrate to fluid slipping, as well as condensation heat transfer augmentation to the introduction of slipping boundary conditions at the solid-condensate interface.

The slip length is defined as the distance behind the interface at which the liquid velocity extrapolates to zero (Pati *et al.*, 2013). Several works relate liquid repellency properties of a substrate with slipping of the fluid over it, due to an enhancement of the slip length, which from the fluid-dynamics point of view can be seen as the introduction of a nonzero slip velocity at the solid-liquid interface (Tretheway and Meinhart, 2004; Voronov *et al.*, 2007; Hsieh and Lin, 2009). To the best of the author's knowledge, no experimental or theoretical analysis about slip driven condensation over nano-engineered surfaces has been presented in the literature, but several computational works relate the presence of slipping boundary conditions at the solid-liquid interface (nonzero slip length, i.e. nonzero slip velocity) to heat transfer performance enhancement during vapor condensation.

Pati *et al.* (2013) theoretically studied slip-driven alteration in filmwise condensation over vertical surfaces. The authors stated that, even if studies on film condensation heat transfer essentially comply with no-slip boundary conditions at the solid surface, under certain circumstances slip between the liquid and the solid surfaces may occur, because of several phenomena (including chemical interaction between the condensate and the substrate). The authors studied the relationship between slippage phenomenon and two-phase heat transfer performances by introducing nonzero slip lengths into the classical Nusselt theory. The authors demonstrated that when considering a slip length $\beta \neq 0$ condensate film thickness over the surface reduces, in comparison to classical Nusselt theory ($\beta = 0$), the more the higher is the slip length. This is because an increase of β is associated with an augmentation of the slip velocity at the surface, which reduces the velocity gradient at the surface and hence the wall shear stress, resulting in a thinner condensate layer. The reduction of the liquid film thickness is associated with a reduced thermal resistance of the condensate layer, thus it leads to an enhancement of the two-phase heat transfer coefficient. Figure 3.8.3.3, taken from Pati *et al.* (2013), shows the variation of the local Nusselt number along the longitudinal distance for different slip lengths β .

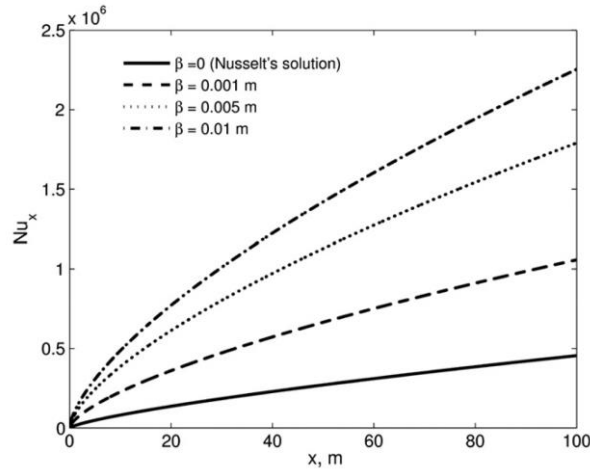


Figure 3.8.3.3. Variation of the local Nusselt number with the longitudinal distance for different values of the slip length β . Graph from Pati *et al.* (2013).

Similar results are shown in Al-Jarrah *et al.* (2008), where variation of the velocity along the liquid layer, of the condensate film thickness and of the condensation mass flow rate per unit width are reported for different slip lengths. The authors showed that a nonzero slip length reflects on a nonzero slip velocity at the solid-liquid interface, leading to reduced condensate film thickness and enhanced condensation mass flow rate per unit width.

The present database reflects the theory presented above. It can be inferred that slipping conditions are achieved at the solid-condensate interface, because of water repellency properties of the surface, which induce a nonzero slip length. As a result, an enhancement on the condensation heat transfer coefficient is obtained, due to reduced liquid film thickness.

Finally, with the purpose of confirming the theory stated above, classical condensation theory has been modified introducing a nonzero velocity boundary condition at the solid-liquid interface, comparing the results with the new experimental database. A detailed description of the phenomena is hereinafter reported.

3.8.4 Theoretical analysis of slip-driven FWC

Nearly two hundred years ago Navier proposed a general boundary condition that incorporates the possibility of fluid slip over a solid wall, assuming that the velocity at the solid liquid interface, i.e. the slip velocity u_s , is proportional to the shear stress at the surface

$$u_s = \beta \frac{\partial u_y}{\partial z} \quad (3.29)$$

naming z the axis orthogonal to the condensation surface and y the one that follows the fluid flow direction. Physical meaning of Eq. (3.31) is shown in Figure 3.8.4.1.

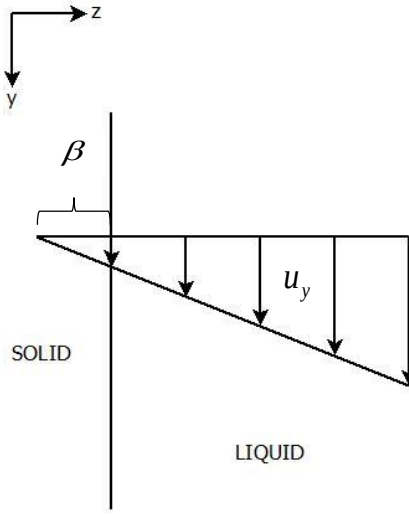


Figure 3.8.4.1. Slip length β and velocity profile over a surface.

As previously said, if $\beta = 0$ then no-slip boundary conditions are assumed while if $\beta > 0$ a nonzero slip velocity can be derived from Eq. (3.29). Different attempts were made to estimate the slip velocity during single phase flow between two infinite parallel plates, while no data concerning two phase flow have been found in the open literature. Because of this, single phase studies were taken as a reference for the evaluation of u_s in the present analysis.

Slip-driven filmwise condensation theory has been studied introducing nonzero velocity boundary conditions in the classical theories of gravity and shear controlled condensation heat transfer.

If condensation occurs in the absence of shear stress forces the velocity gradient along the condensate film is

$$\frac{\partial u}{\partial z} = \frac{(\rho_l - \rho_v)g}{\mu_l}(\delta - z) \quad (3.30)$$

Thus it is possible to write

$$u = \frac{(\rho_l - \rho_v)g}{\mu_l} \left(\delta z - \frac{z^2}{2} \right) + u_s \quad (3.31)$$

with the assumption that at $z = 0$ it is $u = u_s$.

The mean velocity along the liquid film is then evaluated as

$$\bar{u} = \frac{1}{\delta} \int_0^\delta u \cdot dz = \frac{(\rho_l - \rho_v)g}{3\mu_l} \delta^2 + u_s \quad (3.32)$$

The condensate mass flow rate per unit width is

$$\Gamma = \bar{u} \delta \rho_l \quad (3.33)$$

leading to a variation of Γ along the surface equal to

$$\frac{d\Gamma}{dy} = \frac{\rho_l(\rho_l - \rho_v)g}{\mu_l} \delta^2 \frac{d\delta}{dy} + \rho_l u_s \frac{d\delta}{dy} \quad (3.34)$$

Imposing

$$\frac{d\Gamma}{dz} = -\dot{m} = \frac{-q}{H_{lv}} = \frac{\rho_l(\rho_l - \rho_v)g\delta^2}{\mu_l} \left(\frac{d\delta}{dz} \right) \quad (3.35)$$

$$-q = \frac{(T_{SAT} - T_{SUP})\lambda_l}{\delta} \quad (3.36)$$

it is possible to write

$$\frac{(T_{SAT} - T_{SUP})\lambda_l}{\delta H_{lv}} = \frac{\rho_l(\rho_l - \rho_v)g}{\mu_l} \delta^2 \frac{d\delta}{dy} + \rho_l u_s \frac{d\delta}{dy} \quad (3.37)$$

$$\frac{(T_{SAT} - T_{SUP})\lambda_l}{H_{lv}} dy = \frac{\rho_l(\rho_l - \rho_v)g}{\mu_l} \delta^3 d\delta + \rho_l u_s \delta d\delta \quad (3.38)$$

Finally, by integrating with the boundary condition $\delta = 0$ at $z = 0$ it is possible to obtain

$$\frac{(T_{SAT} - T_{SUP})\lambda_l}{H_{lv}} y = \frac{\rho_l(\rho_l - \rho_v)g}{4\mu_l} \delta^4 + \frac{\rho_l u_s \delta^2}{2} \quad (3.39)$$

For simplicity it is possible to name

$$a = \frac{(T_{SAT} - T_{SUP})\lambda_l}{H_{lv}} y \quad (3.40)$$

$$b = \frac{\rho_l(\rho_l - \rho_v)g}{4\mu_l} \quad (3.41)$$

$$c = \frac{\rho_l u_s}{2} \quad (3.42)$$

rewriting Eq. (3.39) as follow

$$bx^2 + cx - a = 0 \quad (3.43)$$

with $x = \delta^2$.

Solving the second order Eq. (3.43) it is possible to obtain the liquid film thickness

$$\delta = \left\{ \frac{-\frac{\rho_l u_s}{2} + \sqrt{\left(\frac{\rho_l u_s}{2}\right)^2 + 4 \frac{\rho_l(\rho_l - \rho_v)g(T_{SAT} - T_{SUP})\lambda_l}{4\mu_l H_{lv}} y}}{2 \frac{\rho_l(\rho_l - \rho_v)g}{4\mu_l}} \right\}^{0.5} \quad (3.44)$$

Thus, it is possible to evaluate the gravity component of the condensation heat transfer coefficient in the presence of condensate slip as

$$HTC_{Nu} = \frac{1}{L} \int_0^L 1.15 \cdot \left[0.026 \left(\frac{H_{lv} \mu_l}{\lambda_l (T_{SAT} - T_{wall})} \right)^{\frac{1}{2}} + 0.79 \right] \cdot \frac{\lambda_l}{\delta} dy \quad (3.45)$$

with the same assumptions described in Sec. 3.6.2.

Once the liquid film thickness is known it is possible to evaluate the condensate mass flow rate per unit width as

$$\Gamma = \frac{\rho_l (\rho_l - \rho_v) g \delta^3}{3 \mu_l} \quad (3.46)$$

Thus, by introducing the shear stress at the liquid-vapor interface as in Eq. (3.26), it is possible to calculate the two-phase heat transfer coefficient due to shear stress forces as

$$HTC_{ss} = \frac{1}{L} \int_0^L \sqrt{\frac{\lambda_l^2 \tau_i \rho_l}{2 \Gamma \mu_l}} dy \quad (3.47)$$

Finally, the global heat transfer coefficient during slip driven condensation can be evaluated as in Eq. (3.27).

Referring to single-phase analysis (Tretheway and Meinhardt, 2002; Hsieh and Lin, 2008) the slip velocity at the solid-liquid interface can be determined as

$$u_s = 0.1 \cdot u_{free} \quad (3.48)$$

where u_{free} is the free stream velocity, i.e. the velocity in the middle line between the two planar plates forming the channel, which is the maximum velocity reached by the fluid. In the present case it has been assumed that the slip velocity is 10% of the velocity at the liquid-vapor interface

$$u_s = 0.1 \cdot u_\delta \quad (3.49)$$

with

$$u_\delta = u_{gravity} + u_{ss} + u_s \quad (3.50)$$

The gravity and shear stress components of the liquid film velocity at $z = \delta$ are:

$$u_{gravity} = \frac{(\rho_l - \rho_v) g}{\mu_l} \left(\delta^2 - \frac{\delta^2}{2} \right) \quad (3.51)$$

$$u_{ss} = \frac{\tau_0 \delta}{\mu_l} \quad (3.52)$$

Figure 3.8.4.2 shows the comparison between the experimental data and the values calculated accounting for the slip of the condensate onto the surface ($\pm 20\%$ error bands are reported).

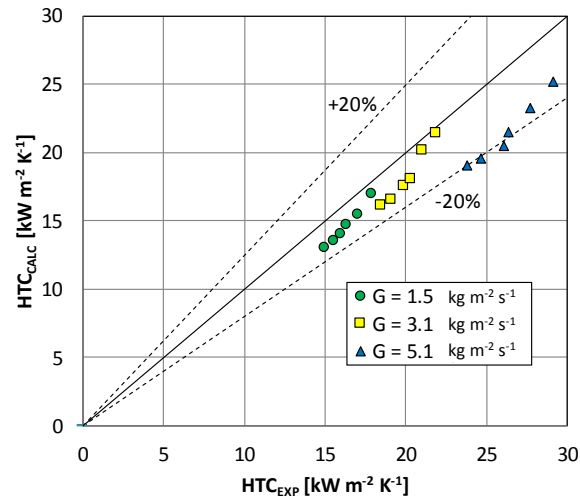


Figure 3.8.4.2. Comparison between the experimental data and the values calculated imposing nonzero slip velocity at the solid-liquid interface. Data refer to different steam mass velocities G .

The Figure shows that by introducing slipping boundary conditions in the evaluation of the heat transfer coefficient it is possible to explain the performances enhancement. Discrepancies between the calculated and the experimental values enhances when increasing the steam mass flow rate. However, such a trend is typical of shear driven condensation process, because of the bigger uncertainty in the estimation of HTC_{ss} than in the estimation of HTC_{Nu} , as shown in Paragraph 3.6.2. Even if research on the conditions at the solid-liquid interface is still ongoing, this analysis shows that by imposing boundary conditions comparable to those found in the literature explanation of the heat transfer enhancement through the slipping theory can be undertaken.

3.9 Conclusions

First of all, in this Section a new experimental apparatus for heat transfer measurement and flow visualization during pure steam condensation has been presented. The set-up has been designed for the study of condensation over micro-/nano- engineered surfaces in presence of vapor velocity. The test section allows the measurement of the heat transfer coefficient from the measurement of the condensation heat flux and of the saturation-to-wall temperature difference. In particular, a new technique that guarantees an easy replacement of the metallic specimen, a low measurement uncertainty and that can be also used with materials having low thermal conductivity is introduced.

The proposed apparatus consists in a thermosyphon flow loop (no circulating pump is used) and it is composed of four main elements: the test section, the evaporator, the cooling water loop and the post-condenser. The experimental test section consists of a rectangular channel (hydraulic diameter equal to 8.6 mm) machined inside a PEEK block. On one side the channel is covered by a glass to allow the visualization of the process and, on the other side, it is fitted with the metallic specimen, cooled on the backside. The specimen is provided with four T-type thermocouples for temperature profile evaluation, since on engineered surfaces a direct measurement of the surface temperature is not possible.

Two-phase tests were performed during filmwise condensation over a standard aluminum surface, varying the saturation-to-wall temperature difference and the vapor mass velocity. The experimental data were successfully compared against correlations available in the open literature, showing the strong influence of the vapor shear stress on the filmwise condensation heat transfer coefficient. The higher the vapor velocity the lower is the condensate film thickness at the wall, leading to a lower thermal resistance associated with the liquid film and thus to a higher heat transfer coefficient.

The aim of the present apparatus is to investigate the effect of surface wetting properties on the condensation heat transfer process, by studying the behaviour of nano-engineered surfaces presenting superhydrophilic, hydrophobic and superhydrophobic characteristics.

Thus, methods to impart wetting properties modification over aluminum substrates are presented. By properly etching the surface it is possible to obtain a nano-structured morphology which leads to extreme water adhesion properties. On the contrary, by forming a Self Assembled Monolayer over the substrate it is possible to lower surface free energy leading to hydrophobic characteristics. Combining the two factors, thus promoting proper superficial roughness and low surface free energy, it is possible to produce surfaces with extreme water repellency and very high droplets mobility. Characterization of the surfaces in terms of contact angles analysis, Scanning Electron Microscope visualization and Atomic Force Microscopy has been presented. The three chemical processes described in this Paragraph can be effectively used for getting superhydrophilic to superhydrophobic aluminum surfaces.

Roughened (superhydrophilic) surfaces were tested with the aim of investigating the effect of surface morphology on the condensation process. It has been found that the roughened sample performs worse than the untreated one, because of condensate layer adhesion to the wall which enhances the liquid film thickness, thus the associated thermal resistance, leading to a reduction of the condensation heat transfer coefficient. This is especially true at low vapor velocity, when the gravity effects are dominant on the two phase phenomenon with respect to the shear stress

ones. At high vapor velocity, the effect of the wettability modification on the heat transfer performance is reduced and the heat transfer coefficient measured over the superhydrophilic sample is similar to those obtained with the standard aluminum sample. This is because the shear stress forces become more relevant and are able to overcome the negative contribution given by the adhesion forces. It can thus be inferred that the condensation process is less efficient on roughened surfaces presenting hydrophilic properties than on a quasi-smooth one, the more the lower the fluid flow rate.

Tests performed over the superhydrophobic substrate showed that extremely low wetting properties are able to promote dropwise condensation mode over filmwise one, enhancing the two phase heat transfer coefficient. Nevertheless, when performing tests on an extremely porous surface in the absence of non condensable gases the dropwise condensation phenomenon is unstable, and it tends to downgrade to filmwise mode. This is because when pure steam condensation occurs surface porosities are filled by condensed vapor, thus drops forming onto the substrate are strongly attached over it. This highly reduces droplets mobility, enhancing the drop departing diameter. When the departing diameter reaches a critical value drops start to coalescence forming a liquid film that covers the surface, thus condensation mode moves to filmwise. Moreover, because of the rough structure of the surface after the transition the liquid film is strongly attached to the substrate, thus the surface performs as a superhydrophilic one.

When steam condenses over the hydrophobic treated substrate enhanced heat transfer performances are achieved, even if condensation occurs in filmwise mode. The augmentation with respect to the untreated sample increases when increasing vapor mass velocity and reduces when increasing saturation-to-wall temperature difference. In the tested conditions, heat transfer coefficient is 10% to 45% higher on the hydrophobic substrate than the one on the untreated sample, depending on steam velocity and mean logarithmic temperature difference between the steam and the wall.

Enhancement of the two-phase heat transfer can be explained by considering condensate slip onto the water repellent surface. It has been shown in the literature that slipping of fluid at the solid-liquid interface (reflecting in a nonzero slip length) can be achieved if the surface has fluid repellency properties. Moreover, several theoretical studies showed that when considering nonzero slip length in the filmwise condensation process higher heat transfer coefficients are achieved, the more the higher is the slip length. This is due to the fact that an augmentation of the slip length is associated with an increment of the slip velocity at the surface, which results in a thinner condensate layer. Lowering liquid film thickness means lowering the associated thermal resistance, this way enhancing the condensation heat transfer coefficient. It can thus be inferred

that when filmwise condensation occurs over hydrophobic surfaces slipping conditions are achieved at the solid-condensate interface, leading to higher heat transfer properties. To consolidate this theory, computational analysis of slip-driven filmwise condensation has been undertaken, by modifying classical equations for predicting condensation heat transfer coefficient introducing nonzero slip velocity conditions at the solid-liquid interface. A satisfactory agreement has been found between the calculated and the experimental data, confirming the theory presented above.

Hydrophobic surfaces are thus a viable solution for enhancing condensation heat transfer performance, due to promotion of condensate slip.

Nomenclature

A = Area, m^2

c_p = Specific heat capacity, $J\ kg^{-1}\ K^{-1}$

dE = Net energy change, $J\ m^{-1}$

D_h = Hydraulic diameter, m

$(dp/dz)_F$ = Two-phase frictional pressure gradient, $Pa\ m^{-1}$

ds = Distance variation, m

$|dT/dz|$ = Temperature gradient inside the sample, $K\ m^{-1}$

$\bar{e} = \frac{1}{n_{points}} \sum_{i=1}^{n_{points}} e_i$ = Average deviation, %

$e_i = \frac{HTC_{CALC} - HTC_{EXP}}{HTC_{EXP}} \cdot 100$ = Percentage

deviation of the i -th point, %

f = Fractional area, /

g = Acceleration of gravity, $m\ s^{-2}$

G = Specific mass velocity, $kg\ m^{-2}\ s^{-1}$

h = Enthalpy, $J\ kg^{-1}$

H = Height, m

H_{lv} = Latent heat of vaporization, $J\ kg^{-1}$

HTC = Heat transfer coefficient, $W\ m^{-2}\ K^{-1}$

k = Coverage factor, /

K_V = Flow coefficient, $m^3\ h^{-1}\ bar^{-1/2}$

L = Length, m

m = Mass, kg

$\dot{m}$ = Mass flow rate, $kg\ s^{-1}$

p = Pressure, Pa

q = Heat flux, $W\ m^{-2}$

Q = Heat flow rate, W

R_a = Arithmetical mean deviation of the assessed profile, m

R_f = Roughness factor, /

S = Cross sectional area, m^2

T = Temperature, K

T' = Sample temperature at z_1 , K

T'' = Sample temperature at z_2 , K

u = Velocity, $m\ s^{-1}$

$\bar{u}$ = Mean velocity, $m\ s^{-1}$

u_A = Type A uncertainty, %

u_B = Type B uncertainty, %

u_C = Combined standard uncertainty, %

u_M = Expanded uncertainty, %

u_s = Slip-velocity, $m\ s^{-1}$

u_y = Velocity along the sample, $m\ s^{-1}$

w = Droplet contact diameter, m

y = Vertical position along the sample, m

z = Axial position along the sample, m

z_1 = z position 1 mm from the sample top surface, m

z_2 = z position 2.75 mm from the sample top surface, m

GREEK SYMBOLS

α = Tilt angle, $^\circ$

β = Slip length, m

γ = Interfacial free energy per unit of area, $J\ m^{-2}$

Γ = Film flow rate per unit width, $kg\ m^{-1}\ s^{-1}$

δ = Liquid film thickness, m

$\Delta\theta$ = Contact angle hysteresis, $^\circ$

Δp = Pressure loss, Pa

ΔT = Temperature variation, K

ΔT_{ml} = Mean logarithmic temperature difference, K

ϑ = Contact angle, $^\circ$

ϑ_0 = Contact angle on a smooth surface, $^\circ$

ϑ_{adv} = Advancing contact angle, $^\circ$

θ_{rec} = Receding contact angle, °

λ = Thermal conductivity, W m⁻¹ K⁻¹

μ = Viscosity, Pa s

ρ = Density, kg m⁻³

$$\sigma_N = \left\{ \frac{\sum (e_i - \bar{e})^2}{n_{points} - 1} \right\}^{0.5}$$

= Standard deviation, %

τ = Time, s

τ_i = Shear stress at the liquid-vapor interface, Pa

SUBSCRIPTS

AVE = Average

BC = Boiling chamber

CALC = Calculated

EXP = Experimental

HY = Hydrophobic

IN = Inlet

L, l = Liquid

LA = Liquid – air interface

m = Mean

Nu = Nusselt theory

OUT = Outlet

post-cond = Post-condenser

SA = Solid – air interface

SAT = Saturation

SH = Superhydrophilic

SL = Solid – liquid interface

SS = Shear stress

SUP = Superficial

TOT = Total

UN = Untreated

V, v = Vapor

CONCLUSIONS

In the present Ph.D. Thesis fire tube heat generators are theoretically and experimentally analyzed with the aim of enhancing heat transfer performances. In Section 1, working conditions of a three pass system have been experimentally investigated and a model for prediction of heat generators performances has been presented. After validation of the model through the new experimental database, simulations have been undertaken for evaluation of the system working conditions when varying inlet parameters or heat generator geometry, with the aim of increasing system efficiency or of reducing system costs maintaining the same performance. Then, strategies for enhancing both single- and two- phase heat transfer have been analyzed. A computational study on the enhancement of convective heat transfer coefficient by the introduction of turbulence generator elements has been performed in the second Chapter of this work. CFD simulations have been undertaken both with geometries already used by the heat generator industry and with a new shape, developed from a critical analysis of the previous ones. Finally, considering that in modern heat generators recovery of latent heat through the condensation of the steam in the exhaust products is widely used, in the last part of the present manuscript two-phase heat transfer enhancement by means of nano-engineered surfaces has been studied. Techniques for modifying wetting properties of aluminum substrates are presented and results of vapor condensation tests on surfaces presenting different wettability are discussed. For the sake of experimental simplicity, pure steam condensation has been tested.

1. Experimental and theoretical analysis of heat transfer inside heat generators

Experimental analysis of heat transfer inside heat generators was performed inside a three pass fire tube system, fed with natural gas; varying fuel mass flow rate, air index and inlet water temperature and mass flux. Moreover, tests were performed with and without turbulators inside the last flue gases pass. The experimental analysis showed that when inserting turbulence generators inside the system lower exit temperatures are achieved, meaning 3% to 10% higher system efficiencies.

A dynamic model for predicting working conditions of the system has been presented and validated through the new experimental database. Comparison between experimental and calculated data showed good agreement both during stationary and transient regimes. The model herein presented can be used for predicting the system behavior when operating with varying

thermal power at the burner, varying conditions of the water inside the shell and with different geometry. Thus, after validation the model has been used for simulating different configurations, with the aim of achieving better performances or of maintaining them reducing the system costs. This analysis showed that reducing the number of pipes in the last flue gases pass it is possible to achieve satisfactory results lowering the number of turbulence generator inserts, which are a high impact elements in the cost of the system.

2. Single-phase heat transfer enhancement by means of turbulators

Several turbulators geometries can be found in the open literature but only some of them are viable solutions for heat generators industry, being manufacture simplicity a mandatory point. Because of this, actual geometries are mainly produced through pressing methods.

CFD simulations have been carried out on two geometries, named single- and double- wave turbulators, actually used by the heat generator industry, evaluating influence of several geometrical parameters such as insert position in the tube and pipe diameter. Results of the simulations showed that turbulator presence enhances the convective heat transfer coefficient inside the tube, but at the same time it increases the frictional pressure losses. Thus, to evaluate the insert global performances the enhancement efficiency (i.e. thermal performance ratio), which accounts both for thermal and frictional augmentation, has been introduced.

Single-wave turbulator simulations showed that when increasing the tube diameter from 36.4 mm to 42.5 mm the Reynolds number range for which the enhancement efficiency is bigger than one increases, because of reduced frictional losses. On the contrary, no influence of the turbulator position inside the tube has been found. Comparing single wave and double wave turbulators it has been shown that the latter presents higher thermal performance ratio in the laminar and transient flow regimes, because of the reduced frictional losses, while in the fully developed turbulent regime the single wave insert performs better, because of the higher Nusselt number enhancement. Since these simulations are representative of real conditions inside fire tube heat generators, equations to predict turbulators performances have been presented.

A modified version of the double-wave turbulator, obtained by inserting central jags to create turbulence in the flue gases flowing in the central part of the tube, has been presented and simulated. It has been found that the jagged double wave turbulator performs better than the standard one for fully developed turbulent flue gases flow regime (which is the case of large scale heat generators or of medium scale systems with a reduced number of tubes in the last flue gases pass), while for low Reynolds number the old geometry works better.

3. Two-phase heat transfer enhancement by means of nano-engineered surfaces

A new experimental apparatus for heat transfer measurement and visualization during pure steam flow condensation, at different vapor velocity, over nano-engineered surfaces has been presented. In particular, a new technique is introduced that guarantees an easy replacement of the metallic specimen, a low measurement uncertainty and that can be also used with materials having low thermal conductivity.

Heat transfer properties of different aluminum substrates, nano-engineered for achieving different wetting characteristics, have been studied.

Methods to impart wetting properties modification over aluminum substrates were presented. By properly etching the surface a nano-structured morphology, which leads to extreme water adhesion properties, has been obtained. On the contrary, by forming a Self Assembled Monolayer over the substrate surface free energy has been lowered leading to hydrophobic characteristics. Combining the two techniques, thus promoting proper superficial roughness and low surface free energy, surfaces with extreme water repellency (superhydrophobic) have been obtained. Moreover, characterization of the surfaces in terms of contact angles analysis, Scanning Electron Microscope visualization and Atomic Force Microscopy has been presented.

Prior to any test over the nano-structured substrates experimental data were acquired during condensation over standard aluminum plates, both for comparison purpose and for validation of the experimental technique (data were successfully compared against literature correlations).

Heat transfer analysis over roughened superhydrophilic specimen showed lower heat transfer coefficients than those obtained over the untreated sample, because of condensate adhesion to the wall which enhances the liquid film thickness, negatively affecting the two-phase process. This is especially true at low vapor velocity, while when steam mass flow rate increases the shear stress forces are able to balance the negative effect of the adhesion forces. From this analysis it has been stated that the condensation process is less efficient on roughened surfaces presenting hydrophilic properties, the more the lower the fluid flow rate, than on a quasi-smooth one.

Tests performed over the superhydrophobic substrate showed that extremely low wetting properties are able to promote dropwise condensation mode over filmwise one, enhancing the two phase heat transfer coefficient. Nevertheless, on a surface presenting nano-scale porosities as the one under examination, such phenomenon is extremely unstable. In fact, when pure steam condensation occurs surface porosities are filled by condensed vapor, thus drops forming onto the substrate stand in a Wenzel wetting regime, being strongly pinned on the surface. This enhances the drops departing diameter until a critical value over which drops start to coalesce forming a liquid film that covers the surface, downgrading condensation mode from dropwise to

filmwise. It has also been found that, since the surface presents a nano-structured rough morphology, after transition the liquid film is strongly attached to the wall, thus the surface performs as a superhydrophilic one.

Finally, when steam condenses over the hydrophobic treated substrate enhanced heat transfer performance is achieved, even if condensation still occurs in filmwise mode. Comparing the hydrophobic and the standard sample 10% to 45% higher heat transfer coefficients were obtained. The higher the enhancement, the higher the steam mass velocity and the lower the saturation to wall temperature difference.

The enhancement of the two-phase heat transfer can be explained by considering condensate slip on the water repellent surface. Water repellency properties of the surface lead to condensate slipping, thus to a nonzero slip length, reflecting in a nonzero slip velocity at the surface. This means a thinner condensate layer, thus an enhanced condensation heat transfer coefficient. Computational analysis of slip-driven filmwise condensation has been undertaken, by modifying classical equations for predicting condensation heat transfer coefficient introducing nonzero slip velocity conditions at the solid-liquid interface. A good agreement has been found between the calculated and the experimental data, confirming that hydrophobic surfaces are a viable solution for enhancing condensation heat transfer performances, due to promotion of condensate slip.

4. Future steps

Beside manufacturing new surfaces to obtain dropwise condensation for longer times, an important future activity should be the study of the two-phase phenomenon in presence of non-condensable gases.

REFERENCES

- Al-Jarrah J.A., Khadrawi A.F., AL-Nimr M.A., 2008. Film condensation on a vertical microchannel. *International Communications in Heat and Mass Transfer* 35 1172-1176.
- Annaratone D., 2008. *Steam Generators, Description and Design*. Springer, Berlin, Germany.
- Antonini C., Innocenti M., Horn T., Marengo M., Amirfazli A., 2011. Understanding the Effects of Superhydrophobic Coatings on Energy Reduction in Anti-Icing Systems. *Cold Reg. Sci. Technol.* 67 58-67.
- Astrom K.J., Bell R.D., 2000. Drum-boiler dynamics. *Automatica* 36 363-378.
- Baehr H. D., Stephan K., 2004. *Waerme- und Stoffuebertragung*. Springer- Verlag, Berlin.
- Berndt V., Zunft S., Muller H., 2008. Theoretical and experimental study on dropwise condensation in plate heat exchangers. *Proc. 5th. European Thermal-Sciences Conference, The Netherlands*.
- Betz A. R., Jenkins J., Kim C. J., Attinger D., 2013. Boiling heat transfer on superhydrophilic, superhydrophobic and superbiphilic surfaces. *Int. J. Heat Mass Tran.* 57 733-741.
- Bico J., Thiele U., Quéré D., 2002. Wetting of textured surfaces. *Colloids Surfaces A* 206 41–46.
- Bonner R.W., 2010. Dropwise condensation life testing of self assembled monolayers. *Proc. 14th Int. Heat Transfer Conf., Washington, DC, USA*.
- Bordbar M.H., Hyppanen T., 2007. Modeling of Radiation Heat Transfer in a Boiler Furnace. *Adv. Studies Theor. Phys.*, Vol 1 no. 12 571-584.
- Boreyko J. B., Chen C. H., 2009. Self-propelled dropwise condensate on superhydrophobic surfaces. *Physical Review Letters* 103 184501.
- Chang S.W., Jan Y.J., Liou J.S., 2007a. Turbulent heat transfer and pressure drop in tube fitted with serrated twisted tape. *International Journal of Thermal Science* 46 506-518.
- Chang S.W., Yang T.L., Liou J.S., 2007b. Heat transfer and pressure drop in tube with broken twisted tape insert. *Experimental Thermal and Fluid Science* 32 489 – 501.
- Chen C. H., Cai Q. J., Tsai C., Chen L., 2007. Dropwise condensation on superhydrophobic surfaces with two-tier roughness. *Appl. Phys. Lett.* 90 173108.
- Chen Z., Guo Y., Fang S., 2010. A facial approach to fabricate superhydrophobic aluminum surface. *Surf. Interface Anal.* 42 1-6.
- Depew C. A., Reisbig R. L., 1964. Vapor condensation on a horizontal tube using teflon to promote dropwise condensation. *I&EC Process Des. Develop.* 3 4 365–369.

Dietz C., Rykaczewski K., Fedorov A. G., Joshi Y., 2010. Visualization of droplet departure on a superhydrophobic surface and implications to the heat transfer enhancement during dropwise condensation. *Appl. Phys. Lett.* 97 033104.

Dorrer C., Ruhe J., 2007. Condensation and wetting transition on microstructured ultrahydrophobic surfaces. *Langmuir* 23 3820-3824.

Eiamsa-ard S., Pethkool S., Thianpong C., Promvong P., 2008. Turbulent flow heat transfer and pressure loss in a double pipe heat exchanger with louvered strip inserts. *International Communications in Heat and Mass Transfer* 35 120-129.

Eiamsa-ard S., Promvong P., 2005. Enhancement of heat transfer in a tube with regularly-spaced helical tape swirl generators. *Solar Energy* 78 483-494.

Eiamsa-ard S., Promvong P., 2011. Influence of Double-sided Delta-wing Tape Insert with Alternate-axes on Flow and Heat Transfer Characteristics in a Heat Exchanger Tube. *Fluid Flow and Transport Phenomena* 19 (3) 410 – 423.

Eiamsa-ard S., Seemawute P., Wongcharee K., 2010b. Influences of peripherally-cut twisted tape insert on heat transfer and thermal performance characteristics in laminar and turbulent tube flows. *Experimental Thermal and Fluid Science* 34 711 – 719.

Eiamsa-ard S., Thianpong C., Eiamsa-ard P., Promvong P., 2010a. Thermal characteristics in a heat exchanger tube fitted with dual twisted tape elements in tandem. *International Communications in Heat and Mass Transfer* 37 39-46.

Eiamsa-ard S., Wongcharee K., Sripattanapipat S., 2009. 3-D Numerical simulation of swirling flow and convective heat transfer in a circular tube induced by means of loose-fit twisted tapes. *International Communications in Heat and Mass Transfer* 36 947-955.

Emara-Shabaik H.E., Habib M.A., Al-Zharna I., 2009. Prediction of risers' tubes temperature in water tube boilers. *Applied Mathematical Modelling* 33 1323-1336.

Friedel L., 1979. Improved friction pressure drop correlations for horizontal and vertical two-phase pipe flow. In: *Proceedings of the European Two-phase Flow Group Meeting Ispra, Paper E2.*

Gao L., McCarthy T. J., 2007. How Wenzel and Cassie were wrong. *Langmuir* 23 3762–3765.

Gómez A., Fuego N., Díez L.I., 2008. Modelling and simulation of fluid flow and heat transfer in the convective zone of a power-generation boiler. *Appl. Therm. Eng.* 28 532-546.

Gutiérrez Ortiz F.J., 2011. Modeling of fire tube boilers. *Applied Thermal Engineering* 31 3463-3478.

Hewitt G. F., Shires G. L., Bott T. R., 1994. *Process Heat Transfer*. CRC Press Inc., Boca Ranton, Florida.

Hsieh S.S., Lin C.Y., 2009. Convective heat transfer in liquid microchannels with hydrophobic and hydrophilic surfaces. *International Journal of Heat and Mass Transfer* 52 260-270.

IAPWS, 1997. Revised Release on the IAPWS Industrial Formulation 1997 for the Thermodynamic Properties of Water and Steam. Available from www.iapws.org, 1997, last access 20/01/2014.

Incropera F.P., DeWitt D.P., 1996. *Fundamentals of Heat and Mass Transfer*, Fourth Edition. John Wiley & Sons, Inc., Hoboken, New Jersey.

ISO Guide to the expression of uncertainty in measurement, 1995.

Jafari R., Farzaneh M., 2011. Fabrication of superhydrophobic nanostructured surface on aluminum alloy. *Applied Physics A* 102 195-9.

Johnson R. E., Dettre R. H., 1964. Contact angle hysteresis, I study of an idealized rough surface. *Advances in Chemistry series* 43 112-35.

Kananeh A. B., Rausch M. H., Froba A. P., Leipertz A., 2006. Experimental study of dropwise condensation on plasma-ion implanted stainless steel tubes. *Int. J. Heat Mass Tran.* 49 5018-5026.

Kananeh A. B., Rausch M. H., Leipertz A., Froba A. P., 2011. Dropwise Condensation Heat Transfer on Plasma-Ion-Implanted Small Horizontal Tube Bundles. *Heat Transfer Eng* 31 10 821-828.

Kim H., Choi S., 2005. A model on water level dynamics in natural circulation drum-type boilers. *Int. Commun. Heat Mass* 32 786-796.

Kumar A., Prasad B.N., 2000. Investigation of twisted tape inserted solar water heaters-heat transfer, friction factor and thermal performance results. *Renew. Energy* 19 379–398.

Lan Z., Ma X., Wang S., Wang M., Li X., 2010. Effects of surface free energy and nanostructures on dropwise condensation. *Chemical Eng. Journal* 156 546-552.

Lemmon E. W., Huber M. L., McLinden M. O., 2013. NIST Standard Reference Database 23: Reference Fluid Thermodynamic and Transport Properties-REFPROP, Version 9.1.

Ma X., Wang S., Lan Z., Peng B., Ma H. B., Cheng P., 2012. Wetting Mode Evolution of Steam Dropwise Condensation on Superhydrophobic Surface in the Presence of Noncondensable Gas. *J. Heat Transf.* 134 021501-1.

Ma X., Wang S., Lan Z., Wang A., Peng B., 2010. Dropwise condensation heat transfer on superhydrophobic surface in the presence of non-condensable gas. *Proc. 14th Int. Heat Transfer Conf.*, Washington, DC, USA.

Ma X., Zhou X. D., Lan Z., Li Y. M., Zhang Y., 2008. Condensation heat transfer enhancement in the presence of non-condensable gas using the interfacial effect of dropwise condensation. *Int. J. Heat Mass Tran.* 51 1728-1737.

Manglik R.M., Bergles A.E., 1993a. Heat transfer and pressure drop correlations for twisted tape inserts in isothermal tubes: part I— laminar flows. *ASME J. Heat Transfer* 115 881–889.

Manglik R.M., Bergles A.E., 1993b. Heat transfer and pressure drop correlations for twisted tape inserts in isothermal tubes: part II – transition and turbulent flows. *ASME J. Heat Transfer* 115 890–896.

Narhe R. D., Beyesen D. A., 2004. Nucleation and growth on a superhydrophobic grooved surface. *Phys Rev Lett* 93 76103.

Neinhuis C., Barthlott W., 1997. Characterization and distribution of water-repellent, self cleaning plant surfaces. *Ann. Bot.* 79 667-77.

Nusselt W., 1916. Surface condensation of water vapour. Parts I and II, *V.D.I.* 60 27, 60 28.

Patankar N. A., 2010. Supernucleating surfaces for nucleate boiling and dropwise condensation heat transfer. *Soft Matter* 6 1613-1620.

Pati S., Som S.K., Chakraborty S., 2013. Slip-driven alteration in film condensation over vertical surfaces. *International Communications in Heat and Mass Transfer* 46 37-41.

Pezo M., Stevanovic V.D., Stevanovic Z., 2006. A two-dimensional model of the kettle reboiler shell side thermal-hydraulics. *Int. J. Heat Mass Tran.* 49 1214-1224.

Ponter A. B., 1992. Contact angles on metal and polymer surfaces in mass transfer environments. *Adv.Colloid Interface Sci.* 39 383–395.

Quéré D., 2005. Non sticking drops. *Rep. Prog. Phys.* 68 2495-2532.

Quere D., Lafuma A., Bico J., 2003. Slippery and sticky microtextured structures. *Nanotechnology* 14 1109-1112.

Rahimi M., Shabanian S.R., Alsairafi A.A., 2009. Experimental and CFD studies on heat transfer and friction factor characteristics of a tube equipped with modified twisted tape inserts. *Chemical Engineering and Processing* 48 762 – 770.

Rausch M. H., Froba A. P., Leipertz A., 2008. Dropwise condensation heat transfer on ion implanted aluminum surfaces. *Int. J. Heat Mass Tran.* 51 1061-1070.

Rausch M. H., Leipertz A., Fröba A. P., 2010. Mechanism of dropwise condensation on ion implanted metallic surfaces. *Proc. 14th Int. Heat Transfer Conf.*, Washington, DC, USA.

Rose J. W., 1981. Dropwise Condensation Theory. *Int. J. Heat Mass Transf.* 24 191-194.

Rose J. W., 1999. Laminar film condensation of pure vapors. in: Kandlikar S. G., Shoji M., Dhir V. K., 1999. *Handbook of Phase Change: Boiling and Condensation*. Taylor & Francis, Philadelphia.

Rothstein J.P., 2010. Slip on Superhydrophobic Surfaces. *Annu. Rev. Fluid Mech.* 42 89-109.

Saleema N., Sarkar D. K., Paynter R W, Chen X. G., 2010. Superhydrophobic Aluminum Alloy Surfaces by a Novel One-Step Process. *Applied Materials and Interfaces* 2 2500-2.

Sarkar D. K., Farzaneh M., Paynter R. W., 2008. Superhydrophobic properties of ultrathin rf-sputtered Teflon films coated etched aluminum surfaces. *Mater. Lett.* 62 1226-9.

Sarkar D. K., Farzaneh M., Paynter R. W., 2010. Wetting and superhydrophobic properties of PECVD grown hydrocarbon and fluorinated-hydrocarbon coatings. *Appl. Surf. Sci.* 256 3698-701.

Schmidt E., Schurig W., Sellschopp W., 1930. Versuche über die kondensation von wasserdampf in film- und tropfenform. *Forschung im Ingenieurwesen* 1 (2) 53-63.

Sivashanmugam P., Suresh S., 2006. Experimental studies on heat transfer and friction factor characteristics of laminar flow through a circular tube fitted with helical screw-tape inserts. *Applied Thermal Engineering* 26 1990-1997.

Sivashanmugam P., Suresh S., 2007. Experimental studies on heat transfer and friction factor characteristics of laminar flow through a circular tube fitted with regularly spaced helical screw-tape inserts. *Applied Thermal and Fluid Science* 31 301-308.

Sørensen K., 2004. Dynamic Boiler Performance – modeling, simulating and optimizing boilers for dynamic operation. Ph. D. Thesis, Aalborg University, Aalborg, Denmark.

Sørensen K., Kartensen C.M.S., Condra T., Houbak N., 2003. Modelling and simulating fire tube boiler performance. 44th SIMS Conference.

Torresin D., Tiwari M. K., Del Col D., Poulikakos D., 2013. Flow condensation on copper based nanotextured superhydrophobic surfaces. *Langmuir* 29 2 840-8.

Tretheway D.C., Meinhart C.D., 2004. A generating mechanism for apparent fluid slip in hydrophobic microchannels. *Physics of Fluids* 16 5 1509-1515.

Trinks W., Mawhinney M.H., Shannon M.A., Reed R.J., Garvey J.R., 2004. *Industrial Furnaces*, Sixth edition. John Wiley & Sons, Inc., Hoboken, New Jersey.

Umur A., Griffth P., 1965. Mechanism of Dropwise Condensation. *ASME J. Heat Transfer* 87 275-282.

Varanasi K. K., Hsu M., Bathe N., Yang W. S., Deng T., 2009. Spatial control in the heterogeneous nucleation of water. *Appl. Phys. Lett.* 95 094101.

Vemuri S., Kim K. J., Wood B. D., Govindaraju S., Bell T. W., 2006. Long term testing for dropwise condensation using self-assembled monolayer coatings of n-octadecyl mercaptan. *Appl. Therm. Eng.* 26 421-429.

Voronov R.S., Papavassiliou D.V., Lee L.L., 2007. Slip length and contact angle over hydrophobic surfaces. *Chemical Physics Letters* 441 273-276.

Webb R.L., 1994. *Principles of Enhanced Heat Transfer*. John Wiley & Sons, New York.

Wier K. A., McCarthy T. J., 2006. Condensation on ultrahydrophobic surfaces and its effect on droplet mobility: ultrahydrophobic surfaces are not always water repellent. *Langmuir* 22 (6) 2433-2436.

Wongcharee K., Eiamsa-ard S., 2011. Friction and heat transfer characteristics of laminar swirl flow through the round tubes inserted with alternate clockwise and counter-clockwise twisted-tapes. *International Communications in Heat and Mass Transfer* 38 348 – 352.

Zhou H., Wang H., Niu H., Gestos A., Wang X., Lin T., 2012. Fluoroalkyl Silane Modified Silicone Rubber/Nanoparticle Composite: A Super Durable, Robust Superhydrophobic Fabric Coating. *Adv. Mater.* 24 2409-2412.

PUBLICATIONS

Journal publications:

- Del Col D, Bisetto A, Bortolato M, Torresin D, Rossetto L (2013). Experiments and updated model for two phase frictional pressure drop inside minichannels. *INTERNATIONAL JOURNAL OF HEAT AND MASS TRANSFER*, vol. 67; p. 326-337.
- Bisetto A, Del Col D, Schievano M (2015). Fire tube heat generators: experimental analysis and modeling. *APPLIED THERMAL ENGINEERING*, vol. 78, p. 236-247.
- Bisetto A, Bortolin S, Del Col D. Experimental analysis of steam condensation over conventional and superhydrophilic vertical surfaces. Submitted to *EXPERIMENTAL THERMAL AND FLUID SCIENCE*.
- Del Col D, Azzolin M, Bisetto A, Bortolin S. Frictional pressure drop during two-phase flow of pure fluids and mixtures in small diameter channels. Submitted to *INTERNATIONAL JOURNAL OF CHEMICAL REACTOR ENGINEERING*.
- Bisetto A, Bortolin S, Martucci A, Del Col D. Slip-driven condensation over hydrophobic surfaces. To be submitted to *INTERNATIONAL JOURNAL OF MULTIPHASE FLOW*.
- Bisetto A, Bortolin S, Martucci A, Del Col D. Dropwise condensation over superhydrophobic nanostructured aluminum substrates. To be submitted to *LANGMUIR*.

International conference publications:

- Del Col D, Bisetto A, Bortolato M, Torresin D, Rossetto L (2012). Updated model for two phase frictional pressure drop inside minichannels. 3rd IIR Workshop on Refrigerant Charge Reduction In Refrigerating Systems. Valencia, Spain, 25-26 October 2012.
- Bisetto A, Stamatopoulos C, Tiwari M K, Del Col D, Poulikakos D (2014). Robust superhydrophobic aluminum based surfaces for two-phase heat transfer applications. 101 Eurotherm Seminar on Transport Phenomena in Multiphase Systems. Krakow, Poland, 30 June - 03 July 2014.
- Bisetto A, Bortolin S, Del Col D, Martucci A (2015). Condensation of steam over nano-engineered surfaces. 9th International Conference on Boiling and Condensation Heat Transfer. Boulder, Colorado, USA, 26-30 April 2015.

National conference publications:

- Bisetto A, Del Col D, Schievano M (2012). Development of a dynamic model for a fire tube heat generator. 67° Congresso Nazionale ATI. Trieste, Italy, 11-14 September 2012.
- Bisetto A, Torresin D, Tiwari M K, Del Col D, Poulikakos D (2014). Dropwise condensation on superhydrophobic nanostructured surfaces: literature review and experimental analysis. *JOURNAL OF PHYSICS. CONFERENCE SERIES*, vol. 501, presented at: 31st UIT Heat Transfer Conference. Como, Italy, 25-27 June 2013.
- Bisetto A, Torresin D, Tiwari M K, Del Col D, Poulikakos D (2013). Nanostructured Surfaces for Enhanced Two-Phase Heat Transfer. *NanotechITALY 2013*. Venezia, Italy, 27-29 November 2013.
- Bisetto A, Bortolin S, Del Col D (2014). New experimental apparatus for investigation of condensation over plain and nano-engineered surfaces. 32nd UIT Heat Transfer Conference. Pisa, Italy, 23-25 June 2014.

- Del Col D, Azzolin M, Bisetto A, Bortolin S (2014). A predicting method of frictional pressure drop during two-phase flow in small diameter channels. 13th International Conference Multiphase Flow in Industrial Plant. Sestri Levante (Genova), Italy, 17-19 September 2014.

ACKNOWLEDGEMENTS

During the three years of my Ph.D. project I received both scientific and personal support from a great number of people, to which I want to express my deep gratitude.

Prior to any other I want to acknowledge my supervisor, **Prof. Davide Del Col**. He firstly has been a teacher to me, during the second year of my Undergraduate Degree course. Later on, he has been supervisor of both my Undergraduate and Master Degree Thesis, and finally he leaded me during my Ph.D.. He has been a teacher and a mentor, both in professional and private life. He has been a professor to me, then he became a supervisor, but most of all I'm honored to consider him a friend.

During my Ph.D. I worked at the Two-Phase Heat Transfer Laboratory of the Industrial Engineering Department, together with a lot of extraordinary people. All of them mean a lot to me, thus I will name them in alphabetical order, to be fair. I really had fun together with, and tanks to, **Marco Azzolin**, even if he still owes me a spritz (for a very old matter). I don't know if he will pay off the debt one day or another, but for sure I will always enjoy an apéro with him. Talking about apéro and drinks I have for sure to name **Matteo Bortolato**, who was the first guy of the group that I met, since he has been my Master Thesis co-supervisor. From then on we teased each other with a lot of stupid and funny jokes that I really enjoyed. **Stefano Bortolin** has been a unique colleague and an incredible playmate. Working with him I learned a lot, but most of all I had a lot of fun. I will never forgot our discussions, our basketball challenges (which I mostly won!!) and our jokes. Regarding **Simone Dugaria**, unfortunately I have to point out that, even if he is a very good researcher, he is an absolutely unreliable delivery-boy, since I'm still waiting the "*pizza porchetta e peperoni*" that I ordered him several months ago. I want to acknowledge **Andrea Francescon**, because he has been "the guy who knows everything about everything", since he always had an answer to all my technical questions. In the few months I spent with **Elisa Magro** I had the possibility to appreciate her sense of humor (??) when calling me "werewolf", as well as her kindness. My regards go also to **Andrea Padovan**, which was the only guy of the group (besides me) to really understand something about football. Many thanks go also to **Michele Quaggia**, for bringing me back home when we had our mountain tours as well as for always showing me a lot of positivity in all the issues that I had to face. I want to thank "*The Champion*" **Marco Rossato**, most of all for organizing all the futsal matches. Even if he is no more at the first Ph.D. year, I will always remember him not only as "*The Champion*", but mostly as "*Il Pivello*". Finally, but as I said

only because of alphabetical order, I want to say thanks to **Paolo Toninelli**, for brighten my days with his songs, which have been a piped music during the writing of my Thesis.

Focusing on personal life, I'm incredibly thankful to **my parents**, who supported me during all my life and in all my choices. I could never have achieved this results and I could never have been the man I am without their love, their care and their guidance. Many thanks go also to my brother, **Giovanni**, as well as to all **my family**, for always believing in me and be at my side.

Finally, I want to thank the person who brings the sun in my days, my girlfriend **Chiara**. I met her during my Ph.D. course, and from then on she totally changed my life. We are now starting our life together, and there is nothing I can say to her rather than this:

"You are the most important, the most beautiful and the most precious thing in my life. You are my everything, and I love you so much".

Alberto