Numerical and Experimental Investigation of Variable Transpiration Cooling for Reusable Thermal Protection Systems

by

STEFANO GULLI

Presented to the Faculty of the Graduate School of The University of Texas at Arlington in Partial Fulfillment of the Requirements for the Degree of

DOCTOR OF PHILOSOPHY

THE UNIVERSITY OF TEXAS AT ARLINGTON

December 2014
Dedicated to my lovely wife Alessandra for always being by my side supporting me and encouraging me throughout this period. A special feeling of gratitude goes to my parents, Roberto Gulli and Daniela Rosciani, to my brother Daniele Gulli and to my grandparents for having sustained me throughout my entire life and for always having believed in me.
ACKNOWLEDGEMENTS

I would like to greatly thank my supervising professor Dr. Luca Maddalena for having believed in me since I came here the first time and for having spent countless hours of reflecting, reading, studying and encouraging me throughout the last four years.

I am also grateful to Dr. Frank Lu, Dr. Donald Wilson, Dr. Serhat Hosder, Dr. Panayiotis Shiakolas and Dr. Kambiz Alavi for being part of my committee. Same gratitude goes to my school division for allowing me to conduct my research by providing any assistance requested and the financial support.

I would also like to extend my appreciation to Dr. Claudio Bruno for his continuous guide during my Master’s degree studies and for having encouraged me towards the PhD degree in the United States under the guide of Dr. Maddalena.

Special thanks go to Dr. Serhat Hosder of the Missouri University of Science and Technology for his precious collaboration provided for the numerical validation of the reduced order model developed in this work and also for his valuable suggestions provided in order to improve the quality of our work.

Many thanks go to Mr. Chris McKelvey and Mr. Aaron Brown of C-CAT (Carbon-Carbon Advanced Technologies, Inc.) for their help and support provided for the manufacturing of the prototype model used for the experimental investigation.

I also wish to thank Dr. Yuri Nikishkov and Dr. Andrew Makeev for their substantial contribution toward the completion of this work. I have truly appreciated
their support for letting us use the X-Ray CT machine that provided us fundamental data for the non-intrusive characterization of the prototype sample manufactured by C-CAT.

Sincere gratitude goes to Dr. Kambiz Alavi and Jahangir Mohammadreza for their continuous support in terms of equipment sharing and valuable time spent for calibrating and setting-up their infrared camera used for the experimental campaign on transpiration cooling.

Finally, I would like to express my deep gratitude to the entire staff, faculty and students that I have encountered in the last four years and that helped me to progress on my research.

December 3, 2014
ABSTRACT

Numerical and Experimental Investigation of Variable Transpiration Cooling for Reusable Thermal Protection Systems

Stefano Gulli, Ph.D.

The University of Texas at Arlington, 2014

Supervising Professor: Luca Maddalena

This research work is focused on the numerical and experimental investigation of the transpiration cooling technique for reusable thermal protection systems. The transpiration cooling has been considered because of its higher cooling effectiveness, with respect to other active cooling techniques, in terms of minimum amount of coolant to be used and minimum interference to the external flow. An innovative approach based on the coupling of the hypersonic boundary layer with the thermal response of porous materials is used to numerically study the thermal management potential of the transpiration cooling technique.

The new concept of the variable transpiration is introduced by imposing selected distributions of the coolant transversal velocity at the wall. It allows defining the most promising injection strategies in terms of wall heat flux reduction and coolant-mass saving. Parametric analyses with respect to different combinations of transpiration strategies and properties of the porous material (porosity, permeability, thermal conductivity etc.) allowed defining the main properties of the prototype
axisymmetric porous nose to be used for the experimental campaign on transpiration cooling that is in preparation at the University of Texas at Arlington (UTA).

The experimental non-intrusive characterization of the aforementioned carbon-carbon (C-C) porous structure has also been performed in order to define the blowing properties of the thermal protection system (TPS) prior to the experimental campaign on transpiration cooling in the 1.6 MW arc-heated wind tunnel (AHWT) at UTA. A new methodology is introduced for the calculation of the local effective permeability by using hot-film anemometry coupled to the 3-D computed tomography of the specimen. The asymmetric blowing capability of the cone highlights the importance of characterizing the entire thermal protection system instead of defining the overall properties of the material, which can be drastically different at the full-scale level due to the geometry, the system integration (e.g. boundary constraint on the structure) and the intrinsic defectology coming from the manufacturing process.

The experimental characterization of the high-enthalpy flow downstream the nozzle exit of the AHWT has been carried out by using ablative Teflon® probes in combination with total pressure measurements with the purpose of augmenting the current capabilities for the investigation and qualification of candidate TPS materials. The results obtained in this investigation show the possibility of using inexpensive and rapidly machinable Teflon® probes for the flow characterization of arc-heated wind tunnel facilities by having uncertainties comparable to those characteristics of standard metal heat flux probes and with the added capability of detecting flow non-uniformities by inspection of the ablation pattern on the surface.
TABLE OF CONTENTS

ACKNOWLEDGEMENTS ................................................. iv
ABSTRACT .............................................................. vi
LIST OF ILLUSTRATIONS ........................................... xii
LIST OF TABLES ....................................................... xix

Chapter  Page

1. INTRODUCTION ................................................. 1
   1.1 The Thermal Management of Reusable Hypersonic Vehicles . . . . 1
   1.2 Research Outline and Major Contributions ........................ 4

2. Development of the Reduced Order Model for the Simulation of the Boundary Layer Flow with Transpiration Cooling ........... 8
   2.1 Literature Review on Boundary Layer Solution Procedures ......... 9
   2.2 Modeling of the Boundary Layer Flow ............................ 11
   2.3 Numerical Solution (AERO-Code) .................................. 14
   2.4 Variable Transpiration Cooling ..................................... 19
       2.4.1 Variable Transpiration using Constant Mass-Flow Rate ........ 20
       2.4.2 Variable Transpiration using Different Mass-Flow Rates ...... 21
       2.4.3 Mesh Sensitivity Analysis and Validation of AERO-Code .... 22
   2.5 Results ..................................................................... 27
       2.5.1 Variable Transpiration using Constant Mass-Flow Rate ....... 27
       2.5.2 Variable Transpiration using Different Mass-Flow Rates .... 30
   2.6 Conclusions ........................................................ 32

3. Integrated Analysis of the Transpiration Cooling Technique ....... 34
3.1 Mathematical Models and Numerical Solutions .................................. 34
  3.1.1 Mathematical Model and Numerical Solution of the Boundary-
         Layer Flow ......................................................... 35
  3.1.2 Mathematical Model and Numerical Solution of the Fluid Flow
         and Heat Transfer Through the Porous Material ................. 36
3.2 Integrated Analysis of Transpiration Cooling ................................. 43
  3.2.1 Flat Plate Geometry .............................................. 44
  3.2.2 Two-Dimensional Body with Blunt Leading Edge .................... 51
3.3 Matching Procedure of the Integrated Analysis ............................... 56
3.4 Parametric Analysis with Respect to the Thermophysical Properties
         of the Material ....................................................... 59
  3.4.1 Parametric Analysis with Respect to the Porosity ................. 60
  3.4.2 Parametric Analysis with Respect to the Thermal Conductiv-
         ity of the Porous Material ....................................... 61
3.5 Conclusions ................................................................................. 62

4. Characterization of Complex Porous Structures for Reusable Thermal Pro-
   tection Systems ........................................................................... 66
  4.1 Introduction ................................................................................. 67
  4.2 Basic Concepts for Permeability Measurements ............................ 68
  4.3 Measurements Methodology and Data Analysis ............................. 74
    4.3.1 Carbon-Carbon Cone Description .................................... 75
    4.3.2 Hot-Film Anemometry ..................................................... 76
    4.3.3 Surface Blowing Measurements ....................................... 78
    4.3.4 Characterization of the Internal Porous Structure .............. 86
  4.4 Results and Comparisons .......................................................... 88
    4.4.1 Porosity Measurements ................................................... 89
4.4.2 Dimensions of the Probed Area .............................. 94
4.4.3 Hot-Film Probe Distance from the Wall ................. 98
4.4.4 Effective Permeability Calculation ......................... 107
4.5 Conclusions .................................................... 113

5. Intrusive Flow Characterization of the Plasma Wind Tunnel Facility at UTA by using Teflon® Probes in Combination with Total Pressure Measurements .................................................. 116
5.1 Introduction ..................................................... 117
5.2 Theoretical Background ....................................... 120
  5.2.1 Ablative Properties of Teflon® .......................... 120
  5.2.2 Control Volume Analysis .................................. 123
  5.2.3 Simplified Version of the Stagnation Point Convective-Heating Equation ...................... 126
  5.2.4 The Blockage Factor and Dimensional Analysis of the Ablation Mechanism ..................... 128
5.3 Proposed New Correlations for the Blockage Factor ............ 130
5.4 Experiment Design .............................................. 134
5.5 Facility Description and Experimental Setup .................... 137
5.6 Results .......................................................... 140
  5.6.1 Teflon® Probing ......................................... 140
  5.6.2 Pitot Probing ............................................. 147
5.7 Data Analysis and Comparisons ................................ 148
  5.7.1 Control Volume Analysis using the Proposed Correlation for the Blockage Factor ........... 148
  5.7.2 NASA Correlation ....................................... 151
  5.7.3 Numerical Simulations .................................... 152
5.8 Conclusions .................................................. 154

6. Conclusions and Future Work ............................. 158
   6.1 Conclusions ............................................. 158
   6.2 Future Work ........................................... 163

REFERENCES .................................................. 169

BIOGRAPHICAL STATEMENT ................................. 183
# LIST OF ILLUSTRATIONS

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>DDM scheme with implicit formulation [1]</td>
<td>17</td>
</tr>
<tr>
<td>2.2</td>
<td>Wall velocity distributions with constant coolant mass-flow rate</td>
<td>21</td>
</tr>
<tr>
<td>2.3</td>
<td>Wall velocity distributions with variable coolant mass-flow rates</td>
<td>21</td>
</tr>
<tr>
<td>2.4</td>
<td>Grid convergence study for the AERO-Code: Boundary layer profiles.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>a) Self-similar solution using $10^6$ points; b) SSM+DDM using $0.25 \cdot 10^6$ points; c) SSM+DDM using $0.05 \cdot 10^6$</td>
<td></td>
</tr>
<tr>
<td>2.5</td>
<td>Grid convergence study for GASP: Wall heat fluxes [2]</td>
<td>23</td>
</tr>
<tr>
<td>2.6</td>
<td>Wall heat flux comparison between the AERO-Code and GASP predictions</td>
<td>24</td>
</tr>
<tr>
<td>2.7</td>
<td>Skin friction coefficient for two variable transpiration strategies</td>
<td>26</td>
</tr>
<tr>
<td>2.8</td>
<td>Boundary layer thickness comparison between AERO-Code and GASP with...</td>
<td>26</td>
</tr>
<tr>
<td></td>
<td>without transpiration</td>
<td></td>
</tr>
<tr>
<td>2.9</td>
<td>Temperature and velocity profiles across the boundary layer thickness</td>
<td></td>
</tr>
<tr>
<td></td>
<td>at $x = 0.7$ m. a) Temperature profiles; b) Velocity profiles</td>
<td>27</td>
</tr>
<tr>
<td>2.10</td>
<td>Comparison of the boundary layer profiles with constant coolant mass-flow rate</td>
<td>28</td>
</tr>
<tr>
<td>2.11</td>
<td>Wall heat flux comparison when using constant (total) coolant mass-flow rate</td>
<td>28</td>
</tr>
<tr>
<td>2.12</td>
<td>Temperature profiles across the boundary layer at the end of the flat plate. a) Complete profiles; b) Zoom at the wall</td>
<td>29</td>
</tr>
</tbody>
</table>
2.13 Comparison of the boundary layer profiles with variable coolant mass-flow rates ............................................. 30
2.14 Wall heat flux comparison with variable coolant mass-flow rates ............................................................... 31
3.1 Schematics of the porous material. a) Flat plate; b) 2-D blunt body ................................................................. 35
3.2 CFD grid (fine level) used for the blunt-body calculations [3] ................................................................. 37
3.3 Schematic representation of the porous material with the main parameters involved in the thermo-fluid analysis .................................................. 37
3.4 Schematic of the porous material’s structure ................................................................. 39
3.5 Schematic of an annular sector of the leading edge ................................................................. 42
3.6 Wall velocity distribution used for the simulations of the flat plate. a) Constant wall velocity; b) Sawtooth wall velocity .................................................. 43
3.7 Wall velocity distribution used for the simulations of the blunted body. a) Constant wall velocity; b) Sawtooth wall velocity .................................................. 44
3.8 AERO-Code results. a) Boundary layer thickness; b) Wall heat flux .................................................. 46
3.9 Wall heat flux profiles calculated with the LAURA code .................................................. 46
3.10 Boundary-layer simulations performed with the LAURA code. a) Contour plot of pressure inside the shock layer; b) Pressure profiles at the wall .................................................. 47
3.11 Chosen locations to plot the MAT-Code results. a) Flat plate profile; b) Sawtooth wall velocity .................................................. 48
3.12 Temperature of the coolant across the material’s thickness. a) Uniform transpiration; b) Variable transpiration .................................................. 49
3.13 Temperature of the material across the material’s thickness. a) Uniform transpiration; b) Variable transpiration .................................................. 50
3.14 Pressure of the coolant across the material’s thickness. a) Uniform transpiration; b) Variable transpiration .................................................. 51
3.15 Pressure drop along the flat plate. a) Uniform transpiration; b) Variable transpiration ................................................................. 52

3.16 LAURA code results [3]. a) Pressure at the wall; b) Wall heat flux . . 52

3.17 Velocity contours around the 2-D blunted body (LAURA code simulations) [3]. a) No-transpiration ($V_W = 0 \, m/s$); b) Variable transpiration (Sawtooth wall velocity distribution) ........................................... 53

3.18 Chosen locations to plot the MAT-Code results. a) Blunt body profile; b) Sawtooth wall velocity distribution ........................................... 54

3.19 Temperature of the coolant across the material’s thickness. a) Uniform transpiration; b) Variable transpiration ........................................... 55

3.20 Pressure of the coolant across the material’s thickness. a) Uniform transpiration; b) Variable transpiration ........................................... 56

3.21 Pressure drop along the blunted body surface. a) Uniform transpiration; b) Variable transpiration ........................................... 57

3.22 Updated temperature of the coolant across the material’s thickness. a) Uniform transpiration; b) Variable transpiration ........................................... 59

3.23 Updated pressure drop along the flat plate. a) Uniform transpiration; b) Variable transpiration ........................................... 60

3.24 Blowing strategies at the stagnation point for different values of the material’s porosity. a) Coolant temperature; b) Pressure drop ........................................... 61

3.25 Blowing strategies at the stagnation point for different values of the material’s thermal conductivity. a) Coolant temperature; b) Pressure drop ........................................... 62

4.1 Manufacturing process of the C-C specimen. a) intermediate stage (after layup); b) final stage (after pyrolysis and surface finishing) ... 75
4.2 Assembly of the C-C cone used for the material characterization. a) Section of the CAD model; b) Assembled specimen ................. 76
4.3 Hot-film output voltage for different frequencies of the wind tunnel motor ............................................................... 77
4.4 Calibration curve of the hot-film .................................................. 78
4.5 Test setup for permeability measurements. a) Assembly; b) Enlarged view of the probed area .......................... 79
4.6 Schlieren photography using nitrogen as coolant. a) Average field \( \dot{m}_{N2} = 0.3 \, g/s \); b) Instantaneous image \( \dot{m}_{N2} = 0.7 \, g/s \) .... 80
4.7 Schlieren photography by using helium as coolant (average field). a) Original image; b) Contrast mask ......................... 81
4.8 Longitudinal planes probed for the permeability tests. a) Planes surveyed; b) Control points on each plane ................. 82
4.9 Hot-film orientation at the stagnation point \( d_w = 0.05 \, in \). a) SP-Horizontal; b) SP-Vertical ............................................. 83
4.10 Pressure measured in the external plenum for some of the air flow-rate utilized .......................................................... 84
4.11 Preliminary permeability tests at \( d_w = 0.05 \, in \) from the wall. a) Linear curve fits for the gathered data; b) Hot-film velocity measurements at \( CP7 \) ................................................................. 85
4.12 Longitudinal cut of the 3-D reconstruction. a) 0° Cut-plane; b) 180° Cut-plane (both cut angles are referred to Fig. 4.8-a) ........ 87
4.13 Transversal cut of the 3-D reconstruction. a) Constant-thickness region \( (H = 0.15 \, in) \); b) Variable-thickness region \( (H = 0.13 \, in) \); c) Nose-tip region \( (H = 0.1 \, in) \) ................. 87
4.14 3-D reconstruction of the nose region having variable thickness. a) Axisymmetric view; b) Longitudinal cut; c) Transversal cut 88
4.15 Interrogation areas used to calculate the void-to-material ratio. a) Transversal cut plane; b) Longitudinal cut plane 89
4.16 Comparison of different thresholding methods applied to a portion of material 91
4.17 Zoom of the gray-levels histogram for a selected region of the 3-D reconstruction 93
4.18 RES calculation for two representative longitudinal cut planes 95
4.19 REV calculation [4]. a) Constant-thickness region; b) Variable-thickness region 96
4.20 Hot-film voltage traces across CS9 ($\dot{m}_{N2} = 16$ SLM). a) 0.05 in backward to CP9; b) center of the CS; c) 0.05 in forward to CP9 97
4.21 Network elements detection from the CT-scan image 99
4.22 Dimensions of void structures for the longitudinal cut at 230°. a) Caverns’ diameter; b) Throats’ diameter; c) Throats’ length; d) Pores’ diameter; e) Pores’ length 100
4.23 Statistical distribution of the voids diameter 101
4.24 Probability density functions corresponding to a Gaussian distribution for throats’ and pores’ diameters 102
4.25 Velocity measurements at CP8 for the 270° longitudinal cut ($\dot{m}_{air} = 10$ SLM). a) $d_w = 0.05$ in; b) $d_w = 0.1$ in; c) $d_w = 0.15$ in 105
4.26 Velocity measurements at CP8 for the 270° longitudinal cut ($\dot{m}_{air} = 16$ SLM). a) $d_w = 0.05$ in; b) $d_w = 0.1$ in; c) $d_w = 0.15$ in 105
4.27 Linear curve fit (Darcy’s law) at different probing distances from the wall. a) Correct distances; b) Constant \( d_w = 0.1 \) \( in \) from the wall; c) Constant \( d_w = 0.15 \) \( in \) from the wall ........................................ 106
4.28 Local velocity measurements collected at the center of the CP and for the 90° longitudinal cut. a) Region above the graphite cone (CP1); b) Region above the internal plenum chamber (CP7); c) SP region probed vertically ................................................................. 108
4.29 Darcy’s law trendlines for the selected CP surveyed (referred to Fig. 4.8-b) ................................................................. 109
5.1 Moving control volume on the Teflon® specimen ..................... 124
5.2 Blockage factor correlations from literature ................................. 130
5.3 Exponential-law correlation for the blockage factor ...................... 132
5.4 Power-law correlation for the blockage factor ............................. 134
5.5 Samples positioning and shock-expansion patterns for: a) test #1; b) test #2; c) test #3; d) test # 4 ................................................. 137
5.6 Arc-Jet Facility at UTA ...................................................... 138
5.7 The Teflon® probing setup. a) Swinging system; b) Teflon® test sample mounted in insulating graphite cone ................................. 138
5.8 Total pressure probe. a) Un-shielded probe; b) Shielded probe ...... 139
5.9 Starting phases of the AHWT ................................................. 140
5.10 Post test result for test #1. a) In-Test; b) Post-Test .................... 141
5.11 Modified test sample. a) Simulation of the shock/expansion patterns; b) Alignment procedure ................................................. 142
5.12 Expansion ratio for test #1 .................................................... 143
5.13 Post-test Teflon® specimens ................................................ 143
5.14 Mapping pattern used for depth measurements .......................... 144
5.15 Reference measurement used to calculate the regression length of the tested specimens. a) Reference depth of the un-tested specimen; b) Measurement of the mapping disk thickness ........................................ 144

5.16 Regression length of Teflon® samples. a) test #1; b) test #1-A; c) test #2; d) test #3; e) test #4 ........................................ 145

5.17 Pitot tube experiment. a) In-test; b) Post-test .............................. 147

5.18 Updated exponential-law correlation for the blockage factor ............ 151

6.1 Test setup used for measuring the total pressure losses along the coolant line. a) Internal plenum; b) External plenum ....................... 164

6.2 Total pressure losses along the coolant line ................................. 164

6.3 Schematic of the test setup used for the experimental campaign on transpiration cooling ......................................................... 165

6.4 Test setup used for the calibration of the infrared camera. a) Thermocouples rake; b) Thermal image with cone in-focus .................... 166

6.5 Preliminary results obtained from the calibration procedure. a) Thermal snapshot with reflected image in-focus; b) Calibration curve related to thermocouple 2 (Fig. 6.4-a) ........................................ 167
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Wall heat flux comparison between GASP and AERO-Code at three different locations along the flat plate ($T_W = 1000, K$)</td>
</tr>
<tr>
<td>2.2</td>
<td>Skin friction comparison between GASP and AERO-Code at three different locations along the flat plate ($T_W = 1000, K$)</td>
</tr>
<tr>
<td>2.3</td>
<td>Wall heat flux reduction for a selected location on the flat plate ($T_W = 1000, K$)</td>
</tr>
<tr>
<td>2.4</td>
<td>Total coolant mass-flow rate blown into the boundary layer ($T_W = 1000, K$)</td>
</tr>
<tr>
<td>3.1</td>
<td>Total coolant flow rate per unit width</td>
</tr>
<tr>
<td>3.2</td>
<td>Deviation of the blowing conditions for uniform transpiration at $P1$</td>
</tr>
<tr>
<td>3.3</td>
<td>Deviation of the blowing conditions for variable transpiration at $P1$</td>
</tr>
<tr>
<td>4.1</td>
<td>Nominal test conditions used for the permeability tests</td>
</tr>
<tr>
<td>4.2</td>
<td>2-D local and average porosity calculated for sectioning planes cut longitudinal to the C-C mask axis</td>
</tr>
<tr>
<td>4.3</td>
<td>2-D local and average porosity calculated for sectioning planes cut transversal to the C-C mask axis</td>
</tr>
<tr>
<td>4.4</td>
<td>Average volumetric porosity calculated by two additional contouring methods</td>
</tr>
<tr>
<td>4.5</td>
<td>Length-scales separation for the void elements constituting the C-C nose-tip</td>
</tr>
<tr>
<td>4.6</td>
<td>Average number of voids elements per unit RES</td>
</tr>
</tbody>
</table>
4.7 Effective-permeability values for three longitudinal planes surveyed on
the C-C nose tip .......................................................... 110
4.8 Maximum percent standard deviation on the effective permeability . 112
5.1 Nominal test matrix used for the experimental campaign .......... 134
5.2 Test conditions calculated after the ablation tests ................. 144
5.3 Average percentage deviation of the outer and inner diameter regres-
sion with respect to the stagnation point ................................ 146
5.4 Test conditions and results calculated for the Pitot tests .......... 148
5.5 Average effective radius of the Teflon® specimens .............. 149
5.6 Stagnation enthalpy and relative quantities calculated using the control
volume approach ........................................................... 149
5.7 Stagnation enthalpy and related quantities calculated using the NASA
correlation ................................................................. 152
5.8 Bulk total enthalpy and relative quantities calculated using the nu-
merical code ............................................................. 153
5.9 Stagnation heat flux and stagnation enthalpy deviations with respect
to CV analysis using the new correlation ($R^2 = 0.67$) .......... 153
CHAPTER 1
INTRODUCTION

1.1 The Thermal Management of Reusable Hypersonic Vehicles

Many challenges must be overcome to enable sustained hypersonic flights. Among these, the management of heat loads and of the surface temperatures has to be mentioned. The viscous dissipation of the high-enthalpy flow across the boundary layer can increase the flow temperature to levels that exceed the thermomechanical limits of current materials. The surface temperatures can be further increased by the interaction of the dissociated flow with the vehicle’s surface (catalytic effects). Neither current carbon-based nor ceramic-based materials can sustain the generated extreme surface temperatures for prolonged exposure times. In this harsh environment, the cooling of the exposed structure (external flows as well as internal flows as those typical of scramjet combustors) is a stringent requirement. The transpiration cooling technique is considered in this work because, in previous numerical studies by Eckert and Livingood [5], its capability of saving coolant mass coupled to the reduced interference with the external flow field, with respect to other active cooling techniques, was favorably assessed. In addition, the issuing phenomenon near the wall generates a protective layer of coolant, which favorably acts on preventing active oxidation phenomena that are known to quickly deteriorate carbon-carbon (C-C)/silicon-carbide materials (SiC), two of the most promising candidate materials for reusable thermal protection systems (TPSs), when particular combinations of elevated temperatures and low partial pressures of oxygen occur at the surface. The transpiration mechanism is based on the issuing of a coolant fluid from a porous
surface into the boundary layer with the purpose of decreasing the material temperature to a prescribed level. Both the skin friction and the thermal stress are reduced when the wall heat flux is decreased because of the effects of the injected mass into the boundary layer (blockage effect). Numerical investigations by Glass et al. [6] and Bucchi et al. [7] defined a methodology to couple the boundary-layer flow to the thermal response of the porous material by assuming constant wall heat flux. These studies were focused, respectively, on investigating the potential of using the natural porosity of refractory composite materials to blow hydrogen fuel from the combustor’s walls [6] as well as on exploring the possibility of cooling down the throat region of a rocket nozzle [7]. The key role of the aforementioned coupling between the boundary layer flow and the thermal response of porous materials has been also experimentally emphasized, first, by Martin Marietta Aerospace [8] and, later, by the DLR (German aerospace center) [9]. The former experimental campaign was focused on verifying the feasibility of using transpiring porous tungsten as thermal shield of axisymmetric sphere-cone nose-tips used for an advanced interceptor missile (AIM) system. The tests, conducted in both the Wright-Patterson 50 MW plasma arc facility and in the Martin Marietta ramburner facility, showed the feasibility of the transpiration cooling concept but pointed out the necessity of improving the prediction capability about the flow/material-response interaction in order to provide the correct coolant flow rate required to guarantee the thermostructural integrity of the test samples. In 2008, Reimer et al. [9] and Forest et al. [10] performed two different experimental campaigns in the arcjet facilities at DLR aimed to prove the effectiveness of the transpiration cooling technique in terms of reducing the surface temperatures for C-C flat-faced specimens [9] and ceramic axisymmetric samples [10]. Two different gaseous coolant fluids (nitrogen and argon) and liquid water were blown using arbitrary flow rates. The formation of ice around the test
samples, detected when using liquid water as coolant, remarked the necessity of coupling the boundary layer flow with the thermal response of porous materials with the purpose of building up an efficient TPS based on transpiration cooling. Additionally, the temperatures map from the infrared termography of the conical sample showed the presence of non-uniform cooling during the experiments [10]. Specifically, during the filling time of the specimen, the stagnation-point region cooled down slowly with respect to the straight sidewall of the cone because of the higher external pressure that lowered the driving force needed to push the water outside the pores [10]. The non-uniform distribution of the cooling power due to the pressure field of the external flow can be even accentuated if candidate TPS materials having highly anisotropic void structures and variable thickness are considered. The accurate knowledge of the local blowing capability is, thus, required to avoid hot-spots on the exposed surfaces during the TPS-qualification tests, which are caused by insufficient coolant flow rate, and therefore it is fundamental to improve the prediction capability for the thermo-mechanical response of TPS based on transpiration cooling. It is clear from the literature review presented above that some fundamental questions still require an answer in order to enable the practical use of the transpiration cooling for reusable hypersonic vehicles.

In specific, for prescribed flight conditions, is it possible to define an optimum transpiration strategy that allows saving coolant fluid once the required surface temperature is imposed? Can the blowing profiles simulated numerically be reproduced by using the state of the art manufacturing process of porous materials (i.e. variable material’s thickness, tailored porosity, variable thermal conductivity, diversified surface finishing etc.)? and, more in general, is it possible to define a methodology for the design of reusable TPSs based on transpiration cooling starting from the prelimi-
nary definition of the optimum blowing profile and material’s properties up to the successful test of a full-scale demonstrator?

Additionally, other questions about the aerothermal response of the TPS in ground testing facilities have to be posed based on the following considerations. The Mach number, flight duration, flight altitude, Reynolds number, surface temperature, and gas-surface interaction effects are some of the desired parameters to be matched in order to reproduce the aerothermal environment encountered in real flight conditions. However, existing ground testing facilities are not able to simultaneously reproduce all the aforementioned parameters; therefore, test conditions must be expertly selected in order to replicate some of those parameters with the purpose of studying the physics of interest. Moreover, arc-heated facilities introduce flow disturbances and contaminations (e.g., non-uniformities arising from the arc stabilization techniques and debris due to the erosion of the parts exposed to the hot flow within the heater) unlike the free stream flow in real flight conditions. In this scenario, the accurate knowledge of the flow generated by these facilities is crucial for the development of high-enthalpy flow/TPS surface interaction models and, thus, to have a satisfactory answer to the questions previously posed.

1.2 Research Outline and Major Contributions

The entire research work presented here is thought to provide an answer to some aspects of the questions arisen in §1.1. The first part of this work (Chap. 2 and Chap. 3) is focused towards the development of reduced order models to couple the boundary layer flow to the thermal response of porous materials. In particular, the Navier-Stokes equations written for stationary, non-reacting, hypersonic laminar boundary layers are solved and, then, the governing equations for the heat transfer and fluid-flow across the selected porous medium are solved. The wall heat flux along
the body’s surface is, first, calculated once the transpiration wall velocity profile and the required wall temperature are imposed (AERO-Code in Chap. 2) [2, 11]. Second, these parameters are coupled with the materials thermal response (MAT-Code) [3, 12], enabling the possibility to investigate the cooling effectiveness for different heat load conditions and different transpiration strategies including the thermostructural limits of state of the art porous materials (Chap. 3). The resulting capability of manipulating the boundary layer profile and the consequential impact on the heat transfer rates at the wall allows investigating the benefits and the potential of specific distributions of the cooling power along the body’s surface. In fact, the cooling effects of the transpiration are partially transferred downstream of the particular blowing point considered, and they can be managed on downstream locations by using a lower amount of coolant. This particular feature has been numerically investigated by introducing the new concept of the variable transpiration cooling (Chap. 2). The variable transpiration is considered by implementing selected distributions of the transversal wall velocity of the coolant fluid. The numerical analyses highlight the variable transpiration using a sawtooth wall velocity distribution to be able saving about 37% of coolant mass with respect to the other transpiration strategies analyzed. Two different configurations of interest for hypersonic applications have been considered for the coupled aero-material analysis: a blunt body with cylindrical leading edge and a flat plate (Chap. 3). Parametric analyses with respect to the thermal properties and geometrical characteristics of the porous material have been performed with the double purpose of:

- investigating the influence of different manufacturing variables on the overall thermostructural response of the TPS;
- defining the nominal values and distributions of porosity, thermal conductivity and thickness of the prototype C-C conical nose to be used for the experimental
campaign aimed to assess the cooling effectiveness, determined numerically, of the variable transpiration cooling (Chap. 6).

Chapter 4 presents the material characterization in terms of local blowing capability of the above mentioned prototype C-C nose [13, 4]. Specifically, the coolant (air) mass-flux blown from a conical porous surface is measured by a hot-film anemometer at a distance specified by an appropriate reference elementary area and the Reynolds number based on the characteristic channels’ diameter. These measurements are then related to the pressure gradient across the local material’s thickness by using Darcy’s law. The new technique proposed here for the non-destructive characterization of full-scale components/structures in terms of local blowing capability is fundamental because the presence of intrinsic defects, due for example to the manufacturing processes, can induce asymmetric flow paths and non-uniform heat-transfer coupling between the coolant fluid and the porous matrix. The aforementioned phenomena can generate, respectively, concentrated mechanical loads and hot-spots on the exposed surfaces which can drastically modify the nominal thermomechanical response of the entire TPS. In addition, the knowledge of the actual blowing distribution represents a fundamental information (boundary conditions) for the numerical aerothermal reconstruction of the transpiration cooling experiment (Chap. 6).

Chapter 5 includes the experimental characterization of the high-enthalpy flow generated by the 1.6 MW arc-heated wind tunnel (AHWT) facility residing at the University of Texas at Arlington (UTA) [14]. In particular, an innovative method based on the use of Teflon flat-faced ablative probes coupled to total pressure measurements has been used for calculating the distribution of both the stagnation heat flux and stagnation enthalpy along the centerline of the high-enthalpy flow. A par-
allel analytical work, focused on the dimensional analysis of the ablation process, is also approached with the purpose of improving existing semi-empirical correlations for the heat blockage due to the mass injection inside the boundary layer. A considerable improvement, with respect to the correlations available in literature, has been obtained once including the pressure dependence inside the non-dimensional parameters. The results obtained in this investigation show the possibility of using inexpensive and rapidly machinable Teflon probes for the flow characterization of arc plasma facilities by having uncertainties comparable to those characteristics of standard metal heat-flux probes. The additional capability of detecting relevant ablation processes makes ablative probes an interesting tool to assess the flow uniformity downstream the nozzle exit by inspecting the "footprint" of the flow on the exposed surfaces.
CHAPTER 2

Development of the Reduced Order Model for the Simulation of the Boundary Layer Flow with Transpiration Cooling


In the first section of this chapter (§2.1), a brief overview about the techniques used to solve the boundary layer flow by using reduced order models is reported. The mathematical model used in this work to simulate the hypersonic boundary layer with transpiration at the wall is defined in the second section (§2.2). The third section (§2.3) is focused towards the numerical solution of the governing equations in §2.2 along with the description of the new approach used to simulate the transpiration cooling. Section §2.4 is entirely dedicated to the set-up of the numerical investigations and to the mesh sensitivity analysis and validation of the reduced order model by using the computational fluid dynamics (CFD) code GASP. The main results of the numerical investigations are reported in §2.5.
2.1 Literature Review on Boundary Layer Solution Procedures

Many solution methods have been proposed to solve the boundary layer flow after the first theorization developed by Prandtl (1904) [16]. In general, two main approaches can be used to solve the boundary layer equations: the integral method and the differential method. The integral method solves the Navier-Stokes equations in the integral form by applying the conservation principles to a control volume that is finite in the transverse direction (where the variation of the thermodynamic properties is more important), and differential in the streamwise direction. The differential method applies the conservation principles to a differential volume of fluid. The integral method is computationally less costly compared to the differential method, but it is not recommended for an accurate estimation of quantities such as the skin friction or wall heat flux. This is because the integration process across the boundary layer (BL) does not provide the piecewise distribution of thermodynamic properties across the BL and only the boundary conditions (BCs) are taken into account in the solution process. When using the integral method, polynomial expressions for the velocity (Pohlhausen method) [16, 17] have to be considered in order to extract velocity or temperature profiles across the boundary layer thickness ($y$-direction). Otherwise, one can consider non-dimensional functions to simplify the problem and solve the integral equations (Thwaites-Walz method) [16, 17, 18]. This type of transformation cannot extract the variation of thermodynamic properties along the $y$-direction. The integral method is a good approach for a rapid estimation of the wall heat flux, wall temperature, and skin friction for a first order calculation. In hypersonic flow fields, where a detailed description of the flow inside the boundary layer is necessary, the differential method is preferred. The differential methods are mathematically more complex with respect to the integral methods, but can solve the boundary layer flow without assumptions on the initial profile of the thermodynamic variables. They
enable the computation of the velocity and temperature profiles allowing the subse-
quently calculation of the skin drag and the wall heat transfer. The numerical solution
of the BL equations in the differential form can be approached by using Self-Similar
Methods (SSM) or Difference Differential Methods (DDM). The SSM transforms the
Cartesian space \((x;y)\) to a dimensionless space \((\xi;\eta)\) by means of selected coordinate
transformations (i.e. Illingworth-Levy or Levy-Lees) [1, 18]. For particular cases,
under specific coordinate transformations, the flow field becomes independent of the
streamwise direction, \(x\) (i.e. \(u = u(\eta), T = T(\eta), \text{etc.}\) and the flow field solution
is named as self-similar. Self-similar solutions can be extracted, first, for the flat
plate geometry and incompressible flow (Blasius solution) [1, 16, 17, 18] and then
the Chapman-Rubesin coefficient, \(C\) in Eq. 2.1 [17], can be introduced to solve the
compressible BL flow.

\[
C = \frac{\rho U}{\rho_e U_e} \tag{2.1}
\]

Where \(\rho\) and \(U\) represent, respectively, the density and longitudinal velocity
while the subscript \(e\) indicates the external conditions. The \(x\)-momentum and energy
equations for compressible flows in the transformed space are reported in Eq. 2.2.

\[
\begin{align*}
(Cf'' + ff')' &= \frac{2\xi}{U_e} \left[ (f')^2 - \frac{\rho_e}{\rho} \right] \frac{dU_e}{d\xi} + 2\xi \left( f' \frac{\partial f'}{\partial \xi} - \frac{\partial f}{\partial \xi} f'' \right) \\
(\frac{C}{Pr}g')' + fg' + Cg \left( f' \right)^2 &= 2\xi \left[ f' \frac{\partial g}{\partial \xi} + f' \frac{\partial h_e}{\partial \xi} - g' \frac{\partial f}{\partial \xi} + \frac{\rho_e U_e}{\rho_e U_e} f' \frac{\partial U_e}{\partial \xi} \right]
\end{align*} \tag{2.2}
\]

Where \(f\) and \(g\) are the service functions used for the coordinate transformation.
Equation 2.2 is valid for any geometry and with the assumption that the transversal
pressure remains constant in the boundary layer \((dP/dy = 0)\). Another operation is
then needed to re-transform the \((\xi;\eta)\) space into \((x;y)\) space [1, 16], after calculating
the solution in terms of the service functions \(f\) and \(g\). The SSM has the main advan-

tage of transforming the partial differential equations (PDEs) to ordinary differential equations (ODEs) that can be easily solved. Conversely, the DDM directly solves the PDEs by discretizing, first, the boundary layer domain (i.e. substitution of the \( \partial \) operator with \( \Delta \) operator) and, then, by using a combination of forward, upward and central finite differences [15, 19].

2.2 Modeling of the Boundary Layer Flow

The modeling of the BL flow is approached in this work using the differential method because of the necessity to calculate the fluid dynamics variables across the BL, as mentioned in §2.1. In particular, the model solves the 2-D laminar Navier-Stokes equations written for a stationary, non-reacting hypersonic boundary layer on a flat plate for which the radiative thermal exchange can be also neglected. Moreover, the thermophysical properties of the gas are assumed to be constant with pressure (thermally perfect gas) and are calculated at a reference temperature, \( T^* \), defined by Eckert [17]. The basic assumptions used for the modeling of the BL are summarized below.

- 2-D Flow
- Laminar BL
- \( \partial/\partial t = 0 \) \hspace{1cm} (steady-state condition)
- \( \dot{\omega} = 0 \) \hspace{1cm} (zero production rate of species)
- \( C_p(T) = C_p(T^*), \ \mu(T) = \mu(T^*), \ k(T) = k(T^*) \)

Where \( C_p, \mu, k \) are the specific heat at constant pressure, dynamic viscosity and thermal conductivity of the BL flow, respectively. The order of magnitude analysis of the mass, momentum and energy transportation equations allows defining the simplified governing equations for hypersonic BLs (Eq. 2.3), once the additional hypotheses of thin boundary layer and high Reynolds number are considered [1, 16].
\[
\begin{align*}
\frac{\partial (\rho U)}{\partial x} + \frac{\partial (\rho V)}{\partial y} &= 0 \\
\rho \left[ U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} \right] &= -\frac{\partial P}{\partial x} + \frac{\partial}{\partial y} \left( \mu \frac{\partial U}{\partial y} \right) \\
\frac{dP}{dy} &= 0 \\
\rho U \frac{\partial H}{\partial x} + \rho V \frac{\partial H}{\partial y} &= \frac{\partial}{\partial y} \left[ \mu \frac{\partial}{\partial y} \left( \frac{h}{Pr} + \frac{U^2}{2} \right) \right] + \frac{\partial}{\partial y} \left[ \frac{\mu Le}{Pr} \left(1 - \frac{1}{Le}\right) \sum_i h_i \frac{\partial C_i}{\partial y} \right] \\
\rho U \frac{\partial C_i}{\partial x} + \rho V \frac{\partial C_i}{\partial y} &= \frac{\partial}{\partial y} \left( \frac{\mu Le}{Pr} \frac{\partial C_i}{\partial y} \right) \quad i = 1, 2 \ldots N \\
h &= C_p T \\
P &= \rho RT
\end{align*}
\]  

Where \( V, h, H \) and \( C_i \) represent, respectively, the transversal velocity, static enthalpy, total enthalpy and species concentration inside the BL. It is noteworthy to mention that the equation of the pressure gradient through the BL thickness \( (dP/dy = 0) \) is valid until the term \( 1/(\gamma M^2_{\infty}) \) becomes comparable to the BL thickness, \( \delta \). So, it is no longer valid for very large Mach numbers [16]. \( NS \) in the species transportation equation represents the number of species constituting the boundary layer flow. \( NS > 1 \) allows considering different coolant fluids with respect to the incoming freestream flow in the hypersonic boundary layer equations (Eq. 2.3) [17]. Indeed, it is necessary to consider the transportation of the species due to the diffusion mechanism even though the flow can be considered as non-reacting \( (\dot{\omega} = 0) \).  

In the current study, additional assumptions are made to simplify the numerical solution of the non-linear system of PDEs [14, 15]:

- \( Le=1 \)
- \( Pr=1 \)

The use of unitary Lewis number, \( Le \), implies that the flow inside the boundary layer has the same capacity to diffuse energy and mass. For this reason, air has been chosen for this study. The assumption on unitary Prandtl number, \( Pr \), is not strictly
valid for air \((Pr_{air} \cong 0.75)\) but it can be accepted without significance loss of accuracy since the temperature profiles do not change considerably [1]. The Crocco-Busemann relation [20, 21] can be applied to both the updated energy and species transport equations leading to a substantial simplification from non-linear PDEs to a linear system composed of two PDEs and six algebraic equations (Eq. 2.4).

\[
\begin{align*}
\frac{\partial (\rho U)}{\partial x} + \frac{\partial (\rho V)}{\partial y} &= 0 \\
\rho \left[ U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} \right] &= -\frac{\partial P}{\partial x} + \frac{\partial}{\partial y} \left( \mu \frac{\partial U}{\partial y} \right) \\
\frac{dP}{dy} &= 0 \\
h &= -\frac{U^2}{2} + \left( h_e - h_W + \frac{U^2}{2} \right) \frac{U}{U_e} + h_W \\
C_1 &= \left( \frac{C_{1e} - C_{1W}}{U_e} \right) U + C_{1W} \\
C_1 + C_2 &= 1 \\
h &= C_p T \\
P &= \rho RT
\end{align*}
\]

(2.4)

Where \(C_1\) and \(C_2\) are the concentration of air and coolant, respectively while \(W\) subscript is indicative of the conditions at the wall. The additional assumption of flat plate geometry has been considered in Eq. 2.4 \((\partial P/\partial x = 0)\). In this analysis, the transpiration of air into the boundary layer is investigated. This assumption implicitly leads to a single species to be considered \((NS = 1)\) because, in this case, only air is present in the boundary layer [15]. However, a two-species model has been implemented in the system of equations presented in Eq. 2.4. This allows choosing different transpiration fluids \((NS = 2)\) for future work. The boundary conditions associated to the system of equations in Eq. 2.4 are presented in Eq. 2.5.
\begin{equation}
\begin{aligned}
P(x, 0) &= P_c \\
T(x, 0) &= T_W(x) \\
\rho(x, 0) &= \frac{P_c}{RT_c} \\
U(x, 0) &= 0 \\
V(x, 0) &= V_W(x)
\end{aligned}
\end{equation}

In literature, the transpiration cooling is taken into account as a boundary condition by means of the product of the density and the transversal velocity calculated at the wall [1, 18] \((\rho_W V_W)\). Considering a non-reacting flow, the resulting wall density is constant if a constant wall temperature is assumed and considering that the pressure is imposed by the external conditions (third equation in Eq. 2.4). In this scenario, the transpiration cooling is taken into account only by the transversal velocity at the wall \((V_W)\). It is inherently assumed that the exit temperature and pressure of the coolant fluid are the same of those at the wall. This assumption highlights the importance of coupling the material response to the BL flow [3, 12]. In fact, it is necessary to know the extent of the thermal exchanges within the porous material [3, 12] in order to estimate the thermodynamic properties of the coolant fluid at the exit of the material and, thus, to correctly predict the right amount of coolant fluid that must be blown into the BL.

2.3 Numerical Solution (AERO-Code)

A MATLAB\textsuperscript{®}-based script named AERO-code has been developed to solve the governing equations of the BL flow in Eq. 2.4. The code involves the use of a mix of central and forward finite differences. It is possible to solve the differential transport equations in Eq. 2.4 by using the SSM, the DDM or a combination of
both. The SSM is preferable to the DDM because, as already mentioned in §2.1, it allows transforming the PDEs in ODEs that can be easily solved by using standard numerical techniques. The SSM is not applied to solve the BL equations with transpiration because the transformation from the \((x; y)\) space to the \((\xi; \eta)\) space requires the introduction of a streamline function. The streamline function assumes a well-known form when mass addition is not considered [1, 15]. An additional function, of which we do not know an explicit expression, has to be modeled when transpiration is considered. In literature, all the numerical investigations regarding transpiration cooling, which involve the use of SSM, consider a binding expression for the transpiration parameter (i.e. transpiration velocity at the wall) that is \(V_W \propto 1/\sqrt{x}\) [1, 22]. This condition on the transversal velocity, which is not useful for general transpiration cases, leads to a self-similar solution which allows the ODE system to be easily solved (Hartnett and Eckert solution [1, 16, 18, 19]). In this work, the DDM has been implemented in order to enable the capability of investigating any type of transpiration strategy. This method needs an initializing solution, in addition to the boundary conditions at the wall and at the outer edge of the BL, due to the parabolic nature of the BL equations. In this investigation, the initial part of the flat plate is maintained without transpiration; however, the bleeding starting from the leading edge could have been implemented by imposing the wall velocity proportional to \(V_W \propto 1/\sqrt{x}\). The starting solution for the DDM is extracted from the self-similar solution for a flat plate with no transpiration and, thus, a coupled SSM and DDM solution is adopted. For blunted bodies, it will be possible to use the integral method (stagnation point solution [16]) to initialize the DDM. The system of equations in the transformed space defined by the SSM is reported in Eq. 2.2 and, with the assumption of flat plate geometry \((dP/dx = 0)\), becomes Eq. 2.6 [16].
\begin{align*}
\begin{cases}
(Cf'')' + ff'' = 0 \\
\left(\frac{C}{Pr}g'\right)' + fg' + C\frac{U_e^2}{h_e}(f'')^2 = 0
\end{cases}
\tag{2.6}
\end{align*}

Where:

\[f' = \frac{\partial f}{\partial \eta} = \frac{U}{U_e} ; \quad g = \frac{h}{h_e}\]

(2.7)

The Chapman-Rubesin parameter at the reference temperature, \(C(T^*)\), is calculated by using Sutherland’s law (Eq. 2.8) [20, 23].

\[\mu = \mu_e \left(\frac{T^*}{T_e}\right)^{3/2} \left(\frac{T_e + S}{T^* + S}\right)\]

(2.8)

Where \(S = 111 \text{ } K\) for air. The boundary conditions in the transformed space are reported in Eq. 2.9 once considered the expression of the arbitrary functions in Eq. 2.7.

\begin{align*}
\begin{cases}
f(0) = 0 \\
f'(0) = 0 \\
f'(\eta_\delta) = 1 \\
g(0) = g_w \\
g(\eta_\delta) = 1
\end{cases}
\tag{2.9}
\end{align*}

Where \(\eta_\delta\) is the dimensionless coordinate at the outer edge of the boundary layer that describes the boundary layer thickness. Additionally, \(g(0) = g_w\) implies a constant wall temperature. A boundary condition on \(g'(0)\) has to be imposed if a condition on the wall heat flux is required. The boundary conditions on \(f'(\eta)\) and \(g(\eta)\) are valid for sufficiently high values of \(\eta\). The above formulation is called a boundary value problem (BVP) because the BCs are not uniformly distributed but
are supplied in different points of the domain. In this case, a fifth-order BVP having five BCs is presented. Three BCs are given at the wall and two at the outer edge of the boundary layer. The shooting technique [15, 16] has been used to solve the BVP. The system in Eq. 2.4 has been implemented by using an implicit method to avoid numerical instability, after solving the compressible BL without transpiration for the initial portion of the flat plate. A combination of forward and central finite differences [19, 20] has been used.

The implicit scheme represented in Fig. 2.1 solves the equations (at the $m$ point) using the other unknowns (at the $m-1$ and $m+1$ grid points) and a starting solution ($n$-station). It does not generate any numerical instability by changing the mesh size but it is computationally more expensive and complicated with respect to an explicit method. The non-linear term of the $x$-momentum equation in Eq. 2.4 becomes Eq. 2.10 [19].

$$U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} \approx U_{n,m} \left( \frac{U_{n+1,m} - U_{n,m}}{\Delta x} \right) + V_{n,m} \left( \frac{U_{n,m+1} - U_{n,m-1}}{2\Delta y} \right) \quad (2.10)$$

For the $x$-space resolution a one-sided difference has been chosen while for the $y$-space resolution a central difference has been preferred for a best accuracy. A
forward centered difference has been chosen for the discretization of the viscous term (Eq. 2.11).

\[ \mu \frac{\partial^2 U}{\partial y^2} \approx \mu \left( \frac{U_{n+1,m+1} - 2U_{n+1,m} + U_{n+1,m-1}}{\Delta y^2} \right) \]  

(2.11)

The \( x \)-momentum equation can be expressed in term of the consecutive \( x \)-station \((n + 1)\) along the flat plate length (Eq. 2.12).

\[ -\alpha U_{n+1,m-1} + (1 + 2\alpha)U_{n+1,m} - \alpha U_{n+1,m+1} = U_{n,m} - \beta (U_{n,m+1} - U_{n,m-1}) \]  

(2.12)

Where:

\[ \alpha = \frac{\mu \Delta x}{\rho_{n,m} U_{n,m} \Delta y^2} \quad ; \quad \beta = \frac{V_{n,m} \Delta x}{2U_{n,m} \Delta y} \]  

(2.13)

In Fig. 2.1, \( n = 1 \) represents the starting solution deriving from the SSM, \( m = 1 \) represents the wall and \( m = M \) represents the freestream flow. The BCs applied to Eq. 2.12 are reported below.

- No-slip: \( U_{n,1} = 0 \)
- Transpiration at the wall: \( V_{n,1} = V_W \)
- Initial solution computed by the SSM: \( U_{1,m}; V_{1,m} \)

Equation 2.12 represents \((m - 2)\) number of equations, each one containing three unknowns \( U_{(n+1,m-1)}, U_{(n+1,m)}, U_{(n+1,m+1)} \). The set of algebraic equations are written in a tri-diagonal matrix. The tri-diagonal matrix is then inverted using an \( LU \) factorization. An example for \( n = 1, m = 6 \) is reported below in order to clarify the solution method. Equation 2.12 can be rewritten in a matrix form (Eq. 2.14).

\[ A_m U_{n+1,m-1} + B_m U_{n+1,m} + C_m U_{n+1,m+1} = D_m \]  

(2.14)
The algebraic system to be solved is reported below (Eq. 2.15).

\[
\begin{bmatrix}
A_2 & B_2 & C_2 & 0 & 0 \\
0 & A_3 & B_3 & C_3 & 0 \\
0 & 0 & A_4 & B_4 & C_4 \\
0 & 0 & 0 & A_5 & B_5 & C_5
\end{bmatrix} \begin{bmatrix}
U_1 \\
U_2 \\
U_3 \\
U_4 \\
U_5 \\
U_6
\end{bmatrix} = \begin{bmatrix}
D_2 \\
D_3 \\
D_4 \\
D_5 \\
D_6
\end{bmatrix}
\]

(2.15)

The linear system is composed of four equations with four unknowns \((U_2, U_3, U_4, U_5)\) once the BCs are applied \((U_1 = 0, U_6 = U_e)\). The transversal velocity into the boundary layer \((V_{n+1,m})\) is calculated by using the calculated values of the \(U\)-velocity (Eq. 2.15) into the discretized equation of mass transport (Eq. 2.16).

\[
\rho_{n,m} \left( \frac{U_{n+1,m} - U_{n,m}}{\Delta x} + \frac{V_{n+1,m} - V_{n+1,m-1}}{\Delta y} \right) + U_{n,m} \left( \frac{\rho_{n+1,m} - \rho_{n,m}}{\Delta x} \right) + V_{n,m} \left( \frac{\rho_{n+1,m} - \rho_{n+1,m-1}}{\Delta y} \right) = 0
\]

(2.16)

2.4 Variable Transpiration Cooling

In this section, different distributions of the transversal wall velocity along the flat plate have been implemented in order to investigate the cooling effectiveness of the variable transpiration in terms of coolant mass saving and wall heat flux reduction. The first analysis (§2.4.1) aims to determine the effectiveness of selected distributions of the wall velocity in terms of the wall heat flux minimization, given the constraint of the maximum allowable surface temperature that guarantees the thermostructural integrity of the TPS. The total amount of coolant issued into the boundary layer from the material’s surface has been kept constant for each case. Another simulation campaign (§2.4.2) has been carried out with the scope of ana-
lyzing the cooling efficiency of the same velocity distributions in §2.4.1 for different values of the coolant mass-flow rates blown into the boundary layer. The sets of wall velocity distributions used to simulate the variable transpiration cooling for the aforementioned analyses are depicted in Fig. 2.2 and Fig. 2.3. The input parameters for both cases are listed below [2, 11].

- $M_e = 5$  
  Freestream Mach number
- $\rho_e = 1.23 \text{ kg/m}^3$  
  Freestream density
- $T_e = 300 \text{ K}$  
  Freestream temperature
- $T_W = 1000 \text{ K}$  
  Wall temperature
- $L = 1 \text{ m}$  
  Flat plate length
- The transpiration cooling is activated at $x = 0.3 \text{ m from the leading edge}$

2.4.1 Variable Transpiration using Constant Mass-Flow Rate

The peak values of these distributions (Fig. 2.2) take into account the laminar separation of the flow shown in a previous works by the authors [2, 15]. Indeed, transpiration with high blowing velocity generates reversal of the velocity inside the BL (i.e. laminar separation of the flow). Contrarily, suction of air from the BL (negative transpiration velocity) contributes to maintain the flow attached to the plate. The nominal values of the transversal velocity at the wall considered here are consistent with the requirement of keeping the BL attached to the surface.

The coolant mass-flow rate per unit width of the flat plate is maintained constant at $0.0646 \text{ kg/s} \cdot \text{m}$ (Eq. 2.17).

\[
G_{CF} = \int_{0.3}^{1} \rho_W V_W(x) dx = 0.0646 \text{ kg/s} \cdot \text{m} \tag{2.17}
\]
2.4.2 Variable Transpiration using Different Mass-Flow Rates

The same peak value of the wall velocity has been chosen where the transpiration starts (higher heat load with respect to the downstream locations) and different velocity profiles, downstream of the flat plate, have been analyzed to reduce the coolant mass blown. The maximum value of the transpiration velocity for the cases considered here (Fig. 2.3) has been chosen to avoid laminar separation of the flow as explained in §2.4.1.

The coolant mass-flow rate per unit width for each one of the transpiration distribution is reported below:
2.4.3 Mesh Sensitivity Analysis and Validation of AERO-Code

In this section, the sensitivity analysis is performed for different sizes of the mesh in order to assess the grid independence of the results [2]. The first analysis is performed for the input conditions reported above with the exception of the transpiration velocity that is set to zero. Under this assumption, the coupled SSM and DDM solution must collapse into a self-similar solution (Fig. 2.4-a), for all the selected grid levels.

The BL thickness at the end of the flat plate for the coarser grid (Fig. 2.4-b) is only 1.5% lower with respect to the most refined grid (Fig. 2.4-c). The deviation of the BL thickness between the coupling of SSM+DDM using $250 \cdot 10^3$ points and the SSM using $10^6$ points is about 0.2% at the end of the flat plate (comparison of

- $G_{CF, CONSTANT} = 0.0775 \text{ kg/s} \cdot \text{m}$
- $G_{CF, LINEAR} = 0.0517 \text{ kg/s} \cdot \text{m}$
- $G_{CF, SAWTOOTH} = 0.0409 \text{ kg/s} \cdot \text{m}$
- $G_{CF, STEP} = 0.0560 \text{ kg/s} \cdot \text{m}$
- $G_{CF, CONSTANT-LINEAR} = 0.0675 \text{ kg/s} \cdot \text{m}$
- $G_{CF, LINEAR-CONSTANT} = 0.0482 \text{ kg/s} \cdot \text{m}$
In addition, the three-dimensional, structured, finite-volume, Reynolds-averaged Navier-Stokes (RANS) CFD code GASP from AeroSoft Inc. [24] has been used for the high-fidelity numerical solution of the examined cases with and without transpiration in order to validate the AERO-Code results. The numerical simulations by using GASP have been performed by the research group of the Missouri University of Science and Technology. The detailed description of the parameters used for the simulations is reported in [2]. The CFD solutions of the flat plate cases were obtained at three structured grid levels: grid-1 (101 x 181), grid-2 (51 x 91), and grid-3 (26 x 46) to check the grid convergence. The first dimension of the grid is for the streamwise direction ($x$) and the second dimension is for the normal-to-wall direction ($y$). For all the cases, the results (wall heat flux, skin friction, boundary layer thickness, etc.) of grid-2 and grid-1 were almost identical indicating the grid convergence at the medium grid level (i.e., grid-2) (Fig. 2.5). The results presented in this paper have been obtained with the grid-1, which is the finest grid level.

The comparisons of the wall heat flux derived from the AERO-Code and GASP, for the analysis described in the initial part of this section, are presented in Fig. 2.6.
Figure 2.6. Wall heat flux comparison between the AERO-Code and GASP predictions.

The maximum heat flux difference between the two codes, for all the wall velocity distributions, is about 24% near the zone where the transpiration starts ($x = 0.3 \text{ m}$). Downstream of the flat plate the differences are reduced (Table 2.1).

The quantitative comparison of the wall heat flux and skin friction has been performed by calculating the percentage error between the two codes at equidistant locations along the flat plate. The results show that the wall heat flux (Table 2.1) and the skin friction (Table 2.2) have almost the same deviation for the same distribution
Table 2.1. Wall heat flux comparison between GASP and AERO-Code at three different locations along the flat plate ($T_W = 1000 \, K$)

<table>
<thead>
<tr>
<th>$q_W$ deviation, %</th>
<th>$x = 0.3 , m$</th>
<th>$x = 0.65 , m$</th>
<th>$x = 1 , m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant $V_W$</td>
<td>21.53</td>
<td>10.60</td>
<td>3.55</td>
</tr>
<tr>
<td>Linear $V_W$</td>
<td>20.80</td>
<td>9.81</td>
<td>8.50</td>
</tr>
<tr>
<td>Sawtooth $V_W$</td>
<td>21.08</td>
<td>9.27</td>
<td>8.34</td>
</tr>
<tr>
<td>Step $V_W$</td>
<td>21.08</td>
<td>9.49</td>
<td>10.83</td>
</tr>
<tr>
<td>Constant-linear $V_W$</td>
<td>21.08</td>
<td>9.49</td>
<td>10.83</td>
</tr>
<tr>
<td>Linear-constant $V_W$</td>
<td>21.08</td>
<td>14.34</td>
<td>15.35</td>
</tr>
</tbody>
</table>

Table 2.2. Skin friction comparison between GASP and AERO-Code at three different locations along the flat plate ($T_W = 1000 \, K$)

<table>
<thead>
<tr>
<th>$q_W$ deviation, %</th>
<th>$x = 0.3 , m$</th>
<th>$x = 0.65 , m$</th>
<th>$x = 1 , m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant $V_W$</td>
<td>23.59</td>
<td>12.93</td>
<td>4.81</td>
</tr>
<tr>
<td>Linear $V_W$</td>
<td>24.00</td>
<td>12.21</td>
<td>8.20</td>
</tr>
<tr>
<td>Sawtooth $V_W$</td>
<td>23.98</td>
<td>12.20</td>
<td>8.50</td>
</tr>
<tr>
<td>Step $V_W$</td>
<td>24.02</td>
<td>12.78</td>
<td>10.06</td>
</tr>
<tr>
<td>Constant-linear $V_W$</td>
<td>11.88</td>
<td>9.49</td>
<td>9.00</td>
</tr>
<tr>
<td>Linear-constant $V_W$</td>
<td>16.65</td>
<td>14.34</td>
<td>16.27</td>
</tr>
</tbody>
</table>

and at the same locations corroborating the validity of the Reynolds analogy in hypersonic flow [1, 16]. For all the transpiration strategies, the AERO-Code estimates slightly higher wall heat fluxes (Fig. 2.6) and skin friction coefficients (Fig. 2.7)

The differences are attributable to:

1. AERO-Code uses constant thermophysical properties ($C_p, \mu$).
2. The GASP solver captures the shock at the leading edge due to the presence of the boundary layer (viscous interaction). This is evident from the comparison of the kinematic boundary layer profiles (Fig. 2.8). The viscous interaction occurs due to the mutual interdependence between the non-zero displacement thickness at the leading edge of the flat plate and the shock generated at the same location [25].
An additional simulation, performed for lower freestream Mach number ($M = 4$) and by using the same temperature ratio used for previous simulations ($T_W/T_t = 0.5$), corroborated the above mentioned hypotheses used to explain the deviations between AERO-Code and GASP [2]. In fact, for the new flight conditions, the maximum deviation in terms of wall heat flux between AERO-Code and GASP is reduced from 24% to about 8% where the transpiration starts. Moreover, both the longitudinal velocity (Fig. 2.9-a) and temperature (Fig. 2.9-b) profiles inside the BL confirm the capability of AERO-code to capture the main features of the laminar hypersonic BL flowfield over a flat plate.
2.5 Results

The results for the two analyses described in §2.4.1 and §2.4.2 are presented below.

2.5.1 Variable Transpiration using Constant Mass-Flow Rate

In this analysis, the nature of the flow regime is forced to be laminar even if, using the input conditions, the resulting flow would be turbulent ($10^7 < Re_x < 10^8$). However, a separate work has been performed in order to investigate the cooling effectiveness of the variable transpiration in turbulent flow [26]. The imposed wall temperature is specifically maintained at a constant value, generally prescribed by the necessity to preserve the thermostructural integrity of the selected TPS. For this analysis, a value of 1000 $K$ is adopted. The computational time required to solve the equations of the hypersonic laminar boundary layer and to extract the thermal and kinematic boundary layer profiles, as well as all the other thermodynamic properties, is about 30 seconds on a typical personal computer. The BL thickness for the transpiration strategies investigated is reported in Fig. 2.10.
The analysis of the results shows that there is no particular distribution of the wall velocity which minimizes the wall heat transfer along the entire flat plate length since the BL thickness is very similar for all the transpiration strategies considered (Fig. 2.10 and Fig. 2.11).

Zooming on the temperature profile at the wall (Fig. 2.12), it can be inferred that the constant wall velocity generates the lower heat flux at the end of the flat plate because the temperature gradient at the wall is lower with respect to the other transpiration strategies. However, the constant wall velocity does not minimize the heat transfer for $0.4 \, m < x < 0.9 \, m$. In this region, where the heat flux is larger with
Figure 2.12. Temperature profiles across the boundary layer at the end of the flat plate. a) Complete profiles; b) Zoom at the wall.

respect to the end of the plate, the constant-linear wall velocity generates the lower heat flux (Fig. 2.11). Considering only the distributions of Fig. 2.2, the variable transpiration constant-linear is the best strategy to cool down the structure if the same amount of coolant blown into the boundary layer is considered.

In Table 2.3, the evaluation of the wall heat flux reduction is reported. The no-transpiration case ($V_W = 0 \text{ m/s}$) is considered as reference value. For the comparison, an intermediate location on the flat plate is selected ($x = 0.7 \text{ m}$).

Table 2.3. Wall heat flux reduction for a selected location on the flat plate ($T_W = 1000 \text{ K}$)

<table>
<thead>
<tr>
<th>$x = 0.7 \text{ m}$</th>
<th>$q_W$, kW/m²</th>
<th>$\Delta q_W$, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>No-Cooling ($V_W = 0$)</td>
<td>47.47</td>
<td>0.00</td>
</tr>
<tr>
<td>Constant $V_W$</td>
<td>16.04</td>
<td>-66.21</td>
</tr>
<tr>
<td>Linear $V_W$</td>
<td>15.22</td>
<td>-67.94</td>
</tr>
<tr>
<td>Sawtooth $V_W$</td>
<td>14.58</td>
<td>-69.29</td>
</tr>
<tr>
<td>Step $V_W$</td>
<td>14.03</td>
<td>-70.44</td>
</tr>
<tr>
<td>Constant-linear $V_W$</td>
<td>11.03</td>
<td>-76.76</td>
</tr>
<tr>
<td>Linear-constant $V_W$</td>
<td>16.09</td>
<td>-69.04</td>
</tr>
</tbody>
</table>
2.5.2 Variable Transpiration using Different Mass-Flow Rates

Another potential requirement for TPSs based on transpiration cooling includes the identification of variable transpiration strategies for the minimization of the coolant mass (i.e., meaning a reduced amount of coolant fluid to be carried on-board) given the constraint of the maximum allowable surface temperature for the TPS structure. The general strategy adopted here reduces the transpiration velocity at the end of the flat plate which is exposed to the lower thermal loads. The same wall velocity distributions of Fig. 2.2 are selected but using different nominal values (Fig. 2.3). The same value of the transpiration velocity is imposed where the cooling starts ($x = 0.3 \ m \rightarrow V_{W} = 0.3 \ m/s$). The simulations have been performed by using the same input parameters reported in §2.4. The BL profiles for all the transpiration strategies considered (Fig. 2.13) have the same shape of the test case analyzed in §2.5.1 (Fig. 2.10) but the BL thicknesses change accordingly to the nominal values of the wall velocity distributions.

The choice of lower values of the transpiration velocity at the end of the flat plate generates higher wall heat flux (comparison of Fig. 2.11 and Fig. 2.14) but a significant reduction of the total mass-flow rate is obtained (Table 2.4).
Figure 2.14. Wall heat flux comparison with variable coolant mass-flow rates.

The reference value of the mass-flow rate is taken from the variable transpiration analysis using constant mass-flow rate (Eq. 2.17). Table 2.4 shows that the sawtooth wall velocity distribution allows reducing by about 37% the total coolant mass with respect to the other transpiration strategies.

Table 2.4. Total coolant mass-flow rate blown into the boundary layer ($T_W = 1000 \, K$)

<table>
<thead>
<tr>
<th></th>
<th>$G_{CF}, , kg/s \cdot m$</th>
<th>$\Delta G_{CF}, %$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference Value</td>
<td>0.0646</td>
<td>0.00</td>
</tr>
<tr>
<td>Constant $V_W$</td>
<td>0.0775</td>
<td>19.97</td>
</tr>
<tr>
<td>Linear $V_W$</td>
<td>0.0517</td>
<td>-19.97</td>
</tr>
<tr>
<td>Sawtooth $V_W$</td>
<td>0.0409</td>
<td>-36.69</td>
</tr>
<tr>
<td>Step $V_W$</td>
<td>0.0560</td>
<td>-13.31</td>
</tr>
<tr>
<td>Constant-linear $V_W$</td>
<td>0.0482</td>
<td>-25.39</td>
</tr>
<tr>
<td>Linear-constant $V_W$</td>
<td>0.0675</td>
<td>4.49</td>
</tr>
</tbody>
</table>

The higher wall heat flux encountered for the sawtooth wall velocity (Fig. 2.14) is not necessarily transferred to the structure. In fact, as it can be inferred by the coupling between the boundary layer and the materials thermal response (Chap. 3) [3, 12], the heat capacity of the coolant fluid, together with the calculated mass-flow
rate and the temperature difference between a reservoir and the injection condition, allow a substantial heat rejection.

2.6 Conclusions

The numerical simulation of the transpiration cooling technique and the investigation of the effects of air blowing into the hypersonic laminar boundary layers on a flat plate have been presented in this section. In particular, a reduced order model has been developed and implemented in a computationally inexpensive code named AERO-Code that solves the Navier-Stokes equations written for stationary, non-reacting, hypersonic laminar boundary layers and neglecting the radiative thermal exchange. Additionally, the new concept of the variable transpiration cooling has been introduced for a preliminary design of the TPS in terms of optimum distribution of the cooling power along the flat plate. The numerical results have been, thus, validated by using the CFD simulations with GASP performed by the research group at the Missouri University of Science and Technology. The main conclusions that can be drawn are listed below.

- The numerical solution of the reduced order model, that is based upon the coupled use of Self-Similar Method and Difference Differential Method, allows implementing any type of transpiration strategy without binding the choice of the wall velocity to a particular distribution like in the reduced order models developed in the past (i.e. $V_W \propto \sqrt{x}$).

- The analysis in §2.5.1 showed that the constant-linear wall velocity distribution, with respect to the other selected distributions, minimizes the wall heat flux along the entire flat plate length if the coolant mass-flow rate is kept constant (Fig. 2.11 and Table 2.3).
• In §2.5.2, the investigation of the selected distributions with respect to the total mass-flow rate blown from the porous wall has been considered. The sawtooth wall velocity distribution allows reducing by about 37% the coolant mass with respect to the other transpiration strategies analyzed (Table 2.4). This result highlights the potential thermal-management implications of this concept applied to hypersonic vehicles.

• The comparisons with the results obtained by using the high-fidelity code GASP show that AERO-Code is capable of predicting the flow features with a maximum deviation of about 24% (detected at the location where the transpiration starts) on both wall heat flux and skin friction. The deviation between the two codes in terms of wall heat flux prediction (mainly due to the viscous interaction and to the variation of the thermophysical properties of the flow inside the boundary layer) is decreased to about 8%, when a lower flight Mach number is considered.

• The capability of AERO-Code to perform a quick and complete analysis of the boundary layer flow makes it a useful tool for sensitivity and parametric studies to assess the most promising cooling strategies and, thus, to optimize the thermal protection system based on transpiration. The code is also an ideal tool for an integrated aerodynamic-material study and multidisciplinary design optimization (MDO) applications for high-speed aerospace vehicle design.
CHAPTER 3

Integrated Analysis of the Transpiration Cooling Technique


The opening of this chapter (§3.1) is dedicated to the mathematical models and numerical solutions used to couple the boundary layer flow to the thermal response of selected porous materials. In §3.2, the input parameters and the results of the integrated analysis for a flat plate (§3.2.1) and a blunt body with circular leading edge (§3.2.2) are reported. The matching procedure used to couple the boundary layer solver with the material solver is briefly described in §3.3 while the parametric analysis with respect to the thermophysical properties of the porous material is shown in §3.4.

3.1 Mathematical Models and Numerical Solutions

This section presents the mathematical models and numerical solutions used for the integrated analysis of transpiration cooling. Two characteristic geometries of interest for hypersonic applications are analyzed: a flat plate and a two-dimensional body with blunt leading edge (Fig. 3.1-a and Fig. 3.1-b, respectively). The simu-
lations of the hypersonic boundary layer on the flat plate have been performed by using two different codes: the high-fidelity LAURA code [27] and the reduced-order model developed in Chap. 2 (AERO-Code) [2]. The simulations of the BL for the blunt body of Fig. 3.1-b have been performed with the LAURA code only. All the numerical simulations by using LAURA have been performed by the research group of the Missouri University of Science and Technology [3]. The uniform and variable transpiration cooling have been accounted for, in both codes, as a boundary condition on the transversal velocity at the wall. The material’s thermal response for both geometries has been analyzed by using a one-dimensional (1-D) model that solves the fluid-flow and heat-transfer equations inside the porous medium (MAT-Code).

3.1.1 Mathematical Model and Numerical Solution of the Boundary-Layer Flow

The mathematical modeling of the BL flow by using AERO-Code has been described in detail in §2.2 while the numerical simulations have been performed for
a mesh grid with 500 points along the $y$-axis (normal-to-wall) and 150 points along the $x$-axis (streamwise direction). The invariability of the results with respect to the mesh grid has also been verified. The LAURA code [27], from the NASA Langley Research Center, has been used for the high-fidelity numerical solutions of the 2-D flowfield, including the shock-layer and the boundary-layer regions, over the flat-plate and blunt-body geometries. LAURA is a threedimensional, structured, Navier-Stokes code, which uses a finite-volume shock-capturing approach to solve high-speed flows with frozen, equilibrium, or non-equilibrium thermochemistry [3]. For the applications in this work, the thin-layer form of the Navier-Stokes equations has been solved. The cross-flow and the transpiration gases have been specified as pure nitrogen. The temperature and pressure values encountered during the simulations were below the onset temperature and pressure values of the dissociation of the $N_2$ gas, and so no chemical reactions were considered. The CFD solutions of the flat-plate cases have been obtained at three structured grid levels (coarse, medium, and fine) to check the grid convergence. For all of the cases, the results (pressure, wall heat flux, skin-friction, etc.) of the medium and fine grid levels were almost identical, indicating grid convergence at the medium grid level. The results presented in this paper have been obtained with the fine grid level, which had 101 (streamwise) x 181 (normal-to-wall) grid points for the flat plate and 201 (streamwise) x 129 (normal-to-wall) grid points for the blunt body. Figure 3.2 shows the fine grid level used in the blunt-body simulations [3].

3.1.2 Mathematical Model and Numerical Solution of the Fluid Flow and Heat Transfer Through the Porous Material

In this section, the analysis of the porous material is described to relate the properties of the flow to those of the coolant fluid issued from the porous wall. The
characterization of the material response underneath the boundary layer is essential to understand the influence of the blowing parameters of the coolant fluid on the resulting boundary-layer profile. Figure 3.3 shows the schematic of the representative porous media with the relevant quantities involved in the analysis considered here.

With reference to Fig. 3.3, the following assumptions are used for the thermal modeling of the porous material:

- 1-D Analysis
- $\partial/\partial t = 0$ \hspace{1cm} *(steady-state condition)*
- \( T_{m,i} \cong T_{CF,i} \) (equal matrix, \( T_m \), and coolant temperature, \( T_{CF} \), at the cold wall)
- \( q_R = 0 \) (negligible radiative heat flux)
- \( k_{m,x} \neq k_{m,y} \) (thermal conductivity of the material constant with respect to the temperature)
- \( \epsilon = \text{constant} \) (isotropic porosity)
- \( B_0 = \text{constant} \) (isotropic permeability)
- \( N_2 \) as coolant fluid

Nitrogen has been considered as coolant fluid to neglect any chemical interaction with the freestream flow. The temperatures of the material, as well as the coolant’s temperature at the cold wall, have been assumed similar within a tolerance of 10 K for the steady-state condition. The basis for introducing the tolerance chosen is clarified in the discussion of the results for the flat-plate geometry (§3.2.1). The thermal conductivity has been assumed constant with respect to the temperature because carbon-based materials have been selected for this analysis [3]. The heat exchange inside the porous material has been modeled using the differential method. In this work, the differential method has been preferred over the integral method because, as already mentioned in Chap. 2, a detailed characterization of the thermodynamic parameters into the material is necessary [1, 16]. The coolant’s pressure and temperature \( (P_{CF} \text{ and } T_{CF}, \text{respectively}) \), the temperature of the material \( (T_m) \) and the convective heat transfer \( (h_V) \) represent the unknowns of the problem. The material is modeled here as homogenous. Porous media can be regarded as such when the distribution and size of the pores is statistically uniform across the entire bulk volume of the material [28]. This assumption allows for modeling of the convective heat transfer at the interface solid-fluid by using the well investigated heat exchange properties of a bed of packed spheres [28] (Fig. 3.4).
In both Fig. 3.3 and Fig. 3.4, the reference system has been placed on the hot wall to simplify the initialization of the numerical method. 1-D modeling has been approached, for both blunted body and flat plate, because the longitudinal heat transfer ($x$-direction) has been considered negligible with respect to the heat transfer across the material’s thickness ($y$-direction). This assumption is valid when the material is homogenous, and orthotropic thermal properties of the porous matrix are considered. If the material is homogenous, then the coolant fluid has no preferential direction of flow inside the pores network until a pressure gradient is applied in one or more directions. In this analysis, the transpiration is achieved by applying a pressure gradient across the material’s thickness, and thus, once the pores are completely filled (steady-state condition), the coolant will have a preferred direction of flow (fluid passages along $dy$ in Fig. 3.4). The net mass flux of coolant fluid entering and exiting transversally to the materials thickness (fluid passages along $dx$ in Fig. 3.4) is zero, due to the material’s homogeneity. In this way, the same coolant mass injected at the cold wall will be issued at the same $x$-location of the hot wall, and it will be possible to consider the coolant mass-flow rate constant across the material’s thickness. Furthermore, if the longitudinal thermal conductivity of both coolant and material, $k_m(x)$ and $k_{CF}(x)$, are lower with respect to the transversal thermal conductivity of the material, $k_m(y)$, then the conductive heat transfer along
the $x$-direction can be neglected. These conditions can be matched for laminate carbon composites when the fibers are aligned along the material’s thickness and when thermally insulator’s coolant fluids are considered (e.g., nitrogen and air). The equations that describe the heat exchange in both material and coolant fluid are extracted by using the local averaging technique applied to a control volume including both coolant and solid material (reference elementary volume) [28]. The system of equations to solve (Eq. 3.1) is based on a previous model presented by Glass et al. [6] and Bucchi et al. [7]. The heat transfer in the matrix (first equation in Eq. 3.1) is mainly governed by conduction (Fourier’s law) into the porous material and convection between the coolant fluid and the matrix. The equation describing the heat transfer in the coolant fluid derives from the heat balance between the convection and the variation of the coolant’s enthalpy, when the conduction in the coolant fluid is neglected (second equation in Eq. 3.1) [7]. Darcy’s law is implemented into the model to calculate the pressure losses of the coolant fluid through the pores (third equation in Eq. 3.1) [6, 7].

\[
\begin{align*}
\frac{d^2 T_m}{dy^2} - \frac{h_v(T_m - T_{CF})}{k_m} &= 0 \\
\frac{dT_{CF}}{dy} - \frac{h_v(T_m - T_{CF})}{G_{CF}C_{P,CF}} &= 0 \\
\frac{dP_{CF}}{dy} &= -\frac{\mu_{CF}G_{CF}}{\rho_{CF}B_0}
\end{align*}
\]  

(3.1)

Where $C_{P,CF}$ and $\rho_{CF}$, are the specific heat at constant pressure and the density of the coolant fluid, respectively. Both the viscosity, $\mu_{CF}$, and thermal conductivity of the coolant fluid are calculated by using Sutherland’s law [17, 18]. The convective heat transfer coefficient has been calculated by relating the thermophysical properties of the porous material (porosity and permeability) with those of the coolant fluid (thermal conductivity and viscosity) [6, 29]. The permeability coefficient is modeled
here by using the Brennan-Kroliczek relationship [6, 7] between porosity and permeability. It is valid when the porous material is modeled as a series of solid spheres of radius $r_P$ (Fig. 3.4). The boundary conditions in Eq. 3.2 are used to couple the solution of the boundary layer with the thermal response of the porous material (MAT-Code). The boundary conditions at the wall ($q_W, V_W, T_W$) and the blowing parameters ($T_{m,i}, P_{CF,i}, T_{CF,i}$) are used, respectively, for the hot wall ($y = 0$) and for the cold wall ($y = H$, where $H$ is the material’s thickness):

$$
\begin{align*}
T_{m}(H) &= T_{m,i} \\
T_{CF}(H) &= T_{CF,i} \\
\frac{T_W - T_{m}(0)}{\Delta y} &= -\frac{q_W}{k_m} \\
P_{CF}(H) &= P_{CF,i}
\end{align*}
$$

(3.2)

Where $q_W$ is the wall heat flux while the $i$ subscript represents the initial conditions at the cold wall before the iterative procedure to match the boundary conditions (shooting technique) is initiated (§3.2.1). The local coolant mass-flow rate per unit area or mass flux, $G_{CF}$, is calculated as (Eq. 3.3) [18].

$$
G_{CF} = \rho_{CF}V_W
$$

(3.3)

The reduced-order model of Eq. 3.1 has been implemented by using finite differences [1]. The material code calculates the coolant and matrix temperatures throughout the thickness of the material and along the body’s surface once the required wall temperature $T_W(x)$ and the wall heat fluxes $q_W(x)$ are prescribed. An iterative process extracts the blowing conditions at the cold wall of the porous material maintaining the wall temperature in the required range. A series of nested cycles take into account for the operational limits of the coolant fluid (if the coolant
Figure 3.5. Schematic of an annular sector of the leading edge.

is liquid, no boiling and/or freezing occur inside the matrix) and ensure that the transpiration phenomenon through the material’s hot wall takes place while preventing the injection phenomenon (i.e. minimum pressure differential at the pores’ exit). The solution of the same equations (Eq. 3.1) for the blunted body is possible with some limitations. In fact, the formulation of the model in Cartesian coordinates does not account for the variation of area due to the material’s thickness and to the radius of curvature of the leading edge. The change of area, which in a 2-D model is a length, is calculated by a simple geometric analysis referred to Fig. 3.5 (Eq. 3.4).

\[
\frac{A}{A'} \approx 1 - \frac{H \vartheta}{2\vartheta R_N} \quad (3.4)
\]

The closer the area ratio is to 1, the more the Cartesian 1-D model is applicable to curved shapes. \(A/A'_0 = 0.8\) for the blunted body analyzed in §3.2.2. The value of \(A/A'_0 = 0.75\), which means \(H/R_N = 0.5\), is considered as the lower limit of the area ratio for which the Cartesian coordinates cannot be applied to curved geometries.

In the following sections, the integrated analysis of the boundary layer coupled with the thermal response of the porous material is performed for both the flat plate (§3.2.1) and the blunt body (§3.2.2) configurations.
3.2 Integrated Analysis of Transpiration Cooling

Two different transpiration strategies are considered for both configurations of Fig. 3.1: a uniform transpiration and a variable transpiration along the body’s surface. A constant value of the transversal wall velocity is imposed for the strategy with uniform transpiration (Fig. 3.6-a and Fig. 3.7-a). The variable transpiration is obtained by imposing a sawtooth distribution of the transversal wall velocity (Fig. 3.6-b and Fig. 3.7-b).

The sawtooth distribution has been selected because, as reported in the results presented in Chap. 2, of its capability to save coolant fluid with respect to other transpiration strategies when laminar flow over a flat plate is considered. Two different nominal values of the wall velocity distribution are chosen for both transpiration strategies and both geometries (Fig. 3.6 and Fig. 3.7). It is because the total amount of coolant issued into the boundary layer (Eq. 3.5) has been constrained to be constant in order to compare the effectiveness of the transpiration in terms of wall heat flux reduction.
Figure 3.7. Wall velocity distribution used for the simulations of the blunted body. a) Constant wall velocity; b) Sawtooth wall velocity.

\[ G_{CF, tot} = \int_0^L \rho_W V_W(x) dx \]  

(3.5)

Where \( V_W \) is the transversal wall velocity. The total flow rate of coolant injected per unit width is calculated by integrating along the body’s length, \( L \), the mass flux with a quadrature numerical formula (Table 3.1):

<table>
<thead>
<tr>
<th></th>
<th>( G_{CF(\text{constant } V_W)}, \text{ kg/s} \cdot \text{m} )</th>
<th>( G_{CF(\text{sawtooth } V_W)}, \text{ kg/s} \cdot \text{m} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flat plate</td>
<td>0.0039</td>
<td>0.0039</td>
</tr>
<tr>
<td>2-D Blunt body</td>
<td>0.0037</td>
<td>0.0038</td>
</tr>
</tbody>
</table>

3.2.1 Flat Plate Geometry

The simulations of the flat plate have been performed by using AERO-Code [2] and the LAURA code [27]. The wall velocity, for both transpiration strategies, has been activated at \( x = 0.01 \) m. The following input parameters have been used:
- $M_e = 8$ *Freestream Mach number*
- $h = 35000 \, m$ *Flight altitude*
- $T_e = 237 \, K$ *Freestream temperature*
- $T_{W} = 1500 \, K$ *Wall temperature*
- $L = 0.3 \, m$ *Flat plate length*
- $q_R = 0$ *Negligible radiative heat transfer*
- $N_2 \, flow$ *Freestream chemical composition*
- $Re_x = 1.3 \cdot 10^6 \, 1/m$ *$Re_x = 3.8 \cdot 10^5$ at the end of the flat plate*

The length of the flat plate has been imposed in order to comply with the laminar flow assumption. The wall temperature selected maintains almost the same temperature ratio used in the analyses in Chap. 2 ($T_{W}/T_t = 0.5$) [2]. The higher wall temperature with respect to the external temperature does not imply that the plate is being heated. The adiabatic wall temperature corresponding to the flight conditions considered in this analysis is in the order of $T_t \approx 3000 \, K$. The uniform value of the wall temperature represents only a base case used for this analysis. In fact, in relation to the material’s technology, the highest value of the wall temperature is desired to minimize the blowing of coolant and to reduce the thermal gradient across the material’s thickness. The results obtained from AERO-Code are summarized in Fig. 3.8.

The activation of the blowing increases the BL thickness (Fig. 3.8-a). The growth of both the cinematic and thermal boundary layer thickness generates a decrease, respectively, of the velocity and temperature gradient at the wall, which correspond to a diminishing of the wall shear stresses and wall heat fluxes (Fig. 3.8-b). The uniform transpiration increases the BL thickness by about 15% and allows a decreasing by 45.7% of the wall heat flux at $x = 0.1 \, m$. The variable transpiration reduces the wall heat flux by 49.4% at the same location. The oscillations on the BL
thickness plot (Fig. 3.8-a) depend on the number of points chosen for the grid. The same base case has been simulated with the LAURA code [27]. The comparison of the wall heat flux calculated with the LAURA code and AERO-Code highlights the differences between the two solvers (Fig. 3.8-b and Fig. 3.9).

The difference is due to the strong viscous interaction at the leading edge of the flat plate that AERO-code cannot take into account. The viscous interaction occurs due to the mutual interdependence between the non-zero displacement thickness at the leading edge of the flat plate and the shock generated at the same location (Fig. 3.10-a).
Figure 3.10. Boundary-layer simulations performed with the LAURA code. a) Contour plot of pressure inside the shock layer; b) Pressure profiles at the wall.

The thicker BL calculated with the LAURA code decreases the wall heat flux with respect to the prediction of AERO-Code. The deviation of the heat flux is about 23% for the no-transpiration case and about 35% for both uniform transpiration and variable transpiration. The variation of the pressure at the wall, $P_W$, or external pressure, $P_e$, in Fig. 3.10-b points out the influence of the hypersonic viscous effects in transpiration cooling. In fact, the variation of the blowing velocity affects directly the inclination of the shock, which, in turn, modifies the thermodynamic properties of the flow. The wall heat flux $q_W(x)$, the wall temperature $T_W$, the pressure inside the BL $P_W(x)$, and the wall velocity $V_W(x)$ calculated by using the LAURA code are implemented as input parameters to perform the thermal analysis of the porous material. The base case is referred to the thermophysical properties of carbon-carbon (C-C) foams for aerospace applications [30]:

- $k_m = 0.4 \text{ W/m} \cdot \text{K}$  \textit{Material thermal conductivity}
- $H = 0.1'' = 0.0254 \text{ m}$  \textit{Material thickness}
- $\epsilon = 0.2 \text{ K}$  \textit{Porosity}
- $B_0 = 2.37 \cdot 10^{-12} \text{ K}$  \textit{Permeability}
The permeability has been calculated, as already mentioned, by using the Brennan-Kroliczek relation [6, 7, 29], which relates the permeability coefficient to the dimension of the pores and to the porosity. Noteworthy is the choice of $k_m$. The values of the thermal conductivity range from $0.035 \, \text{W/m} \cdot \text{K}$ to $180 \, \text{W/m} \cdot \text{K}$ [30]. These values depend on the stacking sequence of the composite material, if composite laminates are considered, or on the average orientation of the carbon/graphite ligaments, if sintered materials are considered. In this analysis, a low value of the transversal thermal conductivity $k_{m,y}$ has been considered. It means that the longitudinal thermal conductivity $k_{m,x}$ has to be almost an order of magnitude lower in order to respect the hypothesis about negligible heat transfer along the x direction ($k_{m,y} = 0.4 \, \text{W/m} \cdot \text{K}; \, k_{m,x} = 0.04 \, \text{W/m} \cdot \text{K}$) [30]. The results of the thermal analysis are reported for the locations represented in Fig. 3.11.

The results of the thermal analysis for both transpiration strategies are reported in Fig. 3.12 to Fig. 3.15.

- $N_2$ as coolant fluid

Figure 3.11. Chosen locations to plot the MAT-Code results. a) Flat plate profile; b) Sawtooth wall velocity.
Figure 3.12 shows that, if the thermophysical properties of the porous material ($k_m, \epsilon$) are kept constant, it is necessary to change the blowing conditions of the coolant fluid at the cold wall in order to maintain a constant wall temperature and the transpiration velocity of Fig. 3.6. In general, if the wall temperature is maintained constant, both the coolant and material temperatures have to increase along the flat-plate length to compensate the decrease of the heat flux. The temperature of the coolant fluid changes with the transpiration strategy based on its dependence on the local mass-flow rate issued from the porous wall. In fact, at $P_1$, $P_2$, and $P_3$, the blowing temperature for the variable transpiration (Fig. 3.12-b) is lower with respect to the constant $V_W$ case (Fig. 3.12-a) due to the higher local coolant flow rate in the range $0.01 \leq x \leq 0.15$. The temperature distribution inside the material (Fig. 3.13) is linear because only the heat transfer by conduction has been considered (Eq. 3.1). It is possible to verify the accuracy of the BC’s applied to the hot wall ($T_m = T_W; P_{CF} = P_e$) and to the cold wall ($T_{m,i} = \epsilon_{tol} \cdot T_{CF,i}$) from Fig. 3.12 to Fig. 3.14. $\epsilon_{tol}$ is the numerical tolerance used in the technique to verify that the coolant temperature and matrix temperature are similar at the cold wall. The shooting
Figure 3.13. Temperature of the material across the material’s thickness. a) Uniform transpiration; b) Variable transpiration.

The technique is used to guess the coolant temperature at the hot wall by obeying the relation $T_{m,i} = \epsilon_{T_{ol}} \cdot T_{CF,i}$ at the cold wall. In this analysis $\epsilon_{T_{ol}} = 10 \text{ K}$ has been imposed. This represents a good compromise between the computational time and the invariability of the results. The pressure drop across the material’s thickness (Fig. 3.15) is a critical parameter to be checked for both refractory porous materials manufactured by a sintering process and laminated composite materials. In sintered materials, a high-pressure difference can generate the cracking of the pores with the instantaneous propagation of the crack across the material’s thickness. A different failure mechanism can be detected for those composite materials manufactured by lamination. In this case, a high-pressure gradient across the material’s thickness generates high interlaminar shear stress, which can determine the delamination of two or more layers of material. A reference value for the maximum pressure drop, in the case of thin wall structures, can be considered around $\Delta P_{CF} \approx 50 \text{ psi}$. The pressure drop across the material is below the reference limit for both transpiration strategies analyzed here (Fig. 3.15) but the nominal values increase consistently if thicker materials are considered [3]. Important emphasis has to be placed on the
blowing conditions of the coolant fluid. At $P_1$, the nominal values of the coolant’s
temperature at the hot wall are lower than the imposed temperature of the wall
($T_W$). The apparent discrepancy of this result is related to the fact that it represents
the first step of an iterative process that has to be established to reach convergence
of the material-fluid coupling system (§3.3).

3.2.2 Two-Dimensional Body with Blunt Leading Edge

The simulations of the BL flow have been performed with the LAURA code
by imposing the same input parameters used for the flat-plate case (§3.2.1). The
transpiration is activated at the stagnation point and the wall velocity distributions
of Fig. 3.7 are implemented. The same input parameters used for the flat-plate
simulations are imposed:

- $M_e = 8$ \textit{Freestream Mach number}
- $h = 35000$ m \textit{Flight altitude}
- $T_e = 237$ K \textit{Freestream temperature}
- $T_W = 1500$ K \textit{Wall temperature}
Figure 3.15. Pressure drop along the flat plate. a) Uniform transpiration; b) Variable transpiration.

Figure 3.16. LAURA code results [3]. a) Pressure at the wall; b) Wall heat flux.

- $L = 0.3 \text{ m}$ \textit{Flat plate length}
- $q_R = 0$ \textit{Negligible radiative heat transfer}
- $N_2$ as freestream flow
- $Re_x = 1.3 \cdot 10^6 \text{ 1/m}$ $Re_x = 0.5 \cdot 10^5 \text{ at the end of the body}$

The pressure at the wall and the heat-flux distributions obtained from the LAURA code simulations are reported in Fig. 3.16.

Figure 3.16-a shows that, contrary to the flat-plate geometry, the wall pressure is not remarkably influenced by the transpiration. In this case, the transpiration
velocity does not affect considerably the location of the shock and, thus, the flowfield outside the boundary layer. The cooling effectiveness of the transpiration strategies can be compared analyzing the stagnation point (SP) region (Fig. 3.16-b). For $x = 0$ m, the uniform transpiration generates a decrease of 48% of the wall heat flux with respect to the uncooled case. The variable transpiration with sawtooth velocity distribution generates a decrease of 56% of the wall heat flux at the same point. The advantages of using the variable transpiration are clear, considering that the total amount of coolant injected into the BL is the same for both transpiration strategies (Table 3.1). Figure 3.17 shows the BL manipulation along the entire surface of the body when the variable transpiration is adopted. When the transpiration is activated, the BL thickness increases a few tenths of a millimeter. The output data returned by the LAURA code (the wall heat flux, the wall velocity along the body’s surface, and the pressure inside the boundary layer) are used as initializing parameters for the thermal analysis of the porous material.
The integrated analyses have been performed by using 150 grid points across the material’s thickness and 136 points along the leading-edge surface. The thermophysical properties of the porous material are selected considering C-C based materials [30]. The input parameters for the base case are the same used for the integrated analysis of the flat plate. The results are reported for the locations represented in Fig. 3.18.

The same features encountered for the integrated analysis of the flat plate are recognizable from Fig. 3.19 and Fig. 3.20. If the wall temperature is kept constant, both the coolant and material temperatures have to increase along the flat-plate length to compensate for the decrease of the heat flux.

The thermal response of the stagnation point is not reported in this section because it violates the constraint regarding the use of Cartesian coordinates on curved surfaces (Eq. 3.4). The coolant pressure for the variable-transpiration strategy is higher with respect to the constant $V_W$ distribution (Fig. 3.20) due to the higher blowing velocity of the sawtooth distribution at the stagnation point: $V_{W_{\text{Sawtooth}}} = 6 \text{ m/s}; V_{W_{\text{Constant}}} = 5 \text{ m/s}$ (Fig. 3.7). The pressure drop across the material’s
Figure 3.19. Temperature of the coolant across the material’s thickness. a) Uniform transpiration; b) Variable transpiration.

thickness (Fig. 3.21) increases rapidly in the curved surface of the blunted body and then becomes almost constant on the sidewall of the body.

On the curved surface, the pressure drop increases faster for the variable transpiration. This occurs due to the higher transversal velocity at the same locations \( 5.2 \, \text{m/s} \leq V_W \leq 6 \, \text{m/s} \) for \( 0 \, \text{m} \leq x \leq 0.01 \, \text{m} \). The peaks of pressure drop for the variable transpiration cooling at \( x = 0.011 \, \text{m} \) and \( x = 0.024 \, \text{m} \) (Fig. 3.21-b) are due to the rapid increase of the local wall velocity. The same considerations made for the flat-plate analysis are valid here. In fact, downstream \( P_1 \), it is not necessary to keep the wall temperature at 1500 K, because, with respect to the stagnation point heat flux, the local wall heat flux drops about 80% downstream \( s \approx 0.005 \, \text{m} \) (corner region of Fig. 3.18). Downstream \( P_1 \), it is possible to let the wall reach the equilibrium temperature that, in turn, allows saving a consistent amount of coolant. Furthermore, if the temperature of the coolant at the cold wall is too high in the region between \( SP \) and \( P_1 \), it is possible to increase the material’s thickness or decrease the thermal conductivity of the porous material. In this scenario, the use of
Reduced-order models is extremely valuable in selectively studying the influence of the input parameters with the aim to define the cooling strategy.

### 3.3 Matching Procedure of the Integrated Analysis

The analysis of the material thermal response for the flat-plate geometry provides lower blowing temperatures of the coolant at the hot wall with respect to those imposed in the LAURA code ($T_{CF-HW} = T_W$), which creates a deviation on the injection velocity values between the LAURA code and MAT-Code. This is because the local coolant flow rate per unit area ($G_{CF}$) has been kept constant, and therefore if the specific thermophysical and geometrical properties of the porous material are prescribed ($\epsilon = 0.2$, $k_m = 0.4 W/m \cdot K$, and $H = 2.5 \text{ mm}$), an iterative process is needed to match the wall velocity distribution (Eq. 3.3). For instance, at $P1$, the BL solution has to be recalculated by using the new temperature of the coolant fluid at the hot wall and the updated transversal wall velocity of Tables 3.2 and Table 3.3.

The percentage deviations of the wall velocity downstream $P1$ decrease for both transpiration strategies because the blowing temperatures of the coolant at the
Figure 3.21. Pressure drop along the blunted body surface. a) Uniform transpiration; b) Variable transpiration.

Table 3.2. Deviation of the blowing conditions for uniform transpiration at $P_1$

<table>
<thead>
<tr>
<th>Location</th>
<th>$T_{CF-HW}$, K</th>
<th>$P_{CF-HW}$, Pa</th>
<th>$\rho_{CF-HW}$, kg/m$^3$</th>
<th>$G_{CF}$, kg/s·m</th>
<th>$V_W$, m/s</th>
<th>$\Delta V_W$, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>LAURA</td>
<td>1500</td>
<td>559</td>
<td>0.00126</td>
<td>0.00126</td>
<td>10.00</td>
<td></td>
</tr>
<tr>
<td>MAT-Code</td>
<td>935</td>
<td>559</td>
<td>0.00201</td>
<td>0.00126</td>
<td>6.03</td>
<td>40.0</td>
</tr>
</tbody>
</table>

hot wall get closer to the temperature of the material at the same location. The updated wall heat flux, calculated by using LAURA, is implemented again in MAT-Code to define the new blowing conditions. The iterative process is stopped when the differences between $T_{CF-HW}$ and, thus, $V_W$ of two consecutive simulations are lower than a prescribed tolerance. In this scenario, a reduced-order model to calculate the BL solution is beneficial to reduce the computational time. If the thermophysical and geometric properties of the porous material are not constrained to specific values and

Table 3.3. Deviation of the blowing conditions for variable transpiration at $P_1$

<table>
<thead>
<tr>
<th>Location</th>
<th>$T_{CF-HW}$, K</th>
<th>$P_{CF-HW}$, Pa</th>
<th>$\rho_{CF-HW}$, kg/m$^3$</th>
<th>$G_{CF}$, kg/s·m</th>
<th>$V_W$, m/s</th>
<th>$\Delta V_W$, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>LAURA</td>
<td>1500</td>
<td>559</td>
<td>0.00126</td>
<td>0.00145</td>
<td>11.50</td>
<td></td>
</tr>
<tr>
<td>MAT-Code</td>
<td>900</td>
<td>559</td>
<td>0.00209</td>
<td>0.00145</td>
<td>6.93</td>
<td>39.7</td>
</tr>
</tbody>
</table>

57
are considered one of the problem’s variables, an effective matching procedure can be implemented with the scope of keeping the same $G_{CF}$ and $V_{w}$ initially prescribed (this is particularly important when a selected, effective, wall velocity distribution has been identified, as in the described case of the sawtooth distribution [2]). Only the flat plate is considered because, for the 2-D blunt body, the sole variation of the porosity and thermal conductivity of the material, within allowed range, are not sufficient to obtain convergence. Additionally, the increase of the material’s thickness in the stagnation point region violates the constraint regarding the use of Cartesian coordinates on curved surfaces (Eq. 3.4). The following input parameters for the thermal analysis of the porous material have been imposed according to the thermal properties of carbon based materials [30]:

- $k_{m} = 0.8 \text{ W/m} \cdot \text{K}$  \textit{Material thermal conductivity}
- $H = 0.4'' = 0.0102 \text{ m}$  \textit{Material thickness}
- $\epsilon = 0.1 \text{ K}$  \textit{Porosity}
- $B_{0} = 2.37 \cdot 10^{-12} \text{ K}$  \textit{Permeability}
- $N_{2}$ as coolant fluid

Figure 3.22 shows that, at $P_{1}$, the temperatures of the coolant fluid at the hot wall, for both wall velocity distributions, are about 40% higher with respect to the first iteration of the integrated analysis (Table 3.2 and Table 3.3). It allows reducing the deviation on the transversal wall velocity by 12% for both uniform and variable-transpiration strategies.

Figure 3.23 highlights the substantial effect of increasing the thickness and reducing the porosity of the material with respect to the nominal input conditions used in §3.2. In fact, for both blowing strategies, the pressure drop across the material’s thickness almost triplicates, consequently generating higher mechanical
loads on the structure. In this scenario, it is fundamental to investigate the effects of the individual thermophysical properties of porous media (§3.4).

3.4 Parametric Analysis with Respect to the Thermophysical Properties of the Material

For explicative purposes, only the stagnation point of the blunted body is considered in this analysis once the procedure to match the blowing condition at the hot wall is clarified. This is because it is the most critical region where higher thermal loads are encountered. The variable transpiration with sawtooth distribution is uniquely considered in this section due to its higher cooling effectiveness with respect to the uniform transpiration. The input parameters of the base case are the same used in §3.2, and the results are represented next with solid lines. The porosity/permeability (§3.4.1) and thermal conductivity (§3.4.2) effects have been studied via a parametric analysis because they represent the driven parameters defining the thermomechanical response of different porous materials of interest for hypersonic...
3.4.1 Parametric Analysis with Respect to the Porosity

Three different values of the porosity are considered in this analysis. The lower value ($\epsilon = 0.1$) is imposed, taking into account the average porosity of laminated carbon composites materials. The higher value of the porosity considered here ($\epsilon = 0.4$) is based on sintered porous materials. Porosity values over $\epsilon = 0.4$ (typical of carbon foams) are not considered because of the low structural properties. The reduction of the porosity, with respect to the base case analyzed in §3.2, has a small effect on the coolant temperature at the cold wall while generating a consistent increase of the coolant’s temperature at the hot wall (Fig. 3.24-a). It is due to the fact that the smaller the porosity is, the smaller the permeability is. In this scenario, the coolant encounters more mechanical resistance while being issued and has more time to exchange heat with the hotter surrounding matrix. The permeability coefficient, calculated by the Brennan-Kroliczek relation [6, 7, 29], for the three cases is:
Figure 3.24. Blowing strategies at the stagnation point for different values of the material’s porosity. a) Coolant temperature; b) Pressure drop.

\begin{align*}
  - B_{01} & = 8.09 \cdot 10^{-12} \text{ m}^2 \cong 10 D \quad \text{for } \epsilon = 0.4 \\
  - B_{02} & = 1.07 \cdot 10^{-12} \text{ m}^2 \cong 1 D \quad \text{for } \epsilon = 0.2 \\
  - B_{03} & = 1.65 \cdot 10^{-13} \text{ m}^2 \cong 0.1 D \quad \text{for } \epsilon = 0.1
\end{align*}

Where the nominal value of $B_{01}$ is representative of a high-semipervious material while $B_{02}$ and $B_{03}$ resemble the permeability values typical of semipervious and low-semipervious materials, respectively. The maximum pressure drop remains below the reference limit for carbon-based materials because the external pressure is low enough to favor the issuing of coolant (Fig. 3.24-b). The peaks of pressure drop at $x = 0.006 \text{ m}$ are due to the strong variation of the external pressure in the region that connects the rounded nose with the straight sidewalls (expansion region).

3.4.2 Parametric Analysis with Respect to the Thermal Conductivity of the Porous Material

The range of the values considered has been selected based on data available on current porous materials for aerospace applications [30]. The thermal conductivity
of the matrix has a high influence on the blowing temperatures of the coolant (Fig. 3.25-a).

At the stagnation point, a difference of about 700 $K$ is detectable, passing from the base case (solid line) to the higher values of $k$ because the convective heat transfer between the porous material and the coolant is directly related to the thermal conductivity of the material itself (Eq. 3.1). The pressure drop increases as the thermal conductivity is increased (Fig. 3.25-b) because the higher coolant temperature generates higher values of the coolant viscosity and, in turn, higher values of the coolant pressure. The pressure drop for the higher value of $k$ (dash-dotted line) even decrease on the curved part of the blunted body. This occurs because of the combined effect of both external pressure and wall heat flux decrement along the $S$ coordinate.

3.5 Conclusions

The coupled aero-material analysis for the numerical simulation of the transpiration cooling technique has been investigated in this chapter by coupling the
hypersonic boundary layer with the thermal response of selected porous materials. The integrated analysis is essential for establishing particular combinations of the thermophysical properties of the porous material and transpiration strategies capable of guaranteeing the thermostructural integrity of the thermal protection system for prescribed flight conditions. A 1-D model of the fluid flow and heat transfer across porous media has been implemented in a MATLAB®-based script and integrated with AERO-Code solver in order to determine the blowing conditions of the coolant fluid at the material’s cold wall. Two geometries of interest for hypersonic applications have been analyzed: a flat plate and a 2-D body with a blunt leading edge. Two different transpiration strategies have been considered for the integrated analysis of both configurations. A constant value of the transversal wall velocity has been imposed to reproduce the uniform transpiration and a sawtooth wall velocity distribution, which potentially allow substantially reducing the amount of coolant to be used (Chap. 2), has been selected to simulate the variable transpiration. The nominal values of the velocity at the wall have been chosen to provide the same total amount of coolant mass-flow rate blown from the porous material. The most remarkable conclusions which can be drawn from the results of the integrated analysis are reported below.

- The analysis of the results obtained by using the high-fidelity CFD code LAURA confirmed the capability of the reduced order model developed in Chap. 2 (AERO-Code) to capture the flow physics of the laminar boundary-layer flow over flat plates.
- Although the LAURA code has been used to perform the boundary-layer simulations for both geometries, the reduced-order model AERO-Code enables the capability of performing quick parametric analyses for different flight condi-
tions with the purpose of defining the most promising injection strategy in terms of wall heat flux reduction and coolant mass saving.

- The use of the variable transpiration with sawtooth wall velocity distribution reduces the heat flux at the stagnation point by 56% and the wall heat flux on the flat plate by 35% at \( x = 0.1 \) \( m \) from the leading edge. Additionally, for the test case analyzed, the variable transpiration for the blunt body is 8% more efficient, in terms of wall heat reduction at the stagnation point, with respect to the uniform transpiration.

- The simulations of the laminar boundary layer for the blunted body show that the heat flux drops by 80% from the stagnation point to the sidewalls. It suggests applying the transpiration only in the stagnation region reducing the overall amount of coolant to carry on-board.

- The material’s thermal analysis for the flat plate shows the other beneficial effect that derives from the usage of the variable transpiration. In fact, the sawtooth wall velocity distribution generates a pressure drop across the materials thickness that is 6% lower with respect to the uniform transpiration. The limited decrement of the pressure drop becomes fundamental to guarantee the structural integrity when higher external pressures are considered (e.g., during the descendant phase of hypersonic cruise vehicles or inside a supersonic combustion ramjet engine).

- The parametric analyses with respect to the thermophysical properties of the porous media show the remarkable influence of the porosity on defining the pressure drop across the material’s thickness while the material’s thermal conductivity is identified as the main parameter that determines the heat transfer between coolant and the solid material, if thermal insulator coolant fluids are considered. In this scenario, the use of the proposed reduced-order models at
the initial stage of TPSs design enables quick sensitivity analyses with respect to various control variables (wall velocity distribution, porosity, thermal conductivity, and material thickness). It allows also defining customized materials able to comply with the thermostructural requirements of the thermal protection system (e.g., the localized decrease of material’s thickness, such as the local decrease of the porosity, can be considered to reproduce the peaks of the sawtooth wall velocity distribution simulated numerically).
CHAPTER 4

Characterization of Complex Porous Structures for Reusable Thermal Protection Systems

This chapter has been extracted from the journal papers: S. Gulli and L. Maddalena, "Characterization of Complex Porous Structures for Reusable Thermal Protection Systems: Effective Permeability Measurements", 2014, Journal of Spacecraft and Rockets [13] and S. Gulli, L. Maddalena, C. McKelvey, A. Brown, Y. Nikishkov and A. Makeev "Characterization of Complex Porous Structures for Reusable Thermal Protection Systems: Porosity Measurements", 2014, Journal of Spacecraft and Rockets [4]. Small paragraphs have been selected from the invited paper: S. Gulli, L. Maddalena, C. McKelvey, A. Brown, Y. Nikishkov and A. Makeev "Permeability Measurements of Complex Porous Structures for Reusable Thermal Protection Systems (Invited Paper)", 2014, presented at the 19th AIAA International Space Planes and Hypersonic Systems and Technologies Conference [31]. The introductory part of this chapter (§4.1) describes the strategical importance of being able characterizing the blowing properties of the TPS specimen in order to enable the complete aerothermal reconstruction of the experiments and to characterize the blowing profile prescribed at the design stage of TPS components based on transpiration cooling. Section §4.2 has been dedicated to the basic theory behind the permeability measurement and the main challenges related to the non-destructive characterization of full-scale components in terms of local effective permeability. In §4.3, the methodology and data analysis used to experimentally calculate the local effective permeability characteristic of the prototype specimen is described in detail while the main
results and conclusions are reported in §4.4 and §4.5, respectively. §4.3 has been subdivided in several paragraphs. Specifically, §4.3.1 is focused on the description of the carbon-carbon specimen manufactured by Carbon-Carbon Advanced Technologies, Inc. (C-CAT) and used for the experimental campaign on transpiration cooling (Chap. 6). In §4.3.2 and §4.3.3 is illustrated the technique used for measuring the surface blowing by using hot-film anemometry while the characterization of the internal porous structure of the C-C cone by using X-ray computed tomography (CT) is widely described in §4.3.4.

4.1 Introduction

Reusable thermal protection systems, as already mentioned in Chap. 1, are an enabling technology for using hypersonic vehicles as practical, long-range, and affordable transport system. Both the aerodynamic and the material performance of hypersonic vehicles are strongly related to the near-wall effects. In fact, the extreme thermal loads generated by the viscous dissipation across the hypersonic boundary layer are further increased if the temperature is high enough to cause the dissociation of oxygen. Specifically, the thermostructural integrity of the TPS can be drastically reduced when particular combinations of elevated temperatures coupled to low partial pressures of oxygen occur and the active oxidation of candidate TPS material such as carbon-carbon and silicon-carbide is initiated. In this harsh environment, the cooling of the exposed structure is a stringent requirement and the transpiration cooling through a porous medium is investigated in this research work because of its higher cooling effectiveness in terms of wall-temperature reduction, coolant saving and minimum disturbance of the external flow with respect to other active cooling techniques. However, the overall cooling effectiveness of the transpiration technique is strictly related to the local blowing capability of the TPS component and, thus, the
material characterization by using non-destructive evaluation (NDE) techniques is crucial to determine the effective properties of the TPS that can be drastically different at the full-scale level due to the geometry, the system integration (e.g. structural constraints) and the intrinsic defectology coming from the manufacturing process. The latter dependence is of greater influence for materials constituted by two or more components (e.g. composite materials, alloys etc.) [32, 33, 34, 35]. In this chapter, the concept of effective permeability, conceived as the local blowing capability of a complex porous structure with respect to a selected coolant fluid, is introduced. The methodology used in this work for the non-intrusive characterization of the local effective permeability of a conical carbon-carbon porous structure is also discussed in detail. Specifically, the coolant (air) mass flux blown from a conical porous surface is probed by using hot-film anemometry measurements. Additionally, a study based on the X-ray computed tomography scan of the specimen has been carried out with the purpose of defining the most important guidelines for the permeability tests which are the minimum area to be probed with a hot-film anemometer, and the correct distance of the mass-flux sensor from the wall. The X-ray CT has been preferred to other NDE techniques because of its higher spatial resolution with respect to thermal tomography and due to the absence of structure superposition and signal dispersion that affect, respectively, conventional X-ray radiography and ultrasonic techniques [32].

4.2 Basic Concepts for Permeability Measurements

The permeability, in the most general definition, can be regarded as the capability of a selected material of letting pass a determined fluid through its internal tortuous structure [36, 37, 38, 39]. It is usually also referred as hydraulic conductivity [40, 41, 42] or flow conductivity [40, 41, 42, 43]. The permeability, as well as the
porosity, is a geometrical property of porous media but, contrarily to the porosity, it depends on the internal networks morphology (e.g. pores size distribution, connectivity, porosity itself etc.). In fact, composite materials having similar porosity (e.g. low-porosity carbon foams and highly-porous C-C layered materials) can have totally different permeability values due to the connectivity of the voids (presence of closed pores, blind channels etc.), diversified tortuosity and distribution of the voids’ dimensions. Although several correlations between porosity and permeability have been developed and validated for unconsolidated media modeled as a series of packed spheres [38, 39, 44, 45, 46, 47], a lack of established correlations for consolidated media, and specifically for layered composite materials, is still present because of the difficulty to model the aforementioned lattices features [41, 45, 48, 49, 50]. The permeability parameter, for most of filtering applications, is necessary to predict the pressure drop across the high- and low-pressure sides of the specimen and, thus, to guarantee the structural integrity of the porous media. Contrarily, the permeability distribution for transpiration cooling of reusable TPS is also necessary for selecting the correct control pressure (high-pressure side of the specimen) needed to reproduce the blowing profile/mass-flux distribution simulated numerically, which is prescribed by the cooling requirement, once the low-pressure field is imposed by the external flow. Permeability is strictly related to the characteristic flowfield inside the porous network [43, 44] and different simplified relations can be used to calculate the characteristic resistance that the fluid encounters when flowing across the tortuous maze. The Darcy’s law in Eq. 4.1 is the most simplified relation that describes the permeability dependence and it is valid in the case of laminar, non-inertial flow (creeping or Stokes flow) [28, 41, 51, 52].
\[ \vec{U}_D = -\frac{\vec{K}}{\mu_f} \cdot \vec{\nabla} \vec{P} \] (4.1)

The average permeability tensor \( \langle \vec{K} \rangle \) [43, 44, 52, 53] can be calculated once the driving force \( \langle \vec{\nabla} \vec{P} \rangle \), the average velocity across the probed area (Darcy's velocity, \( \vec{U}_D \)), and the fluid viscosity \( \mu_f \) are known. The formal derivation of the Darcy's law is obtained from the averaging technique of the Navier-Stokes equations applied to a reference elementary volume (REV) being statistical in terms of porosity [28, 52, 53]. The averaging process on the entire domain is needed since it is quite complicated to solve the Navier-Stokes equations inside the fluid domain due to the complex internal geometry of porous medium and due to the difficulty of defining the boundary conditions [28, 52, 54]. The use of the averaging technique, coupled to the average theorem formulated by Slattery [55] and described in [56], leads to a modified steady-state momentum equation that describes the fluid flow through the porous medium (Eq. 4.2). Equation 4.2 can be simplified in base of the flow regime inside the porous lattice.

\[ \rho_f \langle \vec{u} \rangle \cdot \vec{\nabla} \langle \vec{u} \rangle = -\vec{\nabla} \langle P \rangle + \mu_f \nabla^2 \langle \vec{u} \rangle - \frac{\mu_f}{\langle \vec{K} \rangle} \epsilon_{3-D} \langle \vec{u} \rangle \] (4.2)

Where the angled brackets indicate the averaged quantities across the REV [48, 52]. In particular, \( \langle P \rangle \) and \( \epsilon_{3-D} \) are the average pressure and volumetric porosity, respectively. \( \langle \vec{u} \rangle \) is named seepage velocity and it is strictly related to the porosity of the external surface where the velocity measurements are performed (superficial porosity, \( \epsilon_{sup} \) in Eq. 4.3).

\[ \langle \vec{u} \rangle = \frac{\vec{U}_D}{\epsilon_{sup}} \] (4.3)
The discriminant parameter used to define the flow regime is the Reynolds number based on a characteristic dimension of the voids. The characteristic length scale used to define the Reynolds number for consolidated porous media depends on the conformation of the internal network. The average pores’ diameter is usually selected as characteristic dimension for metallic and composite foams [57, 58, 59]. The average fibers’ diameter, or the hydraulic diameter based on the radius and on the volume fraction of the fibers, is commonly used for fibrous composite materials [49, 53]. In this case study, the average channels’ diameter is used to define the Reynolds number because it is the smallest characteristic dimension and, thus, it has the higher impact on defining the pressure drop across the material’s thickness of the layered C-C specimen examined. Both the inertial term (left hand side in Eq. 4.2) and the viscous term (second term of the right hand side in Eq. 4.2) can be neglected [43, 44] when the flow is laminar ($Re_{ch} \leq 10$). In this scenario, the averaged momentum equation is reduced to the generalized Darcy’s law (Eq. 4.1) and the variation of the pressure gradient with respect to the variation of the Darcy’s velocity is linear (e.g. constant permeability). For $10 \leq Re_{ch} \leq 300$, the inertial term (Forchheimer correction factor), the viscous term (Brinkman correction factor) and the Darcy term have the same order of magnitude and, the complete averaged momentum equation (Eq. 4.2) has to be analyzed in order to capture the deviation from the aforementioned linear behavior (e.g. variable permeability) [44]. For $Re_{ch} > 300$ the flow is in the inertial range and only the Forchheimer and Brinkman correction factors can be considered to predict the pressure drop across a porous medium [44, 52]. Throughout the experiments performed in this study, the use of the Darcy’s law has been verified to be legitimate for all the flow rates used due to the small characteristic diameter of the channels ($D_{ch} \approx 30 \, \mu m$), calculated by using the CT-scan images [4], coupled to the use of gaseous coolant fluids that
lead to small Reynolds numbers. The average permeability of a porous material with respect to a selected fluid in laminar regime has been directly calculated from the slope of the straight-line connecting the pressure gradient to the coolant mass flux (gaseous coolant) or to the Darcy’s velocity (liquid coolant). The local permeability for the $r$-direction (across-the-thickness direction) can be calculated by projecting Eq. 4.1 in the same direction and by integrating it across the material’s thickness ($H$) (Eq. 4.4).

\[-\int_{P_{ext}}^{P_H} dP = \int_0^{-H} \frac{\mu_f U_r}{K_r} dr \quad (4.4)\]

Equation 4.4 assumes the empirical form of the Darcy’s law, once the average velocity and the dynamic viscosity of the fluid are assumed constant across the local thickness of the material (Eq. 4.5).

\[\frac{P_0 - P_H}{H} = \frac{\mu_f}{K_r} U_r^D \quad (4.5)\]

Equation 4.5 is strictly valid when the permeability of a material with respect to a liquid fluid has to be determined. A different expression of the left-hand side in Eq. 4.5 can be used when gaseous fluids flowing through the tortuous maze are considered. Indeed, the compressibility of the gas inside the porous material is taken into account by introducing the mean pressure in the core during the measurements (Eq. 4.6) [59]. The permeability with respect to gaseous fluids, provided by Eq. 4.6, approaches the absolute (liquid) permeability, calculated by Eq. 4.5, for increasing values of the mean pressure in the sample [60].

\[\frac{\Delta P}{H} \cdot \frac{\bar{P}}{P_{ext}} = \frac{\mu_f}{K_r} U_r^D \quad (4.6)\]
Where $\overline{P}$ is the average pressure between the high- and low-pressure ($ext$ subscript) sides of the specimen’s thickness. Equation 4.6, likewise Eq. 4.5, is strictly valid when the mass flux in the $r$-direction ($\rho U_D^r$) is conserved across the material’s thickness (i.e. porous samples bounded by solid walls) [60]. The aforementioned requirement is fulfilled when the standard methodology is used to calculate the average permeability of specimens having standard dimensions (i.e. frontal area and thickness) [60, 61, 62, 63, 64, 65] for which the fluid is confined to flow only in the direction of the specimen’s thickness. On the other hand, the average mass flux in the $r$-direction has to be considered ($\overline{\rho U_D^r}$ in Eq. 4.7) when the local permeability of full-scale porous structures, for which the fluid is not necessarily confined to flow only in the $r$-direction, has to be calculated.

$$\frac{\Delta P}{H} \cdot \frac{\overline{P}}{\overline{P_{ext}}} \cdot \rho_{ext} = \frac{\mu_f}{K_r} \rho U_D$$  (4.7)

In practical applications, the fluid follows different flow-paths in relation to the 3-D network and depending on the distribution of the actual boundaries of the structure creating substantial differences between the permeability of the standard sample with respect to that one of the full-scale components. The methodology proposed in this work is intended to provide the tools necessary to determine the local blowing capability for structures having complex shapes like those conceived for aerospace applications. In particular, the local blowing at the low-pressure side is measured with a technique, thoroughly described in §4.3, which employs hot-film anemometry while the correspondent mass flux at the high-pressure side is not explicitly known. In these circumstances, the local permeability, as by definition in Eq. 4.7, cannot be directly calculated and therefore the new concept of the effective permeability ($K_{eff}$) is introduced. The effective permeability is conceived, in this study, as the
local blowing capability of a porous material, with respect to a selected fluid, when a global pressure gradient is imposed across the material’s thickness. It indicates the pressure drop required to blow a prescribed coolant flow-rate across the porous lattice so that the average coolant mass flux in Eq. 4.7 is substitute with the mass flux measured at the low-pressure side of the specimen. Additionally, the mass flux measured on the low-pressure side is directly proportional to the Darcy’s velocity in the radial direction because, for the prescribed distance from the wall where the hot-film measurements have been performed [13], the coolant density can be considered constant and therefore Eq. 4.7 becomes equivalent to Eq. 4.6. The concept of effective permeability is extremely important for the characterization of full-scale components/structures made of composite materials because the presence of intrinsic defects, due for example to the manufacturing processes, can induce asymmetric flow-paths and non-uniform heat-transfer coupling between the coolant fluid and the porous matrix. The aforementioned phenomena can generate, respectively, concentrated mechanical loads and hot-spots on the exposed surfaces which can drastically modify the nominal thermomechanical response of the entire TPS. In addition, the knowledge of the actual blowing distribution represents a fundamental information (boundary conditions) for numerical aerothermal reconstructions.

4.3 Measurements Methodology and Data Analysis

In this section, the proposed methodology for the effective-permeability measurements is described along with the discussion of the most important guidelines necessary to correctly perform the measurements. A general description of the prototype specimen to be characterized is provided in §4.3.1 while the calibration procedure of the hot-film probe along with the experimental setup used to perform the permeability tests are reported, respectively, in §4.3.2 and §4.3.3. The procedure used
to characterize the internal structure of the prototype TPS component is described in §4.3.4.

### 4.3.1 Carbon-Carbon Cone Description

Figure 4.1 shows the full-scale component adopted in this study, manufactured by C-CAT, that is representative of a complex porous structure for which the use of a standard methodology for permeability measurements would be questionable due to the structural morphology and unavoidable defectologies. In fact, the axisymmetric composite mask showed in Fig. 4.1 is characterized by having variable thickness, high porosity prescribed at the manufacturing stage, diversified surface finishing (comparison of Fig. 4.1-a and Fig. 4.1-b) and boundary constraints on the structure. This generates highly asymmetric flow-paths inside the tortuous structure that cannot be captured by using the standard methodology.

The assembly of the C-C nose is reported in Fig. 4.2. The structure is composed of a truncated graphite cone that has the purpose of providing the coolant up to the internal plenum and mechanically supports the hollow transpiring C-C cone. The
external sliding mask along with an internal o-ring has been used to guarantee the sealing between the C-C mask and the graphite support. The variable thickness region has been sized, at the design stage, with the purpose of supporting a variable transpiration capability throughout it.

4.3.2 Hot-Film Anemometry

Hot-film anemometry has been used to conduct mass-flux measurements on the outer surface of the specimen [66, 67]. The measurement system is composed of a hot-film (TSI – 1210 – 20) of 50.8 $\mu m$ diameter used in conjunction with a constant temperature anemometer (CTA) TSI – 1750. The control resistance of the CTA has been selected in order to impose an overheat ratio of the hot-film equal to 1.35, which corresponded to a film temperature of about 441 $K$. The high nominal value of the film temperature with respect to the ambient temperature guaranteed a high sensitivity of the sensing film to the variations of the blowing velocity. The calibration of the hot-film sensor has been performed in the low-speed wind tunnel (LSWT) facility at the University of Texas at Arlington where simultaneous calibrated Pitot-static pressure measurements of the freestream velocity have been acquired. The
calibration process has been required in order to correlate the output voltage of the hot-film (Fig. 4.3) with the velocity measurements from the Pitot-static tube. The mass flux measured by the hot-film sensor can then be directly related to the velocity calculated from the Pitot measurements (constant air density during the calibration procedure).

Temperature corrections (Eq. 4.8) for the measurements during the permeability tests have been considered to compensate for the difference in temperature that naturally occurs between the calibration process and the permeability tests [66].

\[
V_{cal} = V_m \cdot \sqrt[2]{\frac{T_{HF} - T_{cal}}{T_{HF} - T_m}}
\]  

Where \( V_m \) is the output voltage from the hot-film while \( T_{HF}, T_{cal} \) and \( T_m \) are the hot-film temperature, the corrected or calibration temperature and the temperature measured during the permeability tests, respectively. The energy equation written for a control volume coincident with the film leads to an explicit expression of the Nusselt number, when the film temperature is kept constant by the CTA [67, 68, 69]. The Nusselt number can be, thus, correlated to the Reynolds number based on the diameter of the film [68, 69]. In this work, the use of a power-law correlation has
been chosen because, for the range of Reynolds number expected, it gives the lower root-mean-square error (RMSE) (Fig. 4.4).

The uncertainty on the calibration curve has been calculated from the errors pertaining to both the corrected hot-film output voltage \( V_{\text{cal}} \) and the velocity calculation from the Pitot-static tube measurements. The uncertainties on the voltage can be directly calculated from the standard deviation of the hot-film output signal while the uncertainties on the flow velocities have been estimated based on the propagation of the errors on the Bernoulli’s equation given the accuracy of the static and total pressure sensors.

### 4.3.3 Surface Blowing Measurements

The proposed technique for the characterization of the full-scale specimen in terms of the local effective permeability leverages on the capability of being able to measure the local mass flux blown from a porous wall at ambient pressure, once the stagnation conditions are obtained on the other side of the specimen (internal plenum chamber in Fig. 4.2-a). A traversing system has been used to move the hot-film parallel to the specimen’s surface (Fig. 4.5-a). Additionally, a calibration
Figure 4.5. Test setup for permeability measurements. a) Assembly; b) Enlarged view of the probed area.

ductile (Fig. 4.5-b) and an optical system, used for the alignment procedure, have been used to guarantee a perpendicular position between the hot-film and the porous wall of the sample. It allowed minimizing the angularity effects that would impact the hot-film measurements [66]. The total pressure in the plenum of the sample ($P_{st}$) has been measured inside the external plenum chamber of Fig. 4.5-a that is located between the C-C nose-tip and the mass-flow controller. The dimension of the external plenum guaranteed the same pressure conditions inside the internal chamber between the C-C mask and the graphite support (Fig. 4.2-a). The pressure losses inside the pipe that connect the external to the internal plenum have been calculated to be negligible. Both the output signals from the pressure transducer and from the CTA system have been acquired by using a 16-bit data acquisition system (DAQ in Fig. 4.5-a).

The mass-flux measured by maintaining the hot-wire at an appropriate distance from the surface, has been directly related to the blowing velocity by means of the calibration curve in Fig. 4.4, once the density change at the measurement points has
been neglected. The last assumption has been verified by performing a Schlieren test, using nitrogen, with the scope of detecting any sharp changes of density at the exit of the pores. The tests performed for higher values of the mass-flow rate with respect to those used for the permeability tests have not showed any remarkable variation of the density on the straight sidewalls of the specimen (Fig. 4.6). A significant change of the refraction index and, thus, of the density of the coolant fluid blown from the pores has been detected near the stagnation point (SP) region of the nose-tip. This phenomenon is due to the presence of two radial delaminations which span through the entire nose-tip length [4].

Additional Schlieren visualizations, using helium as working fluid, have been performed for different longitudinal planes (referred to Fig. 4.8-a) with the purpose of detecting, qualitatively, the transpiring capability along the specimen’s sidewall (Fig. 4.7). A color-map for the Schlieren images has been added to emphasize the blowing profile (Fig. 4.7-b). Figure 4.7 shows the injection points near the stagnation region, the blowing on the straight walls of the specimen and a near-zero transpiration in the corner region of the nose-tip. The Schlieren tests on different cone’s sections revealed different blowing profiles which, in turn, highlight the presence of an asymmetric blowing capability due to the anisotropy of the internal void structures.
Figure 4.7. Schlieren photography by using helium as coolant (average field). a) Original image; b) Contrast mask.

The nominal conditions selected for the transpiration measurements are reported in Table 4.1.

Table 4.1. Nominal test conditions used for the permeability tests

<table>
<thead>
<tr>
<th>Test #</th>
<th>$\dot{m}_{\text{air}}, \text{SLPM}$</th>
<th>$\dot{m}_{\text{air}}, g/s$</th>
<th>$P_{\text{SP}}, \text{psi}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$12 \pm 3$</td>
<td>$0.259 \pm 0.067$</td>
<td>$34.80 \pm 0.38$</td>
</tr>
<tr>
<td>2</td>
<td>$14 \pm 3$</td>
<td>$0.302 \pm 0.067$</td>
<td>$38.02 \pm 0.33$</td>
</tr>
<tr>
<td>3</td>
<td>$16 \pm 3$</td>
<td>$0.345 \pm 0.067$</td>
<td>$39.83 \pm 0.61$</td>
</tr>
<tr>
<td>4</td>
<td>$18 \pm 3$</td>
<td>$0.388 \pm 0.068$</td>
<td>$45.35 \pm 0.06$</td>
</tr>
<tr>
<td>5</td>
<td>$20 \pm 3$</td>
<td>$0.431 \pm 0.068$</td>
<td>$48.71 \pm 0.55$</td>
</tr>
</tbody>
</table>

The minimum value of the mass-flow rate has been chosen to minimize the influence of the noise from the background velocity in the room (maximization of the signal-to-noise ratio) while the upper bound value has been imposed by the maximum differential pressure able to guarantee the structural integrity of the specimen. The nominal test conditions of Table 4.1 have been targeted for three of the four longitudinal cut planes shown in Fig. 4.8-a. Eleven locations for each plane have been surveyed with the hot-film probe (Fig. 4.8-b).

The longitudinal spacing of the control points (CP) has been prescribed to provide a good spatial resolution for the calculation of the effective permeability...
(Fig. 4.8-b) and to scan the transpiring properties of the mask for the following regions:

- the stagnation point (SP) region;
- the straight sidewall above the graphite support (CP1-CP3);
- the straight sidewall above the plenum region (CP4-CP9);
- the corner region (CP10);

Two orientations of the sensing film have been used to probe the entire stagnation point region. In particular, \(SP – Horizontal\) and \(SP – Vertical\) represent the position of the hot-film perpendicular to the 90°-270° plane (Fig. 4.8-a and Fig. 4.9-a) and perpendicular to the 0°-180° plane (Fig. 4.9-b), respectively.

For this study, the distinction between the longitudinal and transversal orientation of the hot-film probe has been required because of the two radial delaminations (§4.3.4) that created a strong asymmetric velocity field in correspondence of the SP region. Only the orientation perpendicular to the axis of the cone (Fig. 4.5-b and Fig. 4.8-b) has been considered for the remaining 10 points along the specimen’s surface because, contrarily to SP region, the survey near the overall diameter of the
control surfaces (CSs) has not revealed any significant difference in terms of average velocity. For the same reason, the velocity measurements have been collected at the center of the CS and, thus, all results have been referred to the control points (CPs). The round control surfaces have a diameter $D_{CS} = 0.1\ \text{in}$. The dimension of the control surfaces has been based upon the minimum size of the representative area that allows considering the blowing statistically constant across the surveyed control points. In particular, the characteristic diameter of the control surfaces, which determines also the spatial resolution of the effective-permeability map, has been defined by a convergence criterion based on the average-porosity calculation (§4.3.4). The mass-flux measurements have been recorded once the plenum pressure reached the steady-state (Fig. 4.10) and, once the hot-film probe has been placed at the center of each control surface, for the time needed to obtain a meaningful measurement in terms of uniform fluctuations of the mean output voltage (i.e. $t \geq 5\ \text{s}$).

The influence of the probe distance from the wall has been analyzed by applying the theory of fluid flow through perforated plates/screens that allowed identifying the range of optimal distances for the mass-flux measurements [70, 71]. The correct probing distance is fundamental in order to obtain meaningful velocity measurements in terms of minimum fluctuations with respect to the mean-velocity field. In fact,
in the near-wall region, single jets are discernible with a pattern that mimic the “perforated” arrangement of the porous surface while, at the merging distance \((L_m)\), the jets coalesce together [70]. Under these circumstances, the hot-film crosses low velocity and high velocity regions with the periodicity of the pores pattern [71, 72] when it is moved parallel to the surface at a distance \(r < L_m\). The velocity fluctuations increase consistently at the merging distance \((r = L_m)\) while for \(r\) slightly higher than the merging distance a near-continuous blowing velocity can be detected [70]. The merging length depends, mainly, to the network-mesh size \((M)\) and to the Reynolds number based on the channels’ diameter \((Re_{ch})\). Specifically, below a certain critical Reynolds number \((Re_{ch} \approx 20)\), the spreading angle of the jets decreases with the Reynolds number and the merging distance follows the relation \(L_m \propto M \cdot Re_{ch} [70]\). When \(Re_{ch} > Re_{cr}\) (turbulent jets), the spreading angle of the jets increases with \(Re_{ch}\) and the merging distance change its dependence with respect to the Reynolds number \((L_m \propto M \cdot Re_{ch}^{-1}) [70]\). A preliminary test campaign, aimed to shake down the test setup for permeability measurements, highlighted the impact of the distance between the hot-film and the wall on the velocity measurements. The

Figure 4.10. Pressure measured in the external plenum for some of the air flow-rate utilized.
Figure 4.11. Preliminary permeability tests at $d_w = 0.05$ in from the wall. a) Linear curve fits for the gathered data; b) Hot-film velocity measurements at CP7.

measurements performed by using nitrogen as coolant fluid and keeping the hot-film at $d_w = 0.05$ in from the wall provided, for most of the points and planes surveyed, scattered data (Fig. 4.11-a) that cannot be correlated among each other using a linear curve fit (Darcy’s law). It is mainly due to the high-velocity fluctuations coming from the traces of the hot-film measurements (Fig. 4.11-b), which are comparable to the nominal value of the mean-velocity field. This particular phenomenon is typical when the velocity measurements are performed within the interaction region of two or more co-flowing jets.

The statistical analysis of the dimensions and distribution of the void structures inside the porous network [4] have been used, thus, to define the probing distance for which a near-contiguous transversal wall velocity can be obtained while avoiding any local effect, due to the intermittency of the material/pore interface. The above mentioned analysis has been performed by using the output data from the X-ray computed tomography scan of the prototype specimen (§4.3.4).
4.3.4 Characterization of the Internal Porous Structure

The output data from the CT-scan machine that resides at the Advanced Materials and Structures Lab (AMSL) of the University of Texas at Arlington have been used to characterize the internal structure of the carbon layered porous material in Fig. 4.1. The specimen has been scanned with the X5000 X-ray CT system manufactured by North Star Imaging, Inc. The main parameters used for the scan of the specimen are reported in Gulli et al [4]. In particular, a 12X geometric magnification used for the scan has generated a spatial resolution of approximately 10.6 \( \mu m/pixel \) for the 3-D reconstruction of the specimen. The reconstruction of the specimen is represented by the 3-D point cloud where at each point is assigned a gray value that range from zero to 65536 and approximately represents the material density at the respective coordinate. The region near the tip, characterized by a variable thickness distribution (Fig. 4.2-b), has been scanned at the maximum achievable resolution which defines both the dimension of the smallest void structure detectable inside the material, and the sharpness of the voids’ contour. The 2-D segmentation of several planes, longitudinal and perpendicular to the centerline of the sample (Fig. 4.12 and Fig. 4.13, respectively), has been used for defining two fundamental parameters needed to perform correctly the permeability tests by hot-film anemometry:

- The characteristic dimension of the control surface (§4.4.2) where the mass-flux measurements have been performed. It has been defined by a convergence criterion based on both the internal porosity, calculated from the 2-D CT-scan images, and on the volumetric porosity calculated from the 3-D digital reconstruction of the prototype sample.

- The correct distance of a hot-film sensor from the wall that has been calculated by using the statistical distribution of number and dimensions of the void struc-
Figure 4.12. Longitudinal cut of the 3-D reconstruction. a) 0° Cut-plane; b) 180° Cut-plane (both cut angles are referred to Fig. 4.8-a).

Figure 4.13. Transversal cut of the 3-D reconstruction. a) Constant-thickness region ($H = 0.15$ in); b) Variable-thickness region ($H = 0.13$ in); c) Nose-tip region ($H = 0.1$ in).

The statistical analysis of the number and dimensions of the void structures inside the specimen has been necessary to establish both the range of variation of the Reynolds number and the network-mesh size (§4.4.3). Additionally, the variation of the Reynolds number along the specimen sidewall due to the different blowing capability of the material has been also fundamental to define the flow regime and, thus, to verify the use of the Darcy’s law for the calculation of the effective permeability (§4.2).
A supplementary analysis by using the 3-D digital reconstruction of the cone-tip (Fig. 4.14) has been carried out independently by the research group of the AMSL to calculate the average volumetric porosity \( \epsilon_{3-D} \) and the dimensions of the REV [4]. It allows comparing the results obtained from the 3-D analysis with respect to those obtained by using the 2-D images (Fig. 4.12 and Fig. 4.13).

4.4 Results and Comparisons

In this section, the results deriving from the analysis of the CT-scan images have been used to define the aforementioned guidelines needed for the correct execution of the permeability tests by using hot-film probes. A preliminary analysis has been performed in §4.4.1 to assess the most suitable contouring method necessary to calculate both the internal porosity, by analyzing 2-D images \( \epsilon_{2-D} \), and the volumetric porosity \( \epsilon_{3-D} \). Both the average porosity values have been used to infer the minimum dimensions of the area to be probed (§4.4.2) while the analysis of distribution and dimension of the internal voids’ structure has been used to estimate the correct distance of the sensing film from the wall (§4.4.3).
4.4.1 Porosity Measurements

Several longitudinal sections having variable span angles (Fig. 4.12) and transversal cut planes (Fig. 4.13) have been analyzed to calculate the 2-D porosity of the specimen. An automatic procedure has been adopted to calculate the void-to-material ratios once the interrogation area and the contouring method for the images have been properly selected. An annulus region having radius that change accordingly to the variation of the specimen’s thickness has been selected as the interrogation area for the transversal cut (dashed border in Fig. 4.15-a). The interrogation area for the longitudinal cut (dashed border in Fig. 4.15-b) does not include the regions very close to the tip of the samples because of an out-of-focus blemish.

The nominal value of the gray-scale threshold, under which the presence of an empty structure, typically containing air, has been determined for the analysis of the 2-D images by selecting three appropriate threshold selection methods (TSM) [73]. Specifically:

- Minimum TSM
- OTSU TSM
- Moment TSM
These methods are characterized by employing different thresholding/contouring categories and are commonly used in several research fields (e.g. geology, medicine, engineering etc.) to identify density variations across homogenous and non-homogenous media. For example, they are successfully used in petroleum engineering for the calculation of the natural porosity of soils (unconsolidated media) and porous rocks (consolidated media) [74, 75]. The utilization of the aforementioned methods for aerospace applications is relatively new and of particular interest because of the growing use of composite materials that have to be characterized at the full-scale level, by using non-intrusive techniques, due to their intrinsic defectology introduced during the manufacturing processes [34, 75, 76]. The Minimum method belongs to the histogram shape-based thresholding methods for which the background and foreground pixels are detected based on the peaks, valleys and curvatures of the gray-levels histogram [77]. The OTSU method belongs to the clustering-based contouring methods where the gray values are clustered separately in foreground and background classes and the optimum threshold value is computed in order to minimize the weighted sum of within-class variances [77, 78]. The Moment method belongs to the attribute-similarity contouring methods which implement an algorithm to search a measure of similarity between the gray-level image, considered as the blurry version of an ideal binary image, and the binarized image [77]. Figure 4.16 shows the same portion of material analyzed by the aforementioned TSMs.

The Minimum TSM underestimates the real porosity because the low value of the threshold gray-level does not allow considering the smaller channels for the computation of the void-to-material ratio. Conversely, the Moment TSM overestimates the real porosity because the higher threshold value computed takes into account of all the entities having a gray-tone slightly darker than the solid material. The aforementioned darker spots are not necessarily empty spaces, but they can represent
density changes across the carbon matrix and, thus, they are not relevant for the porosity calculation. Additionally, the high value of the threshold gray-level substantially increases the boundaries of the real cavities creating extended defects. In this perspective, the *OTSU* method has been chosen to provide the reference gray-level threshold necessary to calculate the internal porosity of the sample from 2-D images ($\epsilon_{2-D}$). In fact, the *OTSU* method is highly recommended when working with images having extremely defined peaks in the gray-levels histogram (Fig. 4.15-b) [77]. The local and average porosity values for both the longitudinal and the transversal cut planes are reported in Table 4.2 and Table 4.3, respectively. Only the results for few sectioning planes have been included in the tables.

The nominal values of the cut angles in Table 4.2 are referred to Fig. 4.8-a. The uncertainties on the porosity for each cut have been calculated by moving the optimal threshold, estimated by the *OTSU* method, of few counts ($\pm5\%$) in order to include some minor voids detected by eye-inspection of each single image [75]. The 3-D reconstruction of the prototype sample has been also used, by the AMSL research group, to independently calculate the average volumetric porosity.
Table 4.2. 2-D local and average porosity calculated for sectioning planes cut longitudinal to the C-C mask axis

<table>
<thead>
<tr>
<th>Longitudinal-plane angle</th>
<th>90°</th>
<th>140°</th>
<th>190°</th>
<th>230°</th>
<th>270°</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\epsilon_{2-D}$</td>
<td>15.87 ± 0.12</td>
<td>15.22 ± 0.22</td>
<td>13.64 ± 0.15</td>
<td>15.24 ± 0.31</td>
<td>16.42 ± 0.33</td>
</tr>
<tr>
<td>$\epsilon_{2-D}'$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Longitudinal-plane angle</th>
<th>272°</th>
<th>274°</th>
<th>276°</th>
<th>278°</th>
<th>280°</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\epsilon_{2-D}$</td>
<td>16.09 ± 0.27</td>
<td>16.07 ± 0.12</td>
<td>15.96 ± 0.16</td>
<td>16.67 ± 0.31</td>
<td>16.01 ± 0.15</td>
</tr>
<tr>
<td>$\epsilon_{2-D}'$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 4.3. 2-D local and average porosity calculated for sectioning planes cut transversal to the C-C mask axis

<table>
<thead>
<tr>
<th>Transversal-plane angle</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\epsilon_{2-D}$</td>
<td>16.57 ± 0.10</td>
<td>13.33 ± 0.11</td>
<td>14.59 ± 0.15</td>
<td>14.20 ± 0.24</td>
<td>13.55 ± 0.11</td>
</tr>
<tr>
<td>$\epsilon_{2-D}'$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Transversal-plane angle</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\epsilon_{2-D}$</td>
<td>13.79 ± 0.22</td>
<td>14.39 ± 0.23</td>
<td>13.98 ± 0.11</td>
<td>14.03 ± 0.30</td>
<td>13.66 ± 0.22</td>
</tr>
<tr>
<td>$\epsilon_{2-D}'$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

($\epsilon_{3-D}$). The transverse slices of the nose-tip have been used for the calculation of the volumetric porosity by selecting the coordinate of one point on the transversal plane and by extending the third direction accordingly to the side dimension of the volume considered [4, 31]. The transversal sectioning planes, stacked for the computation of the void-to-material ratio, are chosen to maximize the number of extractions from the probing volume and the resulting spacing is approximately 10.6 $\mu$m. Two contouring methods, which have been proven to be promising for the analysis of composite materials [34, 79], have been used to determine the optimal threshold for the 3-D digital reconstruction.

- *K-means* TSM (similar to the *OTSU* TSM)
- *Mean density* TSM
The optimal threshold value for the $K$-means method has been assessed to be gray level 5104 of the total 65536 gray levels available while the gray level 5125 has been found to be the optimum threshold value for the Mean density method. The Mean density TSM allowed assessing also the variation in the threshold definition by finding noisy variation of the air ($\pm 10$) and material gray values ($\pm 40$) resulting in the acceptable threshold variation from 5100 to 5150. Figure 4.17 shows a zoom of the gray-levels histogram of the scan with the optimum threshold values determined by the above mentioned algorithms [4, 31].

The average volumetric porosity values are reported in Table 4.4.

Table 4.4. Average volumetric porosity calculated by two additional contouring methods

<table>
<thead>
<tr>
<th>$\epsilon_{3-D}$ (K-means TSM)</th>
<th>Constant-thickness region</th>
<th>Variable-thickness region</th>
<th>Tip-region</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\epsilon_{3-D}$ (Mean density TSM)</td>
<td>10.4%</td>
<td>10.0%</td>
<td>9.5%</td>
</tr>
<tr>
<td></td>
<td>(11.1 ± 6.3)%</td>
<td>(10.5 ± 4.8)%</td>
<td>(9.9 ± 7.0)%</td>
</tr>
</tbody>
</table>
Table 4.4 shows that the volumetric porosity calculated from the 3-D reconstruction provides slightly different results with respect to the nominal porosity calculated from the 2-D sections (comparison of Table 4.2 and Table 4.3). In fact, the internal porosity calculated from a discrete number of longitudinal and transversal cuts ($\epsilon_{2-D} \cong 14.6\%$) is higher than the volumetric porosity ($\epsilon_{3-D} \cong 10.5\%$) by 39%, if the Mean density is considered [34]. The deviation between the two independent calculations is mainly due to the different total number of gray levels considered. The 256 gray levels used for the analysis of the 2-D images (8-bit pictures) allow detecting only one peak in the histogram (Fig. 4.15-b). The 65536 gray levels used for the analysis of the 3-D reconstructed domain allow distinguishing two peaks (Fig. 4.17) that sharply separate the background pixels (air) from the foreground pixels (material). However, the deviation in terms of nominal porosity does not affect the results in terms of dimensioning of the probed area because the analyses of the reference elementary surface (RES) and reference elementary volume (REV) provide analogous results (§4.4.2).

4.4.2 Dimensions of the Probed Area

The dimension of the control surface is based upon the size of the characteristic area that allows obtaining meaningful local measurements in terms of minimum mean-velocity fluctuations. The mass-flux measurements, obtained by using hot-film anemometry, have been associated to selected control points which corresponds to measurements collected at the center of the correspondent control surface. A detailed description of the procedure used to define the above mentioned dimension, by introducing the RES and/or the REV, is reported in this section. Squared probing boxes, generally of different dimensions, have been used to survey different areas of the CT-scan images and, thus, to define the RES by detecting a plateau of the porosity
Figure 4.18. RES calculation for two representative longitudinal cut planes. A reference dimension, coincident with the spatial resolution of the CT-scan (10.6 \( \mu \text{m/pixel} \)), has been applied to each image with the purpose of defining the nominal dimensions of the probing surfaces. The pixels-to-inch conversion scale has been also used to verify the nominal variation of the specimen thickness prescribed at the design stage.

Figure 4.18 shows that both the plane sections having higher and lower porosity present similar characteristics in terms of porosity convergence for a probing surface of 0.09 in \( \times \) 0.09 in. The maximum variance of the porosity with respect to the average value is about 8% (lower-porosity plane) and 6% (higher-porosity plane) for the probing surface named Area6 in Fig. 4.18. Additionally, the nominal values of the porosity for the larger probing surface are within the range of porosity values calculated on the entire conical profile of the nose-tip (Table 4.2). Similar results in terms of RES have been obtained for the transversal cut. The same procedure described above has been applied by the research group of the AMSL to calculate, in-
dependently, the REV by defining cubic volumes having side dimension varying from 0.02 \textit{in} to 0.08 \textit{in}. One hundred 3-D coordinates distributed in selected transversal planes of the nose-tip have been used for the calculation of the REV (Fig. 4.19). The calculated volumetric porosity does not show a monotonic convergent trend because it starts increasing for side dimensions bigger than 0.08 \textit{in}. In this perspective, the side dimensions of the cubic volume bigger than 0.08 \textit{in} have not been considered for the REV calculation because they exited from the borders of the nose-tip and, thus, the probing volume started to capture areas outside the shape of the cone with the consequent overprediction of the void to material ratio. The aforementioned geometrical issue has been overcome, for the RES calculation in Fig. 4.18, by shaping the reference surfaces around the curved borders of the cone (Fig. 4.15). Figure 4.19 shows the REV determination starting from the transversal cut of a constant-thickness region (Fig. 4.13-a) and of a variable-thickness region (Fig. 4.13-b).

The comparison of Fig. 4.18 and Fig. 4.19 shows similar trends in terms of porosity distribution, respectively, for the square surfaces and the cubic volumes selected. The maximum variance of the volumetric porosity with respect to the average value is about 33\% (Fig. 4.19-a) and about 40\% (Fig. 4.19-b) for the probing
Figure 4.20. Hot-film voltage traces across CS9 ($\dot{m}_{N2} = 16$ SLPM). a) 0.05 in backward to CP9; b) center of the CS; c) 0.05 in forward to CP9.

volume named Volume7 in Fig. 4.19. The higher scattering of the data from the 3-D sectioning with respect to the results in Fig. 4.18 could be attributed to the high anisotropy of the tortuous structures characteristics of consecutive sectioning planes. In this work, a squared CS having dimension of $D_{CS} = 0.1$ in has been selected to survey the blowing capability of the prototype cone in order to obtain meaningful velocity measurements ($D_{CS} \geq \max(D_{RES}; D_{REV})$) and to maintain, at the same time, a high spatial resolution ($D_{CS}$ as small as possible). A preliminary investigation by using a hot-film probe confirmed the statistical steadiness of the mean-velocity field measured across the control surface for all the locations and flow rates used (Fig. 4.20). The control surface, to which Fig. 4.20 is referred (CS9), corresponds to the region above the internal plenum that has variable thickness (Fig. 4.2-a and Fig. 4.8-b).

The variation of the mean velocity across the CS, for all the locations probed and flow rates used for the preliminary test campaign, ranges between 0.5% and 4.8%. For this reason, the mass-flux measurements have been collected at the center of the CS and, thus, are related to the control points [13].
4.4.3 Hot-Film Probe Distance from the Wall

In this section, the statistical analysis of the voids distribution inside the C-C specimen has been aimed to define the porosity of the external surface (superficial porosity, $\epsilon_{\text{sup}}$) and the characteristic dimension of the channels. The former is necessary to calculate the local flow velocities (seepage velocity, $u$) while the latter is used to define the Reynolds number that is fundamental to determine the flow regime and, thus, to estimate the coalescence distance of the jets (§4.3.3). The variation of the Reynolds number along the specimen sidewall, that is due to the different blowing capability of the TPS structure, is also fundamental to define the flow regime and, thus, to justify the use of the Darcy’s law for the calculation of the effective permeability (§4.2). In addition, the statistical characterization of the empty structures inside the porous sample can be used in support of numerical modeling for the thermomechanical response of porous structures by using the equivalent medium theory (EMT) [51, 80, 81]. The analysis of only the 2-D longitudinal and transversal sectioning planes has been used for the statistical description of the porous media since they provided results fairly in agreement with the analysis of the 3-D reconstruction (§4.4.2). Different network elements have been recognized and classified in base of their characteristic length scales for the statistical description of the structure’s internal morphology. In particular, for consolidated fibrous/layered materials, the porous lattice has been subdivided in a series of channels/throats, pores and caverns/chambers [51]. The channels can be defined empty spaces having a low aspect ratio (length/diameter) combined to the smaller diameter within the porous network, while the pores are the void structures connecting two or more channels. The pores are usually distinct by the channels because of their larger diameter [51]. The average number of channels converging to a single pore is named coordination number ($C'$) of the porous structure [51]. The caverns, which for layered materials
are usually identified as delaminations, are defined as empty structures having the higher aspect ratio within the porous network. The threshold values that allowed classifying the voids based on their length scales have been set once the network elements have been clearly identified by eye-inspection of the CT-scan images (Fig. 4.21).

The length-scales thresholds selected for this study are reported in Table 4.5.

Table 4.5. Length-scales separation for the void elements constituting the C-C nose-tip

<table>
<thead>
<tr>
<th>Network elements</th>
<th>Channels/Throats</th>
<th>Pores</th>
<th>Chambers</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aspect ratio (L/D)</td>
<td>&lt; 30</td>
<td>&lt; 30</td>
<td>&gt; 30</td>
</tr>
<tr>
<td>Diameter (D)</td>
<td>&lt; 40 (\mu m)</td>
<td>&gt; 40 (\mu m)</td>
<td>ND</td>
</tr>
</tbody>
</table>

The statistical analysis of the distribution and size of the network elements inside the full-scale specimen has been performed, once the length-scales separation in Table 4.5 have been adopted. Figure 4.22 shows the dimensions of throats and pores for a longitudinal section having the average porosity calculated in Table 4.2.

The two annular caverns of about 65 \(\mu m\) thickness (Fig. 4.22-a) are extended along the entire cone-tip and they are detectable for almost all the longitudinal
Figure 4.22. Dimensions of void structures for the longitudinal cut at 230°. a) Caverns’ diameter; b) Throats’ diameter; c) Throats’ length; d) Pores’ diameter; e) Pores’ length.

and transversal cut planes analyzed. The throats’ diameter (Fig. 4.22-b), the pores’ diameter (Fig. 4.22-d) and the pores’ length (Fig. 4.22-e) show to be centered around an average value. It indicates the possibility to model the highly anisotropic internal porous structure with an equivalent medium composed of slots (equivalent to the caverns) cylindrical channels and pores [51, 80, 81]. The throats’ length (Fig. 4.22-c), that has been measured by following the tortuous paths of the channels, shows a higher variance with respect to the dimensions of the other network elements. The higher data scattering of the throats’ length can be due to either, or both, the
variable thickness of the specimen and/or the enhanced channel length due to the tortuosity ($\tau = l_t/l_{st}$, where $l_t$ and $l_{st}$ are the lengths of the tortuous and straight channels, respectively). Here, the influence of the latter could be neglected because the tortuosity for both the transversal and the longitudinal cut planes, which has been calculated based on a limited count of hundred samples of the channels, is nearly constant to $\tau = 1.32 \pm 0.1$. In addition, the left region of Fig. 4.22-c ($0 < \#Samples < 80$) corresponds to the tip-region (lower thickness) while the right region of Fig. 4.22-c ($80 < \#Samples < 140$) corresponds to the variable- and constant-thickness region. These results open the scenario of using an equivalent medium having a variable channel’s length that is proportional to the thickness variation. However, only the statistical distribution of voids diameter (Fig. 4.23) has been reported for the purpose of defining both the Reynolds number and the porosity of the external surface.

The average diameters along with the standard deviations of the throats, pores and caverns (Fig. 4.22) have been used in a normal distribution function with the scope to verify if the average diameters provided a good representation of the statistical distribution of the network elements (Fig. 4.24) [53].
Figure 4.24. Probability density functions corresponding to a Gaussian distribution for throats’ and pores’ diameters.

The comparison of Fig. 4.23 and Fig. 4.24 shows that both the throats’ and the pores’ diameters follow the Gaussian distribution centered on the average diameters. The probability density function of the pores’ diameter is wider with respect to that one referred to the throats’ diameter because of its higher relative scattering of the data (comparison of Fig. 4.22-d and Fig. 4.22-b). The same analysis can be performed for the lengths of the void structures with the purpose of defining the equivalent model for the internal structure of the porous material. The number of empty structures inside the RES has been estimated by analyzing few longitudinal and transversal cut planes and avoiding the out-of-focus regions (Table 4.6).

Table 4.6. Average number of voids elements per unit RES

<table>
<thead>
<tr>
<th>Network elements</th>
<th>Longitudinal throats</th>
<th>Radial throats</th>
<th>Pores</th>
<th>Chambers</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average counts</td>
<td>12.3 ± 4.9</td>
<td>24.8 ± 3.3</td>
<td>11.8 ± 2.1</td>
<td>1.8 ± 0.4</td>
</tr>
<tr>
<td>(longitudinal cuts)</td>
<td>12.8 ± 3.1</td>
<td>22.0 ± 3.5</td>
<td>10.0 ± 2.1</td>
<td>2.0 ± 0.1</td>
</tr>
<tr>
<td>Average counts</td>
<td>12.8 ± 3.1</td>
<td>22.0 ± 3.5</td>
<td>10.0 ± 2.1</td>
<td>2.0 ± 0.1</td>
</tr>
<tr>
<td>(Transversal cuts)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Table 4.6 shows that the number of void structures across the material thickness is approximately equal for longitudinal and transversal cut planes. The uncertainties have been calculated starting from the standard deviations on the counts of the void elements for each longitudinal and transversal sectioning plane. The distinction between longitudinal throats (x-direction in Fig. 4.2-a) and radial throats (r-direction in Fig. 4.2-a) has been needed to define the average number of void structures that reach the free boundaries and, thus, to estimate the superficial porosity. Only a certain quantity of all the radial throats detectable per unit RES, ranging between 6 and 8 elements, reaches the external surface for both the longitudinal and transversal segmentations. A circular cross section for the throats has been assumed due to the statistical absence of channels extension in the third dimension when two contiguous sectioning planes (1° spacing) have been analyzed. At this stage, the external porous surface of the cone has been modeled in order to estimate the superficial porosity ($\epsilon_{\text{sup}}$) and, thus, to calculate the Reynolds number based on the channels’ diameter. In particular, the CS has been schematized as a perforated flat surface having uniform porosity that is determined by both the average channel’s diameter and by the average number of channels reaching the external surface. In this model, the curvature of the outer surface of the specimen has been neglected due to the small dimensions of both the CS and the interrogation area of the hot-film compared to the overall external diameter of the C-C mask. The porosity of the external surface has been therefore estimated by applying the above mentioned model based on a perforated plate having a variable number of circular channels per side of the CS ($6 < \text{counts} < 8$). The average diameter of the channels, $D_{ch} = (29.47 \pm 5.57) \mu m$, has been imposed based on the results reported in Fig. 4.24. The resulting superficial porosity is $\epsilon_{\text{sup}} = (0.5 \pm 0.1)\%$. The combined use of the average throats’ diameter and porosity of the external surface ($\epsilon_{\text{sup}}$) allows estimating the Reynolds number
with the purpose of assessing the range of variation of the coalescence distance of the jets and, thus, to correctly define the distance of a hot-film probe for the permeability tests [13]. Additionally, the statistical characterization of the internal structures of the porous material has been used also to determine the range of variation of the network-mesh size \((M)\). The coalescence distance of the jets has been estimated by using the CT-scan images of the cone coupled to preliminary velocity measurements obtained by placing the hot-film probe at \(d_w = 0.05\) in from the wall with the scope to quantify the range of variation of \(Re_{ch}\). The merging distance for the selected specimen changes along the specimen sidewall since the blowing velocity and, thus, the Reynolds number based on the channels’ diameter are variable. An estimate of the minimum and maximum merging distance of the single jets from the porous wall is given, as already mentioned in §4.2, by Eq. 4.9 and Eq. 4.10, respectively.

\[
(L_m)_{\text{min}} \approx M_{\text{min}} \cdot (Re_{ch})_{\text{min}} \approx 0.048\ \text{in} \tag{4.9}
\]

\[
(L_m)_{\text{max}} \approx M_{\text{max}} \cdot (Re_{ch})_{\text{max}} \approx 0.140\ \text{in} \tag{4.10}
\]

The minimum and maximum network-mesh sizes measured from the 2-D CT-scan images are \(M_{\text{min}} \approx 0.012\) in and \(M_{\text{max}} \approx 0.014\) in, respectively. The Reynolds number based on the average channel’s diameter \((Re_{ch} = U_D D_{ch})/(\epsilon_{sup} \nu))\) has been estimated from the preliminary velocity measurements of the hot-film \((U_D)\) obtained for selected control points (Fig. 4.8-b) and from the average diameter of the channels (Fig. 4.24) \((\dot{m}_{\text{air}} = 0.26\ \text{g/s}, (Re_{ch})_{\text{min}} = 4; \dot{m}_{\text{air}} = 0.43\ \text{g/s}, (Re_{ch})_{\text{max}} = 13)\). Both the lower and the higher flow rates used for the permeability tests (Table 4.1) have been imposed to determine the range of variation of the Reynolds number. Additional blowing tests, using air as working fluid, have been performed at prescribed distances.
Figure 4.25. Velocity measurements at CP8 for the 270° longitudinal cut ($\dot{m}_{air} = 10\ SLPM$). a) $d_w = 0.05\ in$; b) $d_w = 0.1\ in$; c) $d_w = 0.15\ in$.

Figure 4.26. Velocity measurements at CP8 for the 270° longitudinal cut ($\dot{m}_{air} = 16\ SLPM$). a) $d_w = 0.05\ in$; b) $d_w = 0.1\ in$; c) $d_w = 0.15\ in$.

from the wall in order to verify the overall prediction on the average coalescence distance of the jets (Fig. 4.25 and Fig. 4.26).

The coalescence distance, as predicted by Eq. 4.9 and Eq. 4.10, increased with the imposed air flow-rate. The closest distance of the probe from the wall (Fig. 4.25-a, Fig. 4.26-a), for the range of flow rates considered, produced standard deviations on the transversal wall velocity values larger than the nominal value of the mean-velocity ($\sigma_U > U_D$). The correct probing distance for $10\ SLPM < \dot{m}_{air} < 14\ SLPM$ is around $d_w = 0.1\ in$, (Fig. 4.25-b) because the standard deviation on the velocity measurement is lower than 50% with respect to the mean velocity. The farther distance from the wall (Fig. 4.25-c) has not been suitable for the permeability tests.
because of the natural velocity decay far downstream the coalescence distance of the jets. The correct probing distance for the remaining flow rates used ($\dot{m}_{\text{air}} = 16 \div 20 \text{ SLPM}$) is around $d_w = 0.15 \text{ in}$ (Fig. 4.26-c). The highest velocity fluctuations detectable in Fig. 4.26-b have been determined by the capturing of the merging location of co-flowing jets where local instability of the flow is generated [70]. The comparison of the experimental results with the predictions, obtained via Eq. 4.9 and Eq. 4.10, corroborates the validity of the methodology used to define the range of variation of the optimal hot-film distance from the porous wall. The mass-flux measurements reported in Fig. 4.25 and Fig. 4.26 have been used to assess the impact of the probing distance on the effective-permeability calculations (Fig. 4.27). Each point of the graphs in Fig. 4.27 corresponds to a different air-flow rate. All the measurements have been taken at the same location ($CP8$).

Figure 4.27-a shows the results corresponding to the blowing measurements collected at the prescribed distances defined by Eq. 4.9 and Eq. 4.10 (i.e. $d_w = 0.1 \text{ in}$ for $\dot{m}_{\text{air}} = 10 \div 14 \text{ SLPM}$ and $d_w = 0.15 \text{ in}$ for $\dot{m}_{\text{air}} = 16 \div 20 \text{ SLPM}$). For each condition, the hot-wire distance has been adjusted accordingly. Figure 4.27-b and Fig. 4.27-c report the results corresponding to the blowing measurements collected.
at a constant distance from the wall. Specifically, in both cases the hot-wire distance has not been adjusted with the varying flow rates but has been kept at a value of $d_w = 0.1\ in$ and $d_w = 0.15\ in$, respectively. The comparison between Fig. 4.27-a and Fig. 4.27-b shows that the permeability results at a constant distance of $d_w = 0.1\ in$ from the wall would be very similar to those referred to the tests performed at the correct distances. In fact, the slope of the linear trend lines of the above mentioned figures, which is directly related to the permeability, differs of about 9%. On the other hand, tests conducted at a constant distance of $d_w = 0.15\ in$ from the wall (Fig. 4.27-c) clearly show a decrease in the correlation factor, $R$ ($R^2 = 0.947$ in Fig. 4.27-a compared to the $R^2 = 0.657$ in Fig. 4.27-c). Additional tests, performed at $d_w = 0.05\ in$ show completely uncorrelated results ($R^2 < 0$).

4.4.4 Effective Permeability Calculation

In this section, the results deriving from the data analysis of the hot-film probing at the prescribed distance from the wall ($d_w = 0.01\ in$) coupled to the pressure measurements are presented. Some of the original traces of the instantaneous velocities measured by the hot-film probe, for some representative locations, are reported in Fig. 4.28.

Figure 4.28-a shows, as we expected, a lower blowing capability of the region above the truncated graphite cone because a direct feeding of coolant from the high-pressure side of the C-C specimen has not been present. The blowing in this region has been mainly due to the radial feeding of coolant from the region above the internal plenum chamber (Fig. 4.28-b). Figure 4.28-c highlights the higher blowing capability of the SP region that is due to presence of the radial delaminations [4] which are extended along the entire specimen’s length and create a preferential direction of flowing for the coolant. The instantaneous velocity measurements in Fig. 4.28 have
been obtained by applying the calibration curve to the output voltages of the probe and by subtracting the average background velocity in the room \( U_B \approx 0.004 \text{ m/s} \) to each measurement. The graphs of the pressure gradient along with the Darcy’s velocity are presented in Fig. 4.29 for three longitudinal sectioning planes and the five air flow-rates reported in Table 4.1. A linear curve fit using the least squares method has been applied to reproduce Darcy’s law.

The linear-trend lines in Fig. 4.29 have been imposed to intercept the origin of the charts in order to reproduce the no-blowing condition. The coefficient of determination is, for all the linear curve fits, \( R^2 > 0.89 \) which indicates the good correlation between the data gathered for different air flow-rates at different control points. The linear fits highlight that the permeability, for this specimen, can be considered constant for the range of air flow-rates used. The linear variation of the pressure drop with the fluids velocity measured at the low-pressure side could be due to either, or both, the specimen geometry (thin-wall structure) and/or the morphology of the internal voids’ network. The lower correlation coefficient is obtained for the stagnation point probed perpendicularly to the \( 90^\circ - 270^\circ \) plane cut (\( SP - Horizontal \) in Fig. 4.9). The wider error bars on the velocity measurements for \( CS4 \) and \( CS5 \)
Figure 4.29. Darcy’s law trendlines for the selected CP surveyed (referred to Fig. 4.8-b).
are due to the high fluctuations of the signal from the hot-film probe. It could be generated, for example, by either or both the presence of oblique jets interacting at different distances from the wall or/and the transition from laminar to turbulent jet. The latter phenomenon, which generates a substantial decreasing of the coalescence distance for all the flow rates used [70], can be produced by the presence of adjacent and bigger channels in comparison with the average network-mesh size and the average throats diameter, respectively. The effective permeability has been directly calculated from the Darcy’s law (§4.2) by dividing the dynamic viscosity of air, which has been calculated by using the Sutherland’s law at the recorded temperature, by the slope of each straight line in Fig. 4.29. The corresponding map of the effective permeability is reported in Table 4.7.

<table>
<thead>
<tr>
<th>Location</th>
<th>Longitudinal cut planes</th>
<th>90°</th>
<th>180°</th>
<th>270°</th>
</tr>
</thead>
<tbody>
<tr>
<td>CP1</td>
<td>34.65 ± 0.63</td>
<td>12.53 ± 0.98</td>
<td>15.04 ± 0.65</td>
<td></td>
</tr>
<tr>
<td>CP2</td>
<td>12.93 ± 1.04</td>
<td>42.12 ± 0.70</td>
<td>44.69 ± 0.75</td>
<td></td>
</tr>
<tr>
<td>CP3</td>
<td>10.49 ± 0.33</td>
<td>31.26 ± 0.78</td>
<td>28.52 ± 0.90</td>
<td></td>
</tr>
<tr>
<td>CP4</td>
<td>10.28 ± 0.36</td>
<td>12.65 ± 0.41</td>
<td>10.54 ± 0.21</td>
<td></td>
</tr>
<tr>
<td>CP5</td>
<td>27.47 ± 0.73</td>
<td>25.19 ± 0.71</td>
<td>32.03 ± 1.07</td>
<td></td>
</tr>
<tr>
<td>CP6</td>
<td>26.09 ± 0.81</td>
<td>30.58 ± 0.78</td>
<td>40.29 ± 1.16</td>
<td></td>
</tr>
<tr>
<td>CP7</td>
<td>129.02 ± 5.11</td>
<td>10.29 ± 0.64</td>
<td>29.39 ± 0.89</td>
<td></td>
</tr>
<tr>
<td>CP8</td>
<td>64.94 ± 1.81</td>
<td>12.20 ± 0.52</td>
<td>21.79 ± 0.78</td>
<td></td>
</tr>
<tr>
<td>CP9</td>
<td>34.69 ± 1.13</td>
<td>9.39 ± 0.74</td>
<td>15.39 ± 1.03</td>
<td></td>
</tr>
<tr>
<td>CP10</td>
<td>29.72 ± 0.93</td>
<td>12.32 ± 0.56</td>
<td>19.59 ± 0.77</td>
<td></td>
</tr>
<tr>
<td>SP – Horizontal</td>
<td>151.63 ± 32.4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SP – Vertical</td>
<td>412.58 ± 31.8</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The nominal values of the effective permeability have been expressed in millidarcy \((1D = 1.013 \cdot 10^{12} \text{ m}^2)\) in order to obtain small reference numbers, as it is usually used for the analysis of porous structures in geology and petroleum appli-
The average uncertainties on the effective permeability ($\delta K_{eff}^r$) have been calculated by using Eq. 4.11 and neglecting the errors on the calculation of the dynamic viscosity ($\delta \mu$).

$$\delta K_{eff}^r = \left| \frac{\partial K_{eff}^r}{\partial a} \delta a \right| + \left| \frac{\partial K_{eff}^r}{\partial \mu} \delta \mu \right| \approx \frac{\mu}{a^2} \delta a$$  \hspace{1cm} (4.11)

Where $a$ is the slope of the linear curve fits in Fig. 4.28. The uncertainties associated to the slope of the straight lines ($\delta a$) have been directly assessed from the least squares method. The maximum percent deviations on the effective permeability are reported in Table 4.8. In specific, the positive and negative percent standard deviations on the effective permeability have been calculated by changing the nominal values of the Darcy’s velocity in relation to the error bars in Fig. 4.29. The range of variation of each point has been limited to those combinations generating a coefficient of determination higher than $R^2 = 0.8$ in order to maintain the Darcy’s law meaningful. The last assumption limited the maximum standard deviation for the effective permeability (Table 4.8) and, thus, confined the impact of the wide error bars characteristic of CP4 and CP5 (Fig. 4.29).

The upper bounds of the standard deviation ($\sigma_{K-max}^+$) are referred to the lower slope of the linear curve fits in Fig. 4.29 (higher effective permeability) while the values of $\sigma_{K-max}^-$ are referred to the higher slope of the linear curve fits (lower effective permeability). Only the variation of the Darcy’s velocity across its error bar has been used to determine the lower and upper bounds of the effective permeability values because of the negligible uncertainties on the pressure gradient, $\sigma_{max}(dP/dr) < 2\%$, with respect to those affecting the velocity measurements, $6\% < \sigma(U_D) < 50\%$. In this perspective, meaningful velocity measurements on the control surface along with the correct distance of the hot-film probe from the wall have the higher im-
Table 4.8. Maximum percent standard deviation on the effective permeability

<table>
<thead>
<tr>
<th>Location</th>
<th>( K_{r_{eff}}^c ) (%)</th>
<th>( K_{r_{eff}}^r ) (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>Longitudinal cut planes</td>
<td>( 90^\circ )</td>
</tr>
<tr>
<td>CP1</td>
<td>-6.47</td>
<td>-21.27</td>
</tr>
<tr>
<td>CP2</td>
<td>-18.63</td>
<td>-13.91</td>
</tr>
<tr>
<td>CP3</td>
<td>-19.35</td>
<td>-10.01</td>
</tr>
<tr>
<td>CP4</td>
<td>-15.30</td>
<td>-17.73</td>
</tr>
<tr>
<td>CP5</td>
<td>-20.90</td>
<td>-23.48</td>
</tr>
<tr>
<td>CP6</td>
<td>-8.54</td>
<td>-7.00</td>
</tr>
<tr>
<td>CP7</td>
<td>-4.32</td>
<td>-9.62</td>
</tr>
<tr>
<td>CP8</td>
<td>-5.91</td>
<td>-24.53</td>
</tr>
<tr>
<td>CP10</td>
<td>-10.80</td>
<td>-20.69</td>
</tr>
<tr>
<td>SP - Horizontal</td>
<td>-2.46</td>
<td></td>
</tr>
<tr>
<td>SP - Vertical</td>
<td>-16.3</td>
<td></td>
</tr>
</tbody>
</table>

Impact on the nominal values of the effective permeability. The results presented in Fig. 4.29 and in Table 4.7 allow defining the prototype specimen as a semi-pervious structure accordingly to the classification based on the permeability ranges used for soils [39]. The comparison of the nominal values of the effective permeability in Table 4.7 highlights the asymmetric blowing capability of the C-C cone due to the conformation of the voids’ network. The percent variation of the effective permeability between each plane, by considering the same control points, ranges between 6% (comparison of \( K_{r_{eff}}(CP2 - 270^\circ) \) with \( K_{r_{eff}}^r(CP2 - 180^\circ) \)) and 172% (comparison of \( K_{r_{eff}}(SP - Horizontal) \) with \( K_{r_{eff}}^r(SP - Vertical) \)). The higher permeability values are located at the stagnation point because of the two radial delaminations that create a preferential passage for the coolant fluid when a pressure gradient is applied across the specimen’s wall. The aforementioned feature opens the possibility of voluntarily introduce intrinsic defects at the fabrication level with the purpose of generating an effective permeability diversified accordingly to the cooling requirements of the TPS. The lower effective-permeability values are not detected only for
the control points above the supporting truncated cone (\(CP1, CP2, CP3\)) but they are distributed all over the surface. It is because the coolant fluid flows longitudinally towards the CPs above the supporting cone. The comparison of the effective permeability values for the straight-wall regions having different thicknesses (comparison of the effective permeability for \(CP1\) to \(CP8\) with that related to \(CP9–CP10\)) highlights that, for this case study, the permeability is geometrically independent (the pressure gradient is nearly constant with respect to the coolant flow-rate regardless of the thickness and the effective-permeability values are nearly similar).

4.5 Conclusions

The material’s characterization in terms of local effective permeability for a prototype carbon-carbon structure with respect to air has been experimentally determined in this work by using a novel technique based on hot-film anemometry coupled to pressure measurements. The effective permeability, conceived as the local blowing capability of a porous structure with respect to a selected coolant fluid, has been also discussed. Specifically, the coolant (air) blown from a conical porous surface has been measured by a hot-film anemometer at a distance specified by an appropriate reference elementary area and the Reynolds number based on the channels’ diameter. Five air flow-rates have been used to probe the blowing capability of three planes cut longitudinal to the symmetry axis of the cone. Eleven control surfaces placed in relevant locations, and distributed in order to provide a good spatial resolution of the measurements, have been surveyed for each longitudinal plane. These measurements have been then related to the pressure gradient across the local material’s thickness by using Darcy’s law. A parallel work, based on the CT-scan of the C/C mask, has been aimed to define the fundamental guidelines necessary to correctly perform the permeability tests based on hot-film measurements. In par-
ticular, the calculation of the porosity by analyzing the 2-D images from the X-ray CT machine has been used to define the reference elementary surface that is able to provide a near constant mean-velocity field and therefore to assess the minimum dimensions of the area to be surveyed with the hot-film. An independent investigation based on the 3-D reconstruction of the specimen has been conducted by the AMSL lab with the purpose of comparing the predictions in terms of porosity and reference elementary surface, which for cubic volumes is intended as reference elementary volume. Additionally, the influence of the probe distance from the wall has been analyzed by applying the theory of fluid flow through perforated plates/screens that allowed identifying the range of optimal distances for the mass-flux measurements. The main results obtained in this chapter are summarized below.

- The statistical analysis of the number and dimensions of the void elements inside the porous lattice has been conducted to calculate the optimum distance of the sensing film from the wall by applying the theory of fluid flow through perforated plates. Preliminary mass-flux measurements, performed at different distances from the wall and for different air-flow rates, confirmed the predictions of the correct hot-film distance from the wall that have been obtained theoretically.

- The resulting linear curve-fit, characteristic of the Darcy’s law, correlated fairly well the results obtained by imposing five air flow-rates used to probe the blowing capability of three planes cut longitudinal to the symmetry axis of the cone.

- The carbon-carbon TPS structure can be classified as a semipervious structures based on the range of the local effective-permeability values.

- The local effective-permeability map (Table 4.7) shows a pronounced asymmetric blowing capability of the cone because, for the same control points,
the $K_{\text{eff}}$ ranges between 6% and 172%. This result emphasizes the necessity of defining a methodology for the characterization of full-scale components in terms of local effective-permeability in order to detect asymmetric flow-paths, due to the internal morphology of the porous network, that can create concentrated mechanical loads and hot-spots on the structure. This characteristic opens the promising scenario of investigating the possibility to voluntarily introduce intrinsic defects, at the fabrication level, able to generate an effective permeability diversified accordingly to the cooling requirements of the TPS.

- The local effective-permeability, for the prototype specimen characterized, can be considered geometrically independent (compare the results of $CP9 - CP10$ with those of $CP1 - CP8$ in Table 4.7).

- The effective local blowing capability of a complex porous thermal protection system allows providing also the correct boundary conditions for numerical aerothermal reconstructions.

- The statistical analysis of the internal voids’ lattice can be used in support of numerical modeling for the thermomechanical response of porous structures by defining an equivalent porous network composed of a discrete number of voids elements (equivalent medium theory).
CHAPTER 5

Intrusive Flow Characterization of the Plasma Wind Tunnel Facility at UTA by using Teflon® Probes in Combination with Total Pressure Measurements

This chapter has been extracted from the journal paper: S. Gulli, C. Ground, M. Crisanti and L. Maddalena, “Teflon probing for the flow characterization of arc-heated wind tunnel facilities”, 2014, Experiments in Fluids [14] and from the paper: S. Gulli, C. Ground, M. Crisanti, L. Maddalena, ”Teflon Probing for the Flow Characterization of the 1.6 MW Arc-Heated Wind Tunnel of the University of Texas at Arlington”, 2013, presented at the 51st AIAA Aerospace Sciences Meeting [82].

The introductory part of this chapter (§5.1) highlights the crucial role of being able characterizing the high-enthalpy flow produced by existing ground testing facilities and the reason why Teflon® probes have been selected for the flow characterization of the 1.6 MW arc-heated wind tunnel (AHWT) facility at UTA. The second and third sections of this chapter (§5.2, §5.3) have been entirely dedicated to the ablative properties of Teflon® and to the theoretical analysis performed in this thesis work to improve the existing correlations for the blockage factor. The correct calculation of the blockage factor is fundamental in order to enable the use of ablative probes for the flow characterization of high-enthalpy, extended-test-time facilities. The experiment design and the facility description are reported in §5.4 and §5.5, respectively, while the main experimental results obtained from both the ablation tests and total pressure measurements are reported in §5.6. The data analysis and comparison of the results with those obtained by using different approaches are summarized in §5.7.
5.1 Introduction

From the beginning of space exploration, protecting the surfaces of the vehicle from the high-enthalpy flow encountered during atmospheric reentry has been a primary concern. The testing of thermal protection systems (TPS) plays an essential role in understanding the thermal response of candidate TPS materials. The Mach number, flight duration, altitude, Reynolds number, surface temperature, and gas-surface interaction effects are some of the desired parameters to be matched in order to reproduce the aerothermal environment encountered in real flight conditions. However, existing ground testing facilities are not able to simultaneously reproduce all the aforementioned parameters; therefore test conditions must be expertly selected in order to replicate some of those parameters with the purpose of studying the physics of interest. Moreover, arc-heated facilities introduce flow disturbances and contaminations (e.g. non-uniformities arising from the arc stabilization techniques and debris due to the erosion of the parts exposed to the hot flow within the heater) unlike the free-stream flow in real flight conditions. In this scenario, the accurate knowledge of the flow generated by these facilities is crucial for the development of high-enthalpy flow/TPS surface interaction models. The characterization of the high-enthalpy flow downstream the nozzle exit of the 1.6 MW arc-heated wind tunnel (AHWT) of the University of Texas at Arlington (UTA) is analyzed in this study in order to augment the current capabilities for the investigation and qualification of candidate TPS materials. Ablative Teflon® probes are used in combination with total pressure measurements to calculate both the wall heat transfer and, thus, the stagnation enthalpy with the added capability of detecting flow non-uniformities by inspection of the ablation pattern on the surface. The use of ablative specimens also enable the possibility of probing the entire plane surveyed, contrarily to other intrusive probes such as null-point calorimeters, slug calorimeters etc., which are able
to detect the heat flux variation only across their swinging trajectory. Teflon® has been selected for this investigation because it is a low temperature ablator and it is characterized by a low thermal conductivity. These properties allow the ablation response under high thermal loads to be considered, approximately, steady-state and characterized by one-dimensional heat transfer [83, 84]. Additionally, its low sublimation temperature allows relevant recession of the surface to be measured in high enthalpy flows [85, 86, 87]. For the aforementioned reasons NASA proposed and oversaw an extensive experimental campaign, the Round-Robin program, with the purpose of determining and correlating the most important variables (i.e. stagnation heat flux and total pressure) that determine the ablation rate of selected materials. Low-temperature and high-temperature ablators (Teflon® and Phenolic-Nylon materials, respectively) were tested in nitrogen and air flows. Several arc-jet facilities were selected and provided with the same instruments and test samples, provided by the Standford Reasearch Institute (SRI), in order to conduct an independent study of the ablation mechanism preventing systematic uncertainties on the measurements [86, 88]. The stagnation cold-wall heat flux, measured by using standard transient slug calorimeters, was coupled to total pressure measurements obtained using uncooled Pitot probes of the same geometry of the calorimeters. Several ablation tests on Teflon® specimens, having the same geometry of both the calorimeter and Pitot probes, were performed for the same test conditions with the aim to correlate the Teflon® ablation rate to both heat flux and total pressure measurements [83]. In this work, a correlation selected from those developed by NASA has been used at the design stage of the experiment to define the geometry of the ablative probes and the tests duration. The same correlation has also been used a posteriori to calculate the cold-wall stagnation heat flux ($q_{0,cw}$) by using both Pitot and mass loss rate measurements. The cold-wall stagnation point heat flux, $q_{0,cw}$, represents the
heat transfer that could have been measured if a slug calorimeter of the same geometry of both Teflon® and Pitot probes was used. In addition, a simplified analysis leveraging the control volume (CV) technique has been used to calculate the average stagnation heat flux (q_{abl}) including the effect of the heat blockage due to the mass injection inside the boundary layer. The dimensional analysis of the ablation process has been utilized to quantify the blockage effect and, thus, to calculate the hot-wall heat flux without mass injection (q_{0,hw}), which is the heat flux measured by a non-ablating probe when the surface temperature reaches the temperature of ablation. A substantial improvement of the existing correlations for the heat blockage has been obtained by introducing the stagnation pressure dependence inside the non-dimensional groups of the dimensional analysis. The range of validity of the correlation proposed for the blockage factor has been extended to lower values of the blowing parameter once the data obtained in this study have been included. The wall heat transfer calculations deriving from both the CV analysis using the new correlation and the NASA correlation have been used in conjunction with a simplified stagnation-point convective-heating equation [89] to determine the stagnation enthalpy of the flow for different points of the facility’s performance map. The outcomes from the ablation tests have also been compared with a numerical code developed by the authors, which calculates the flow properties from the plenum chamber of the arc heater up to the specimen surface by using a 1-D fluid-dynamics analysis. Different locations downstream of the nozzle exit have been surveyed with the aim to characterize the centerline of the plume and to estimate the heat flux enhancement factor due to shockwaves interactions. The results obtained highlight the potential of using ablative probes for the flow characterization including the possibility to detect non-uniformities of the flow. In addition, the results of the ablation tests enable creating valuable databases that can support the validation of numerical
models used to simulate the ablation mechanisms and, as it is demonstrated, allow improving existing semi-empirical correlations for the blockage effect.

5.2 Theoretical Background

In this section, the physics of the ablation mechanism for Teflon® is presented along with the relations needed to analyze and interpret the results obtained in this work and in previous experimental campaigns (Round-Robin program).

5.2.1 Ablative Properties of Teflon®

Ablative cooling has been widely proven in literature to be one of the most reliable techniques to protect aerospace vehicles from the aerodynamic heating characteristic of re-entry conditions, when limited mission times are considered [90, 91]. Teflon® has been extensively investigated in the past because of its peculiar characteristics, that are generally desirable for ablative materials [92]. In fact, its plastic nature allows depolymerization reactions that directly produce a gaseous monomer without having liquefaction that could cause variation of the specimen’s shape [84]. Additionally, the thermal insulation property of Teflon® [93, 94] confines the ablation process to the surface without having any local ablation inside the material (volumetric ablation), which is characteristic of ablative thermal shields made of carbon composite materials. The ablative mechanism for Teflon® is composed of two stages: a transient ablation, of the duration of few seconds, followed by a quasi-steady ablation [93, 94, 95, 96, 97, 98]. During transient ablation, Teflon® changes its color, from translucent white to transparent, and its molecular structure, from linear polymeric chains to amorphous structure, by absorbing a certain amount of heat. This transition is characterized by the formation of a high-viscosity gel layer and occurs when the Teflon® reaches a temperature of $T \approx 600 \text{ K}$. When the tempera-
ture further increases, pyrolysis occurs and the polymer decomposes by absorbing an additional quantity of energy, the depolymerization energy \( (H_p) \). At this stage the quasi-steady ablation takes place \[99\]. The temperature at which the decomposition process occurs is strictly related to the local vapor pressure of the Teflon\textsuperscript{®} \[100, 101\]. The equilibrium vapor pressure of heated Teflon\textsuperscript{®} \( (p_0) \) follows the relation in Eq. 5.1 which is a curve fit derived from experimental data by Wentink \[102\].

\[
p_0 = p_C \cdot \exp(-T_C/T_0)
\]  

(5.1)

Where the characteristic temperature and pressure of Teflon\textsuperscript{®} are, respectively, \( T_C = 20815 \) \( K \) and \( p_C = 1.84 \cdot 10^{20} \) \( Pa \) while \( T_0 \) is the temperature at the ablat- ing surface. During the decomposition process, it is possible for some molecules of polymer to diffuse through the surface and, thus, escape before being completely depolymerized when the reactions happen in non-equilibrium conditions \[96, 103\]. This effect can be quantified by including a temperature dependence, as suggested by Arai \[97\] (Eq. 5.2).

\[
H_p = H_{p0} - (H_{p1} \cdot T) = 1.77 - (0.279 \cdot 10^{-3}) \cdot T
\]  

(5.2)

The vapor pressure of Teflon\textsuperscript{®} can be assumed to be equal to the stagnation pressure downstream the normal shock in front of the model \[83\]. The surface temperature of Teflon\textsuperscript{®} and the energy of depolymerization are calculated by using Eq. 5.1 and Eq. 5.2, respectively, once the stagnation pressure is known. The nominal value of the heat of depolimerization is substantially constant for all the test conditions used for both the Round-Robin program and the experimental campaign presented \( (H_p = 1.54 \, MJ/kg \pm 2\%) \). In the complete sublimation process of Teflon\textsuperscript{®}, as previously mentioned, the heat of depolimerization is only a portion of the heat
necessary to vaporize the material. In fact, the heat necessary to rise the surface temperature up to the ablation temperature \((H'_p)\) is added to the latent heat of decomposition in order to take into account for the transient phase of the ablation mechanism (Eq. 5.3 by Hiester and Clark, [95] and Chapman, [85]).

\[
H_A = H_p + H'_p = 2.19 \text{ MJ/kg} \tag{5.3}
\]

Where \(H_A\) is named heat of ablation. The sublimation process at the surface produces a large amount of gas which thickens the boundary layer profile reducing the wall heat transfer [93, 104]. All the aforementioned characteristics led NASA to establish, in the early sixties, the Round-Robin program with the objective of providing a standard method to characterize the ablation properties of low-temperature (Teflon\textsuperscript{®}) and high-temperature (Phenolic-nylon) ablators [83, 85, 86, 87]. One of the main outcomes of the test program was a series of correlations for the ablation rate of the materials characterized. In this study, the correlation in Eq. 5.4 has been selected.

\[
\dot{m}_T = 0.0058 \cdot (q_{0,cw})^{0.58} \cdot p_{sp}^{0.25} \tag{5.4}
\]

Where \(\dot{m}_T\) is named ablation rate or mass loss rate, \(q_{0,cw}\) is the cold-wall heat transfer of a non-ablating surface and \(p_{sp}\) is the stagnation-point pressure. The coefficients and the exponents of the correlation in Eq. 5.4 have been selected based on the results obtained at facilities having similar operational conditions of the AHWT at the University of Texas at Arlington and including those facilities that have used the same standard slug calorimeters provided by the Stanford Research Institute [83]. The representative parameters considered for the selection have been the total enthalpy and the stagnation pressure, which have been demonstrated in §5.2.4 to be
the most important variables determining the recession rate of Teflon® \(\dot{m}_T\) [88]. The left-hand side of Eq. 5.4 is calculated once the Teflon® density \(\rho_T\), the test time \(t_{\text{run}}\), and the recession of the stagnation point \((L_{R,sp})\) are measured (Eq. 5.5).

\[
\dot{m}_T = \frac{\rho_T \cdot L_{R,sp}}{t_{\text{run}}}
\] (5.5)

The average Teflon® density in Eq. 5.5 \(\approx 135 \text{ lb/ft}^3\) has been retrieved from mass and volume measurements of the five samples used for the test campaign.

5.2.2 Control Volume Analysis

A control volume analysis of the receding surface has been used to calculate the convective heat transfer at the stagnation point and, thus, to infer the stagnation enthalpy by using both the mass-loss rates and the Pitot pressure measured experimentally. The moving control volume on a portion of the high-viscous layer at the Teflon’s tip surface near the stagnation point has been chosen (Fig. 5.1). The control volume selected is rigid and it moves with a constant velocity \(L\) equal to the recession rate of the surface. The control surface \(CS_3\) is considered almost coincident with the specimen’s flat surface \((x = 0\) in Fig. 5.1) in order to neglect the side fluxes at \(CS_2\) and \(CS_4\). The width of the CV \(L\) in Fig. 5.1) is considered of the same dimension of the region covered by a micrometer pointer during several measurements of the stagnation-point recession \((L/D = o(0.1))\) and the depth of the CV \(H\) is of the order of the gel-layer thickness \((H/D = o(0.1))\).

The steady-state energy balance at the surface is considered in this analysis (Eq. 5.6).

\[
q_{\text{conv}} + q_{\text{rad}} + q_{\text{comb}} = q_{\text{cond}} + q_{\text{abl}}
\] (5.6)
The characteristic non-catalytic properties of Teflon® allows neglecting the combustion effects at the surface \( (q_{comb} \approx 0) \) [96]. The heat conduction \( (q_{cond}) \) across the control surfaces \( CS_2 \) and \( CS_4 \) can be also neglected due to the uniformity of the surface temperature [93, 105], if the sample is completely surrounded by the plume. The net radiation due to the boundary layer \( (q_{rad}) \) has also been neglected because it does not affect the recession rate in the quasi-steady phase of the ablation mechanism. In fact, the only effect the radiation has is increasing the thickness of the gel layer which, in turn, decreases the temperature gradient at \( CS_1 \) [95]. The temperature in the gel layer can be assumed to decrease linearly from the sublimation temperature, at the exposed surface \( (T_S \approx 900 \, K) \), to the transition temperature \( (T_T \approx 600 \, K) \), at the interface between the amorphous and the crystalline region of the specimen [93, 95, 97, 105]. The amount of heat transferred by conduction to the back face of the specimen can be estimated by assuming a reasonable thickness of the gel layer and calculating the average value of the thermal conductivity of Teflon® at the amorphous stage \( (k_g \text{ in Eq. 5.7 by Holzknecht, [93]}) \).

\[
k_g = (21.04 - 3.34 \cdot 10^{-2} \cdot T + 1.39 \cdot 10^{-5} \cdot T^2) \cdot 10^{-4}
\]  

\(5.7\)
The thermal conductivity corresponding to the average temperatures expected ranges from $k_g \approx 0.03 \text{ W/(m K)}$ to $k_g \approx 0.09 \text{ W/(m K)}$. An order of magnitude analysis reveals the possibility of neglecting the contribution of the conductive term at the control surface $CS_1$. The temperature difference and the thickness of the gel layer are respectively of the order of $o(10^2 \text{ K})$ and $o(10^{-3} \text{ m})$. Therefore, the temperature gradient is of the order $o(10^5 \text{ K/m})$ that multiplied by the average thermal conductivity $o(10^{-2} \text{ W/(m K)})$ gives a contribution of about $10^3 \text{ W/m}^2$ compared to the overall heat transfer, which is of the order of $MW/m^2$. The convective stagnation heat flux at the receding surface ($q_{conv}$) is, therefore, balanced by the ablation heat flux ($q_{abl}$) at the same location once the conductive heat transferred at the surface between the crystalline and amorphous layers, the radiation and combustion heat fluxes are neglected (Eq. 5.8 by Pope [96]).

$$q_{conv} = q_{abl} = \dot{m}_T H_A \quad (5.8)$$

The mass loss rates measured from the ablation tests allow the heat leaving the surface, due to the ablation mechanism, to be quickly determined once the heat of ablation ($H_A$) is known. The convective heat flux in Eq. 5.8 and Fig. 5.1 is the net heat transfer to the ablating surface. It is lower with respect to that one measured by using a calorimeter that has the surface temperature equal to the ablation temperature because of the heat blockage ($\psi$ parameter in Eq. 5.9) due to the gas injection inside the boundary layer ($q_{conv} = \psi \cdot q_{0,hw}$). In this perspective, the stagnation enthalpy ($h_{sp}$) can be calculated using the simplified version of the stagnation point convective-heating equation (§5.2.3) once the blockage effect is quantified (Eq. 5.9).
\[ q_{0,hw} = \frac{q_{abl}}{\psi} \]  

(5.9)

Where \( q_{0,hw} \) represents the cold wall heat transfer or, specifically, the heat transfer to a non-ablating surface (0 subscript) calculated at the ablating temperature \((hw\) subscript).

5.2.3 Simplified Version of the Stagnation Point Convective-Heating Equation

A general equation for the stagnation-point convective heat transfer to an axisymmetrical blunt-body for selected gas compositions was derived by Zoby [106] assuming non-catalytic wall. Then, Sutton and Graves [89] derived a simplified version of this equation for high-enthalpy flow and arbitrary gas mixtures in chemical equilibrium once the heat transfer coefficient \( K \) was introduced (Eq. 5.10).

\[
K = \frac{q_{0,w}}{(h_{sp} - h_w)} \cdot \sqrt{\frac{R_{eff}}{p_{sp}}} \]  

(5.10)

Where \( R_{eff} \) in Eq. 5.10 is the effective radius of curvature of the Teflon® sample [107, 108, 109]. The effective radius is based on experimental results for the stagnation point velocity gradients over blunt axisymmetric bodies conducted by Zoby and Sullivan [110]. For the selected geometry \( R_{eff} = 3.45R_{tip} \). The radius at the tip of the specimen \( R_{tip} \) is the average radius calculated before and after the tests. They are slightly different because of the conical shape of the samples and because the pressure of the flow acting on the highly-viscous Teflon® barely shrinks the material during the tests. The heat transfer coefficient \( K \) has the expression in Eq. 5.11.

\[
K = \frac{0.0885}{(Pr_w)^{0.6}} \cdot \left( \sum \frac{c_{o,i}}{M_{o,i} \cdot \gamma_{o,i}} \right)^{-1/2} \]  

(5.11)
The terms included in the summation are calculated using the molecular theory for gases and liquids [111]. The wall Prandtl number \((Pr_w)\) has been assumed equal to 0.69 and can be considered constant for all the base gases and gas mixtures of interest for planetary reentry [89]. The simplified expression of the heat transfer coefficient (Eq. 5.11) depends on the mass fraction \((c_{o,i})\), molecular weight \((M_{o,i})\) and the transport parameter \((\gamma_{o,i})\) of the initial composition of the free-stream flow. In this study, nitrogen has been used as primary working gas. The heat transfer coefficient referred to nitrogen is reported in Eq. 5.12 [89].

\[
K_{N_2} = 0.1112 \frac{kg}{(s \ m^{3/2} \ atm^{1/2})} \quad (5.12)
\]

The final simplified expression of the enthalpy potential \((\Delta h)\) is reported in Eq. 5.13.

\[
\Delta h_{sp} = h_{sp} - h_w = C \cdot q_{0,sp} \cdot \sqrt{R_{eff} \cdot (p_{sp})^{-0.5}} \quad (5.13)
\]

The gas mixture constant \(C\) in Eq. 5.13 is equal to \(1/K \approx 24 \ s \ ft^{1.5} \ atm^{0.5} \ lb^{-1}\) accordingly to the units of measure used for the NASA correlation. The comparisons between Eq. 5.13 and the numerical solution of the multi-component boundary layer governing equations for equilibrium flow have shown a good agreement in terms of heat transfer coefficient prediction [89]. The simplified heat transfer equation (Eq. 5.13) is, thus, used to calculate the stagnation enthalpy once the total pressure, the enthalpy at the wall, the specimen geometry, the initial gas composition and the wall heat transfer to a non-ablating surface are determined. The nitrogen cold-wall enthalpy, \(h_{cw} = 134 \ Btu/lb\) as suggested by Hiester and Clark [83], is considered when the cold-wall heat flux is calculated by using the NASA correlation (Eq. 5.4).
The hot-wall enthalpy \( h_{hw} = C_{P,N_2}T_{abl} \) is adopted when the hot-wall heat flux \( (q_{0, hw}) \) is calculated using the control volume analysis (Eq. 5.9).

### 5.2.4 The Blockage Factor and Dimensional Analysis of the Ablation Mechanism

In this section, the dimensional analysis of the ablation mechanism is approached with the purpose of evaluating the amount of heat blocked by the transpiration process. The experimental data deriving from the NASA Round-Robin program have been collected in order to identify possible correlations that allow collapsing most of the data into a single trendline. The non-dimensional parameter of interest for this analysis is the blockage factor \( (\psi) \) which is defined as the ratio of the heating rate with mass addition \( (q_{abl}) \) to the heating rate without mass addition \( (q_{0, hw}) \). The heat transfer rate with mass injection is calculated using Eq. 5.8. The heat transfer without mass injection is inferred by using the cold-wall heat flux, measured by the standard slug calorimeter, multiplied by the correction factor for the enthalpy potential (Eq. 5.14 from Pope 1975).

\[
q_{0, hw} = q_{0, cw} \cdot \frac{(h_e - h_{hw})}{(h_e - h_{cw})} \quad (5.14)
\]

Where \( h_e \) is the enthalpy at the outer edge of the boundary layer. The cold-wall enthalpy and hot-wall enthalpy are referred to \( T_{cw} = 300 \) K and \( T_{hw} = T_{abl} \approx 900 \) K, respectively. Several correlations for the blockage factor were developed in previous studies starting from the database of the Round-Robin program. In particular, Adams [112] and Schmidt [113] proposed a linear correlation for the blockage of Teflon\textsuperscript{®} \( (\psi_{lin} \) in Eq. 5.15).

\[
\psi_{lin} = 1 - \beta \cdot \frac{\dot{m}_T (h_e - h_{hw})}{q_{0, hw}} \quad (5.15)
\]
Where $\beta$ is the transpiration factor that, for Teflon® ablation and laminar flow, can be approximated by using Eq. 5.16 [96, 114].

$$\beta = 0.72 \left( \frac{M_{\infty}}{M_\nu} \right)^{0.4} \quad (5.16)$$

Where $M_{\infty}$ and $M_\nu$ are the molecular weights of the freestream flow and ablation products, respectively. A second-order relationship for the convective blockage was proposed by Marvin and Pope (Eq. 5.17) [115].

$$\psi_{quad} = 1 - \beta \cdot \frac{\dot{m}_T(h_e - h_{hw})}{q_{0,hw}} + \beta_2 \cdot \left[ \frac{\dot{m}_T(h_e - h_{hw})}{q_{0,hw}} \right]^2 \quad (5.17)$$

$\beta_2$ can be approximated using Eq. 5.18 [96]:

$$\beta_2 = 0.13 \left( \frac{M_{\infty}}{M_\nu} \right)^{0.8} \quad (5.18)$$

An improvement of the previous two correlations was presented by [116] by using an exponential relationship (Eq. 5.19).

$$\psi_{exp} = 0.85 \cdot e^{-4/3B} + 0.15 \quad (5.19)$$

Where $B$ is the blowing factor defined in Eq. 5.20.

$$B = \frac{\dot{m}_T(h_e - h_{hw})}{q_{0,hw}} \quad (5.20)$$

Figure 5.2 reports the three correlations from literature along with the experimental data collected from some of the selected facilities for the Round-Robin program [83]. In this study, the results from the NASA Manned Spacecraft Center have been excluded because they are referred to subsonic freestream flow. The data from Giannini Scientific Corporation, Martin Company and some of the tests ob-
Figure 5.2 highlights that the correlations available in literature are not able to collapse all the data into a single trendline with reasonable scattering. They fairly accurately predict the blowing effect for different regions in Fig. 5.2 but the analysis of the experimental data does not reveal any threshold of the test conditions that allows selecting one correlation rather than another. In general, the points on the top left in Fig. 5.2 represent high-pressure, low-enthalpy conditions while the data on the bottom right in the same figure represent the high-enthalpy, low-pressure test conditions. The best curve-fit for the complete set of data generates a coefficient of determination (R-squared) of $R^2 = 0.38$.

5.3 Proposed New Correlations for the Blockage Factor

The possibility of using ablative probes for the flow characterization is strictly related to the necessity of reducing the scattering of the experimental data obtained in arc-heated testing facilities using a wide range of test conditions. A dimensional analysis is, consequently, needed to find different non-dimensional groups, similar to
the blowing parameter \((B)\), and correlate them to the blockage factor. The first step of the dimensional analysis [117, 118] identifies the problem variables that define the ablation of Teflon® (Eq. 5.21). The number of these variables is named \(m\).

\[
\phi (\Delta h_{hw}, q_{abl}, q_{0, hw}, p_{tot}, R_{eff}, \dot{m}_T, C') = 0
\]  

(5.21)

In the second step, all the fundamental variables involved in the physics of ablation have to be recognized (e.g. mass, length and time). The temperature is not listed within the fundamental variables since the surface temperature as well as the heat of ablation of Teflon® can be considered fairly constant with respect to the temperature [83]. The number of non-dimensional groups are defined by \(m-n\), where \(n\) is the number of fundamental variables. In the last step, the dimensionless groups are identified from the problem variables once a functional relationship is assumed and once the exponents of the functional relationship satisfy the dimensional homogeneity [118]. In particular, the selected variables in Eq. 5.21 allow creating 4 non-dimensional groups which are excessive for the purpose of correlating the blockage factor. The number of non-dimensional parameters can be reduced by substituting the ratio between \(q_{abl}\) and \(q_{0, hw}\) with the blockage factor. The blockage factor is a non-dimensional number and, therefore, it is not considered in the number of the problem variables [119, 120]. Additionally, the enthalpy potential in Eq. 5.21 can be removed because the simplified convective-heating equation in §5.2.3 suggests that total pressure, effective radius and heat transfer without mass injection are linked. In this study, the enthalpy dependence has been deliberately substituted in favor of the stagnation pressure in order to relate the low pressure region with the high-pressure region of Fig. 5.2. The updated functional relation selected for this study is reported in Eq. 5.22.
Figure 5.3. Exponential-law correlation for the blockage factor.

\[ \phi(\psi^2, p_{tot}, R_{eff}, \dot{m}_T, C) = 0 \]  
(5.22)

The functional relation connecting the selected variables is expressed in Eq. 5.23.

\[ \psi^2 \approx p_{tot}^a \cdot R_{eff}^b \cdot \dot{m}_T^c \cdot C^d \]  
(5.23)

The number of each basic unit (length, mass and time) must be the same on each side of the relation in order to respect the dimensional homogeneity [117]. The final expression for the blockage factor is given in Eq. 5.24 once the balance of the basic units is performed.

\[ \psi^2 = G \cdot \left( C \cdot \dot{m}_T \sqrt{\frac{R_{eff}}{p_{tot}}} \right)^d \]  
(5.24)

Where the constant \( G \) and \( d \) are extracted from the data fit of the experimental results. The term enclosed in the round brackets of Eq. 5.24 is the new blowing factor \( B_2 \). Figure 5.3 reports the results deriving from the new variables grouping correlated by using an exponential trendline.
The new blowing factor $B_2$, with respect to the one suggested in literature (Eq. 5.20), allows reducing consistently the dispersion of the data for the entire range of conditions provided by the arc-jet testing facilities. The R-squared increases from $R^2 = 0.38$ to $R^2 = 0.67$ corresponding to the curve fit in Fig. 5.3. An additional improvement of the correlation is obtained once the explicit dependence of the blockage effect to both stagnation pressure and enthalpy potential is considered (Eq. 5.25).

$$q_{o,hw} \approx p_{tot}^{a} \cdot \Delta h_{hw}^{b} \cdot \dot{m}_{T}^{c} \cdot q_{abl}^{d}$$  \hspace{1cm} (5.25)

The term inside the round brackets is the updated blowing factor ($B_3$). The outcome from the new non-dimensional group is reported in Fig. 5.4. The power-law correlation in Fig. 5.4 generates a $R^2 = 0.8$. The comparison of all the correlations presented in this section highlights the connection existing between all the results obtained during the Round-Robin program.

It suggests the possibility of collapsing all the data on a single trendline if some other effects, neglected in this study, are included in the dimensional analysis (i.e. radiative heat flux, shear stress, chemistry at the surface, free-stream Mach number etc.). The improved correlation in Fig. 5.4, which can be used for other investigation studies, cannot be directly used to estimate the blockage factor without the use of non-ablating heat flux probes because the hot-wall enthalpy potential included in the blowing factor ($B_3$) is not directly measured. In this scenario, the exponential-law correlation of Fig. 5.3 has been used for the flow characterization of the UTA’s
arc-jet facility. The uncertainty related to the correlation used, \( (\sigma_{\psi_2})^+ = 17\% \) and \( (\sigma_{\psi_2})^- = 16\% \), has been calculated accordingly to the procedure used for the Round-Robin program [83].

5.4 Experiment Design

In this section, the procedure used to design the experiment in terms of prediction of the Teflon\textsuperscript{®} ablation rates and selection of the test conditions is presented. The test conditions used for the experimental campaign are shown in Table 5.1. Two points on the facility’s performance map have been selected and different locations downstream the nozzle exit plane have been surveyed.

Table 5.1. Nominal test matrix used for the experimental campaign

<table>
<thead>
<tr>
<th>Test #</th>
<th>( \eta_{th} )</th>
<th>Distance from the nozzle exit, ( m )</th>
<th>Arc Power, ( kW )</th>
<th>( N_2 ) flow rate, ( kg/s )</th>
<th>Bulk total enthalpy, ( MJ/kg )</th>
<th>Test time, ( s )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.64</td>
<td>2.5</td>
<td>750</td>
<td>0.112</td>
<td>4.049</td>
<td>8</td>
</tr>
<tr>
<td>2</td>
<td>0.64</td>
<td>2.5</td>
<td>550</td>
<td>0.112</td>
<td>3.142</td>
<td>16</td>
</tr>
<tr>
<td>3</td>
<td>0.64</td>
<td>1.5</td>
<td>550</td>
<td>0.112</td>
<td>3.142</td>
<td>8</td>
</tr>
<tr>
<td>4</td>
<td>0.64</td>
<td>3.5</td>
<td>550</td>
<td>0.112</td>
<td>3.142</td>
<td>8</td>
</tr>
</tbody>
</table>
The nominal test conditions reported in Table 5.1 have been used for both Teflon® and Pitot probing for a total of eight separate tests. The bulk total enthalpy has been calculated using the energy balance method (Eq. 5.27).

\[ h_B = \eta_{th} \cdot \frac{P}{\dot{m}_{N_2}} \]  

(5.27)

The electric power \((P)\) is calculated by multiplying the voltage applied across the electrodes by the variable current. The efficiency term, \(\eta_{th}\), is calculated by the product of the cooling water flow multiplied by its heat capacity and the temperature difference measured across the inlet and outlet of the manifolds. It represents the amount of energy removed to the high-enthalpy flow by the water system that actively cools the internal arc-heater components. The average value of the thermal efficiency, based on previous results, has been considered constant for the entire experimental campaign \((\eta_{th} = 0.64)\). The test duration has been imposed accordingly to the predictions of the mass loss rates of the sample at the stagnation point (Eq. 5.4) and considering that the minimum and maximum exposure times of the specimens to the high-enthalpy flow are related, respectively, to following required conditions:

1. The regression must be measurable.
2. The exposure time of the Pitot tube must be comparable with that of the Teflon® sample. However, the un-cooled Pitot steel tube cannot be maintained within the high enthalpy flow for longer than several seconds if failure of the steel tubing has to be avoided.

For test #2, the test time has been doubled with the purpose of investigating its effect on the regression length. In particular, the comparison between the tests performed for different exposure times served to prove the steadiness of the ablation mechanism. At the design stage of the ablation experiment, the mass loss rate
of the Teflon® sample has been overestimated by assuming the entire surface to be exposed to both the peak stagnation heat flux and total pressure downstream the normal shock in front of the specimen. The total pressure and peak heat flux have been calculated by using a code developed by the authors, which integrates the Chemical Equilibrium with Applications (CEA) applet, from NASA, to perform a 1-D analysis of the chemical composition and thermodynamic properties of the flow from the plenum chamber of the arc heater to the specimen’s surface. The code has been successfully used in previous experimental campaigns with the main purpose to define the test conditions needed for targeting selected surface temperatures on carbon-carbon specimens. The equilibrium flow in the plenum chamber is computed using the enthalpy value derived from Eq. 5.27 and an iterative process, in which the plenum pressure is varied and CEA repeatedly called, is repeated until the calculated mass flow through the nozzle converges to the one measured by a calibrated mass-flow meter. The thermodynamic properties of the flow from the nozzle’s throat up to the bow shock in front of the specimen are calculated assuming frozen flow due to the short length of the nozzle [121]. A standard iterative method [16] has been used to retrieve the local flowfield conditions downstream the normal shock in thermodynamic equilibrium. The stagnation peak heat flux is calculated using the Fay-Riddell equation [16] along with the flow conditions at the wall calculated in CEA assuming a static temperature of 300 K. For these calculations, the Lewis number, Le, and Prandtl number, Pr, were assumed to have values of 1.4 and 0.7, respectively. Nitrogen dissociation was found to not contribute significantly to the convective heat flux with the power and flow settings utilized during this experimental campaign. The expansion of the plume at the nozzle exit is calculated at the end of the code by using the ratio between the nozzle exit pressure divided by the average test chamber pressure obtained during previous experimental campaigns. A representation of the
shocks (solid lines) and expansions (dashed lines) patterns for the cases considered in Table 5.1 is presented in Fig. 5.5. A separate set of runs targeting the conditions shown in Table 5.1 were required for the total pressure measurements.

5.5 Facility Description and Experimental Setup

The flow to be characterized is produced by a 1.6 MW Huels-type arc heater facility that resides in the Aerodynamics Research Center at the University of Texas at Arlington. The Thermal Dynamics F−5000 model arc heater is powered by a Halmar plasma arc torch system, and is capable of producing bulk enthalpies ranging from 3–8 $MJ/kg$ while operating steadily at mass-flow rates between 0.07–0.18 $kg/s$ for run times up to 200 s. The brass segment housing the nozzle is designed to allow
for the installation of interchangeable facility nozzles with Mach numbers ranging from 0.8 to 4.0. For the current investigation the facility’s nominal Mach 1.8 conical nozzle with an exit diameter of 1.0 in has been selected. The layout of the facility is reported in Fig. 5.6 [82].

A swinging system (Fig. 5.7-a) has been designed in order to introduce the sample in the plume after the steady state conditions are reached. A two-body probe has been developed to perform the Teflon® experiments (insulator + Teflon® specimen).
The same geometry of the test samples used in previous experimental investigations ($R_{\text{tip}} = 0.75\ in$) has been selected for the Teflon® probes in order to interpret the results previously obtained (Fig. 5.7-b). The graphite cone, built by C-CAT, served a dual purpose of both mechanically supporting and thermally isolating the Teflon® specimen. The support cone’s reusability gives way to the possibility of interchanging the Teflon® test samples with other ablative materials and also allows investigating the influence of the specimen’s geometry. Total pressure measurements were performed using an un-cooled steel Pitot tube (Fig. 5.8-a). A plastic tube of 0.125 in inner diameter was used to extend the Pitot outlet outside of the test chamber to remove the complications of integrating a pressure transducer within the swinging arm itself. The Pitot tube was shielded using a graphite insulator having the same geometry of that one used for Teflon® probing (Fig. 5.8-b).

A T-type thermocouple was mounted immediately behind the insulator cone shown in Fig. 5.8-b in order to monitor the temperature within the un-cooled apparatus and ensure that it never reaches dangerous levels (as might happen in the case of a leak being present within the pressure tube line). Figure 5.9 shows the starting
phase of the AHWT, which consists of a very short stage wherein argon is used to establish the arc, quickly followed by a transition to nitrogen and a manual ramp-up of the current until the desired value power setting is reached. The test sample was introduced in the hot flow using the swinging arm shown in Fig. 5.7-a only after the flow and power settings had stabilized to nominal values.

5.6 Results

In this section, the results of the ablation tests and total pressure measurements are provided.

5.6.1 Teflon® Probing

A typical test sample before and after the test is presented in Fig. 5.10. A considerable level of ablation over the sidewalls of the Teflon® sample is detectable. This occurred due to the complete immersion of the sample inside the plume, as predicted by the code mentioned in §5.4 (Fig. 5.5-a), leading to a smoothening of the lateral surfaces. Furthermore, the sample shows a clear annular concavity (Fig.
5.10-b). The resulting peculiar ablation pattern can be due to the impingement of reflecting shock waves on the surface of the sample, leading to an increased regression rate at the outer region of the test sample, to the flow non-uniformity arising from the swirling motion generated by the stabilization system or both.

An additional test, not originally part of the test matrix and, thus, not shown in Table 5.1, was performed in order to clarify the origin of the annular concavity. The new test (test #1 − A) was performed at the same position and for the same nominal conditions as test #1 seen in Table 5.1, but with a different test sample geometry (Fig. 5.11). A smaller tip radius ($R_{\text{tip}} = 0.4 \text{ in}$) was chosen in order to avoid impingement of the shocks on the flat face of the sample (Fig. 5.11-a). An alignment process using a laser system has been used for all the specimens in order to limit the misalignment between the sample and the centerline of the high-enthalpy flow during the run (Fig. 5.11-b). It served to limit the uncertainty on recession measurement at the stagnation point.

The length of the conical portion of the sample was selected considering that at the end of the run, for the estimated mass loss rate (§5.4), the desired effect was to maintain the shock impingement region on the sidewalls of the Teflon® probe. The average value of the expansion ratio of Fig. 5.11-a has been based on the experimental pressure measurements collected during test #1 (Fig. 5.12). The average expansion
Figure 5.11. Modified test sample. a) Simulation of the shock/expansion patterns; b) Alignment procedure.
ratio has been found to be in agreement with the prediction from the aforementioned numerical code.

The qualitative analysis of the Teflon’s surface after test #1 – A (Fig. 5.13) shows the evenness of the sample’s surface, indicating the uniformity of the flow generated by the arc heater within the shock-free region. Consequently, the protrusion detectable in the centerline of the sample of test #1 can be attributed to the shock impingement on the outer region of the specimen. The post-test analysis of the ablated surface of test #1 – A (Fig. 5.13) revealed a step on the sidewall, which was presumed to be due to the shock impingement.

The accurate position of the impinging region on the sidewalls cannot be calculated in a convenient way using geometrical considerations (Fig. 5.11-a), due to the presence of the bow shock in front of the sample. Additionally, the plume was bent due to the presence of the sample itself, resulting in the reflected shock impinging on the Teflon® sample downstream the position predicted analytically and shown in Fig. 5.11-a (solid lines). The mass loss rate measurements for each of the tested Teflon® samples in Fig. 5.13 have been performed for the same locations along the
inner and outer diameter of the tip surface by using a micrometer in combination
with a scheme designed to map the topology of the surface of the sample (Fig. 5.14,
Fig. 5.15).

The measurements of the surface depth were compared to the original dimen-
sions of the test samples (Fig. 5.15) [82].

Figure 5.16 reports the recession measured for the outer and inner regions
mapped. The average uncertainty on the recession measurements is \( \delta L_{R,sp} = 0.002 \text{ in} \).

The test conditions obtained during the tests (Table 5.2) differ from the nomi-
nal conditions reported in Table 5.1 due to the unavoidable uncertainties associated
with the experimental targeting of the selected nominal conditions.
Figure 5.14. Mapping pattern used for depth measurements.

Figure 5.15. Reference measurement used to calculate the regression length of the tested specimens. a) Reference depth of the un-tested specimen; b) Measurement of the mapping disk thickness.

Table 5.2. Test conditions calculated after the ablation tests

<table>
<thead>
<tr>
<th>Test #</th>
<th>$\eta_{th}$</th>
<th>Distance from the nozzle exit, in</th>
<th>Arc Power, kW</th>
<th>$N_2$ flow rate, kg/s</th>
<th>Test time, s</th>
<th>Working gas</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.64</td>
<td>2.5</td>
<td>750</td>
<td>0.115</td>
<td>8.74</td>
<td>$N_2$</td>
</tr>
<tr>
<td>1-A</td>
<td>0.64</td>
<td>2.5</td>
<td>750</td>
<td>0.112</td>
<td>8.88</td>
<td>$N_2$</td>
</tr>
<tr>
<td>2</td>
<td>0.64</td>
<td>2.5</td>
<td>550</td>
<td>0.114</td>
<td>16.84</td>
<td>$N_2$</td>
</tr>
<tr>
<td>3</td>
<td>0.64</td>
<td>1.5</td>
<td>550</td>
<td>0.107</td>
<td>8.97</td>
<td>$N_2$</td>
</tr>
<tr>
<td>4</td>
<td>0.64</td>
<td>3.5</td>
<td>550</td>
<td>0.107</td>
<td>9.00</td>
<td>$N_2$</td>
</tr>
</tbody>
</table>
Figure 5.16. Regression length of Teflon® samples. a) test #1; b) test #1-A; c) test #2; d) test #3; e) test #4.
Figure 5.16-b confirms the uniformity of the tip when the surface is completely surrounded by the flow and when it is in the shock-free region. Figure 5.16-c shows the enhanced regression due to the longer exposure to the high-enthalpy flow. The average deviations of the Teflon™ recession for the inner region and outer region with respect to that of the stagnation point are reported in Table 5.3.

Table 5.3. Average percentage deviation of the outer and inner diameter regression with respect to the stagnation point

<table>
<thead>
<tr>
<th>Test #</th>
<th>( \frac{L_r(\text{SP}) - L_r(\text{outer})}{L_r(\text{outer})} ), %</th>
<th>( \frac{L_r(\text{SP}) - L_r(\text{inner})}{L_r(\text{inner})} ), %</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9.86</td>
<td>2.83</td>
</tr>
<tr>
<td>1–A</td>
<td>N/A</td>
<td>2.09</td>
</tr>
<tr>
<td>2</td>
<td>6.77</td>
<td>0.36</td>
</tr>
<tr>
<td>3</td>
<td>30.51</td>
<td>7.57</td>
</tr>
<tr>
<td>4</td>
<td>2.89</td>
<td>2.39</td>
</tr>
</tbody>
</table>

The deviation between the outer and inner region of test #1 is due to the reflected shocks of the plume impinging directly on the outer borders of the test sample, resulting in a pronounced annular concavity being shaved out of the surface (Fig. 5.5-a). For test #3, however, the high deviation is due to the obtained particular situation for which the plume diameter is the same as the sample diameter leading to a similar sculpting of the front surface seen in test #1 (Fig. 5.5-c). In test #4 the deviations of the inner and outer diameter are nearly identical due to the fact that at \( x = 3.5 \text{ in} \) from the nozzle exit both phenomena described before are negligible (Fig. 5.5-d) on the surface surveyed with the mapping disk. The lower power used for test #2, with respect to test #1, led to a decrease in the expansion ratio, consequently closing the shock diamond and directing the shocks closer to the stagnation point of the sample. The enhanced heat flux due to the shock impingement generated an increase of the mass loss rate which, in turn, resulted in a smoothening of the
ablated surface at the stagnation region. The outer region of the sample shows a higher level of regression due to the fact that the plume, which is of the same diameter of the sample, impinged on the flat face of the Teflon® specimen (Fig. 5.5-b). The schematics of the shock patterns in Fig. 5.5 are still valid for the updated test conditions (Table 5.2) because the average expansion ratios calculated by using the code previously mentioned are very close to those obtained experimentally.

5.6.2 Pitot Probing

Because steady-state total pressure measurements were realized within only a few seconds of the probe being placed in the flow, the un-cooled Pitot tube was not severely damaged (Fig. 5.17).

Several runs at location $x = 2.5 \text{ in}$ from the nozzle exit, for both enthalpy levels, were executed in order to assess the feasibility of the pressure measurements. The test conditions and the quantities of interest obtained from the Pitot probing are reported in Table 5.4. The test number sequence is the same of Table 5.1.

The decay of total pressure downstream the nozzle exit due to the shock-shock interaction along the centerline of the plume (Fig. 5.5-d) cannot be directly detected
Table 5.4. Test conditions and results calculated for the Pitot tests

<table>
<thead>
<tr>
<th>Test #</th>
<th>$\dot{m}_{N_2}, \text{kg/s}$</th>
<th>$P$, kW</th>
<th>$t_{\text{run}}$, s</th>
<th>$p_{sp}$, psi</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.108</td>
<td>750</td>
<td>8.74</td>
<td>33.3 ± 2.9</td>
</tr>
<tr>
<td>2</td>
<td>0.116</td>
<td>550</td>
<td>16.84</td>
<td>35.7 ± 1.9</td>
</tr>
<tr>
<td>3</td>
<td>0.107</td>
<td>550</td>
<td>8.97</td>
<td>32.7 ± 2.2</td>
</tr>
<tr>
<td>4</td>
<td>0.108</td>
<td>550</td>
<td>9.00</td>
<td>24.3 ± 1.9</td>
</tr>
</tbody>
</table>

from the measurements of Table 5.4. This is because the pressure measurements of test #1, #2, #4 were not being performed for the same nitrogen flow rate due to the difficulty on targeting the same test conditions. The corresponding uncertainties on the pressure measurements have been calculated by using the standard deviations of the transducer signal recorded in the steady state conditions.

5.7 Data Analysis and Comparisons

In this section, the results of both the stagnation enthalpy and hot-wall stagnation heat flux obtained from the 1-D code (%5.4), the NASA correlation (Eq. 5.4) and the CV analysis using the correlation developed for the blockage factor (Eq. 5.24) are presented and compared among each other.

5.7.1 Control Volume Analysis using the Proposed Correlation for the Blockage Factor

The exponential-law correlation developed in %5.3 (Fig. 5.3) has been used to evaluate, first, the blockage effect from the mass loss rate and total pressure measurements (Eq. 5.8 and Eq. 5.24) and, then, the hot-wall heat transfer by using Eq. 5.9. The stagnation enthalpy is subsequently calculated from Eq. 5.13 once the hot-wall enthalpy corresponding to a non-ablating surface at $T_{hw} = T_{abl}$ is added (%5.2.4). The effective radius used is the averaged one calculated using the tip radius
before and after the run for each test executed (Table 5.5). Test #2 has a slightly higher effective radius because the specimen has been exposed to the high enthalpy flow for a longer time (Table 5.2).

Table 5.5. Average effective radius of the Teflon® specimens

<table>
<thead>
<tr>
<th>Test #</th>
<th>$R_{tp}$, in</th>
<th>$R_{eff}$, in</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.750 ± 0.062</td>
<td>2.812 ± 0.044</td>
</tr>
<tr>
<td>1-A</td>
<td>0.400 ± 0.062</td>
<td>1.429 ± 0.044</td>
</tr>
<tr>
<td>2</td>
<td>0.750 ± 0.062</td>
<td>2.883 ± 0.044</td>
</tr>
<tr>
<td>3</td>
<td>0.750 ± 0.062</td>
<td>2.812 ± 0.044</td>
</tr>
<tr>
<td>4</td>
<td>0.750 ± 0.062</td>
<td>2.812 ± 0.044</td>
</tr>
</tbody>
</table>

The uncertainty on the radius measurements has been calculated using the sensitivity of the caliper, while the error on the effective radius has been calculated using the propagation of the uncertainties. Table 5.6 reports the results obtained from the control volume analysis.

Table 5.6. Stagnation enthalpy and relative quantities calculated using the control volume approach

<table>
<thead>
<tr>
<th>Test #</th>
<th>$t_{run}$, s</th>
<th>$p_{sp}$, atm</th>
<th>$T_{abl}$, K</th>
<th>$m_{N_2}$, kg/s</th>
<th>$m_T$, kg/m$^2$ s</th>
<th>$H_p$, MJ/kg</th>
<th>$L_{Rsp}$, in</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>8.74</td>
<td>2.266</td>
<td>913</td>
<td>0.115</td>
<td>1.196</td>
<td>2.21</td>
<td>0.191 ± 0.001</td>
</tr>
<tr>
<td>2</td>
<td>16.84</td>
<td>2.429</td>
<td>916</td>
<td>0.114</td>
<td>0.830</td>
<td>2.21</td>
<td>0.367 ± 0.002</td>
</tr>
<tr>
<td>3</td>
<td>8.97</td>
<td>2.225</td>
<td>912</td>
<td>0.107</td>
<td>0.849</td>
<td>2.21</td>
<td>0.123 ± 0.002</td>
</tr>
<tr>
<td>4</td>
<td>9.00</td>
<td>1.655</td>
<td>900</td>
<td>0.107</td>
<td>0.859</td>
<td>2.21</td>
<td>0.141 ± 0.002</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test #</th>
<th>$q_{abl}$, MW/m$^2$</th>
<th>$\psi$</th>
<th>$q_{0, hw}$, MW/m$^2$</th>
<th>$h_{sp}$, MJ/kg</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.761 ± 0.024</td>
<td>0.777 ± 0.124</td>
<td>2.552 ± 0.439</td>
<td>4.915 ± 1.127</td>
</tr>
<tr>
<td>2</td>
<td>0.371 ± 0.024</td>
<td>0.936 ± 0.149</td>
<td>1.487 ± 0.264</td>
<td>3.227 ± 0.637</td>
</tr>
<tr>
<td>3</td>
<td>0.732 ± 0.048</td>
<td>0.910 ± 0.155</td>
<td>1.548 ± 0.316</td>
<td>3.413 ± 0.767</td>
</tr>
<tr>
<td>4</td>
<td>0.984 ± 0.048</td>
<td>0.805 ± 0.129</td>
<td>1.768 ± 0.343</td>
<td>3.587 ± 1.050</td>
</tr>
</tbody>
</table>
Test #1 – A has been excluded from the data analysis since the side effects on the control volume are not negligible anymore when the width of the CV (L in Fig. 5.1) becomes comparable to the tip diameter of the sample (D in Fig. 5.1). For this condition, the higher heat flux located at the shoulders of the samples does not allow neglecting the heat fluxes on CS$_2$ and CS$_4$. The comparison of the Teflon® recession rate for test #2 with those of test #3 and test #4 confirms the steadiness of the ablation mechanism after few seconds from the exposure to the high-enthalpy flow [93, 94, 95, 96, 97, 98]. The stagnation enthalpy, calculated using Eq. 5.13, is nearly constant along the centerline of the plume, as seen from test #3 (1.5 in from the nozzle exit) and test #4 (3.5 in from the nozzle exit). The deviation of about 5% could be due to a small misalignment of the sample and to the overall uncertainty on the regression measurements. The effects of the shock pattern are detectable looking at test #3 and test #4. In these two cases the results are even more accurate because approximately the same nitrogen mass-flow rates have been reproduced for both Teflon® and Pitot tests (comparison of Table 5.2 and Table 5.4). The heat flux calculated far downstream of the nozzle exit (test #4 in Table 5.6) is about 14% higher compared to that calculated at $x = 1.5$ in (test #3 in Table 5.6) because of the shocks located in that region (Fig. 5.5-d). Consequently a 1.14 heat flux enhancement factor is detectable at $x = 3.5$ in from the nozzle exit plane. The errors on the recession of the stagnation point ($L_{R,sp}$) have been calculated using the standard deviation on the regression measurements. The uncertainty on the density has been calculated using the propagation of the errors due to the mass measurements and neglecting the errors on the volume of the specimens. The errors on ablation heat flux, hot-wall heat flux and stagnation enthalpy have been estimated by using the formula for the propagation of the uncertainties to Eq. 5.8, Eq. 5.9 and Eq.
5.13, respectively. Fig. 5.18 reports the same exponential-law correlation of Fig. 5.3 including the data obtained from the experimental campaign illustrated in this work. The results show the data obtained to fit the trendline of the new correlation proposed. The R-squared increases from $R^2 = 0.67$ to $R^2 = 0.72$ once the UTA results are included. The power-law correlation of Fig. 5.4 shows an updated $R^2 = 0.82$. The experimental data obtained in this study also allows extending the validity of the correlation proposed for the blockage effect to the lower range of the blowing factor $B_2$ (comparison of Fig. 5.3 and Fig. 5.18).

5.7.2 NASA Correlation

The same correlation used to predict the recession of the Teflon® probes at the design stage of the experiments (Eq. 5.4) have been used to calculate the stagnation cold-wall heat flux ($q_{0,cw}$) and, thus, the stagnation enthalpy of the flow (Eq. 5.13). The hot-wall stagnation heat flux ($q_{0,hw}$), which has been used to compare the predictions of the NASA correlation with the results from the CV analysis, is calculated by using the correction factor in Eq. 5.14. Table 5.7 shows a summary of

![Figure 5.18. Updated exponential-law correlation for the blockage factor.](image)
the quantities calculated from the experimental tests and using the selected NASA correlation.

Table 5.7. Stagnation enthalpy and related quantities calculated using the NASA correlation

<table>
<thead>
<tr>
<th>Test #</th>
<th>$p_{sp}$, psi</th>
<th>$\dot{m}_T$, kg/(s·ft$^2$)</th>
<th>$q_{0,hw}$, MW/m$^2$</th>
<th>$h_{sp}$, MJ/kg</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.266 ± 0.197</td>
<td>1.196 ± 0.011</td>
<td>3.956 ± 0.681</td>
<td>5.227 ± 1.260</td>
</tr>
<tr>
<td>2</td>
<td>2.429 ± 0.129</td>
<td>0.830 ± 0.011</td>
<td>2.656 ± 0.311</td>
<td>3.539 ± 0.535</td>
</tr>
<tr>
<td>3</td>
<td>2.225 ± 0.150</td>
<td>0.849 ± 0.022</td>
<td>2.538 ± 0.372</td>
<td>3.725 ± 0.678</td>
</tr>
<tr>
<td>4</td>
<td>1.655 ± 0.129</td>
<td>0.859 ± 0.022</td>
<td>2.874 ± 0.620</td>
<td>3.899 ± 1.267</td>
</tr>
</tbody>
</table>

The use of the NASA correlation highlights a 13% increase of the heat flux at the centerline of the plume between the locations $x = 1.5$ in and $x = 3.5$ in that is due to the shock-shock interaction. The uncertainties on both stagnation heat flux and enthalpy have been estimated using the formula for the propagation of the errors to Eq. 5.4 and Eq. 5.13, respectively, by accounting for the uncertainties calculated by Hiester and Clark [83] on the coefficient and exponents of the correlation ($a = 0.0058 ± 0.0016$; $n = 0.58 ± 0.03$ and $m = 0.25 ± 0.02$) and imposing the gas-mixture constant $C$ to be affected by a 3.6% error as suggested by Sutton and Graves [89].

5.7.3 Numerical Simulations

The test conditions obtained experimentally for the Teflon® probing (Table 5.2) have been used to calculate the bulk total enthalpy and the cold-wall heat flux measured by a hypothetical copper calorimeter of the same dimensions of the specimens by using the numerical code described in §5.4 (Table 5.8).

The cold-wall heat flux has been calculated using the Fay-Riddell equation [16] while the hot-wall heat flux is derived from the correction factor in Eq. 5.14. The bulk total enthalpy is readily calculated using the energy balance method (Eq. 5.27).
Table 5.8. Bulk total enthalpy and relative quantities calculated using the numerical code

<table>
<thead>
<tr>
<th>Test #</th>
<th>$P$, kW</th>
<th>$\eta_{th}$</th>
<th>$\dot{m}_{N_2}$, kg/m²·s</th>
<th>$q_{0, hw}$, MW/m²</th>
<th>$h_B$, MJ/kg</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>750</td>
<td>0.64</td>
<td>0.115</td>
<td>2.410</td>
<td>3.165</td>
</tr>
<tr>
<td>2</td>
<td>550</td>
<td>0.64</td>
<td>0.114</td>
<td>1.421</td>
<td>3.088</td>
</tr>
<tr>
<td>3</td>
<td>550</td>
<td>0.64</td>
<td>0.107</td>
<td>1.524</td>
<td>3.290</td>
</tr>
<tr>
<td>4</td>
<td>550</td>
<td>0.64</td>
<td>0.107</td>
<td>1.566</td>
<td>3.290</td>
</tr>
</tbody>
</table>

The percentage deviations for both the results from the NASA correlation and the numerical code with respect to the control volume analysis have been summarized in Table 5.9.

Table 5.9. Stagnation heat flux and stagnation enthalpy deviations with respect to CV analysis using the new correlation ($R^2 = 0.67$)

<table>
<thead>
<tr>
<th>Test #</th>
<th>$\delta(q_{sp})_{hw}$, %</th>
<th>$\delta h_{sp}$, %</th>
<th>$\delta(q_{sp})_{hw}$, %</th>
<th>$\delta h_{sp}$, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>+25.97</td>
<td>+6.35</td>
<td>-5.57</td>
<td>-15.10</td>
</tr>
<tr>
<td>2</td>
<td>+28.99</td>
<td>+9.67</td>
<td>-4.43</td>
<td>-4.28</td>
</tr>
<tr>
<td>3</td>
<td>+21.05</td>
<td>+9.14</td>
<td>-1.55</td>
<td>-3.60</td>
</tr>
<tr>
<td>4</td>
<td>+22.72</td>
<td>+8.70</td>
<td>-11.43</td>
<td>-8.28</td>
</tr>
</tbody>
</table>

The maximum deviation between the stagnation heat flux calculated by using the new correlation for the blockage factor and the stagnation heat flux calculated with the numerical code, which has been validated using results from previous experimental campaigns, is lower than 12%. The small differences between the two approaches (Table 5.9) can be attributable to the negligible effect of the plasma “coring”, proven also in test #1 – A, indicating a uniform distribution of the flow energy across the plume diameter. In fact, the bulk total enthalpy and, thus, the peak heat flux calculated by using the numerical code provide the average total enthalpy while both the NASA correlation and the control volume approach allow calculating
the local value. The higher deviations between the hot-wall heat fluxes predicted by the NASA correlation and those derived from the new correlation for the blockage factor are due to both the uncertainty associated with the NASA correlation and the one related to the stagnation enthalpy, which has to be considered when the hot-wall heat flux is calculated using the NASA correlation (Eq. 5.14). The maximum uncertainties estimated for the stagnation heat flux calculated using both the correlation developed and the NASA correlation are about 20% (Table 5.6) and 22% (Table 5.7), respectively. Similar deviations on the stagnation heat flux measurements are obtained using heat flux probes such as null-point calorimeters and slug calorimeters [86, 105, 122]. In this scenario, the use of Teflon® for the characterization of high-enthalpy flows represents a simple and inexpensive technique which also provides the additional benefit of verifying the uniformity of the free-stream flow. The use of flat-faced cylinders as specimens for both ablation and Pitot probes is suggested to avoid the mutual interaction between the shape changing of the tip surface and the wall heat flux at the same location ($R_{eff} = const.$). The swing of the Pitot and Teflon® probes in the same run is also recommended in order to avoid repeatability errors.

5.8 Conclusions

The flow characterization of the arc-heated wind tunnel of the University of Texas at Arlington has been investigated in this work by using ablative probes in combination with total pressure measurements. Teflon® has been selected because of its well-documented ablation properties coupled to its low-temperature of ablation that allows detecting a significant recession of the surfaces when exposed to high-enthalpy flows. An analytical work based on dimensional analysis has been conducted in parallel to the experimental campaign with the purpose of correlating the blockage
factor to the parameters that characterize the ablation of Teflon® and that can be directly measured from experiments. In particular, the experimental data-set deriving from the Round-Robin program has been used for the dimensional analysis. At the design stage of the experiment, a correlation developed by NASA during the Round-Robin program has been selected to define the test time necessary to provide measurable recession of the specimen. Different locations downstream of the nozzle exit have been surveyed in order to characterize the centerline of the plume. A control volume analysis at the receding surface of the sample has been used in conjunction with the exponential-law correlation for the blockage factor in order to calculate, first, the ablation heat flux and, then, the hot-wall heat flux corresponding to a non-ablating surface. The NASA correlation selected to define the test conditions has also been used a posteriori to calculate the hot-wall heat flux. A simplified version of the convective-heating equation has been used to calculate the stagnation enthalpy for both analyses based on the control volume and the NASA correlation. The main results obtained from the experimental test campaign for the flow characterization of the 1.6 MW AHWT residing at UTA have been reported below.

- A considerable improvement of the blockage factor for Teflon®, with respect to the correlations available in literature, has been obtained including the pressure dependence inside the non-dimensional parameters. An exponential-law and a power-law correlation using a modified blowing factor, with respect to that one used in literature, increase the coefficient of determination of about 76% and 110%, respectively. An additional 7% improvement for the exponential-law correlation is obtained when the results from the experimental campaign presented are included.
• The range of validity of the correlation proposed for the blockage factor has been extended to lower values of the blowing parameter $B_2$ once the data obtained in this study have been included.

• The analysis of the ablated surfaces revealed the effects of the plume diameter with respect to the specimen frontal area, and the influence of the shock pattern on the ablated surface of the sample. A specimen with a reduced tip diameter was machined in order to survey the flow while avoiding any shock interaction on the surface. The analysis of the flat-faced ablated surface revealed a uniform regression which is an indication of the flow uniformity in the shock-free region.

• The results obtained by using the NASA correlation and the control volume analysis developed in this work show that both approaches provide similar results in term of stagnation heat flux and enthalpy prediction.

• The maximum uncertainty on the hot-wall heat flux calculation is about 20% and 22% for the control volume analysis using the new correlation for the blockage factor and the NASA correlation, respectively.

• The calculations of the stagnation heat flux by using both the NASA correlation and the control volume approach revealed a nearly 1.14 heat flux enhancement factor due to the shock interactions at $x = 3.5\text{ in}$ from the nozzle exit plane.

• The results obtained in this investigation show the possibility of using inexpensive and rapidly machinable Teflon® probes for the flow characterization of arc plasma facilities by having uncertainties comparable to those characteristics of standard metal heat-flux probes [105, 122]. The additional capability of detecting relevant ablation processes makes ablative probes an interesting tool to assess the uniformity of the flow downstream the nozzle exit by inspecting the footprint of the flow on the exposed surfaces. These features will serve to increase the capabilities on understanding the thermal response of candi-
date materials for thermal protection systems and to validate and calibrate numerical models simulating the ablation mechanism.

- The flow characterization in terms of nominal stagnation enthalpy and flow uniformity will allow contributing also to the design of the experimental campaign on variable transpiration cooling (Chap. 6) and to the interpretation of the post-test results.
6.1 Conclusions

The entire research work presented here shows the first systematic study of the main aspect to be considered in order to enable the practical use of the transpiration cooling for reusable hypersonic vehicles. A methodology has been presented for the development of reusable thermal protection systems starting from the numerical simulations of the boundary layer flow coupled with the thermal response of porous materials up to the design and characterization of customized porous materials complying with the cooling requirements of the thermal protection system. In particular, this work has addressed some of the fundamental questions such as: for prescribed flight conditions, is it possible to define an optimum transpiration strategy that allows saving coolant fluid once the required surface temperature is imposed? Can the blowing profiles simulated numerically be reproduced by using the state of the art manufacturing process of porous materials (i.e. variable material’s thickness, tailored porosity, variable thermal conductivity, diversified surface finishing etc.)? and, more in general, is it possible to define a methodology for the design of reusable thermal protection systems based on transpiration cooling starting from the preliminary definition of the optimum blowing profiles and material’s properties up to the successful test of a full-scale demonstrator? Additionally, other basic questions arisen from the necessity to demonstrate experimentally the cooling capability of the blowing profile selected and to prove the effectiveness of the proposed design methodology by the next experimental campaign on transpiration cooling that will be conducted in the
The most remarkable conclusions that can be drawn from the numerical and experimental work presented in this dissertation are summarized below:

• A reduced-order model for the analysis of the boundary layer flow over a flat plate coupled to the thermal response of porous material has been developed in the first part of this work. The numerical solution of the reduced order model, that is based upon the coupled use of Self-Similar Method and Difference Differential Method, allows implementing any type of transpiration strategy without binding the choice of the wall velocity to a particular distribution like in the reduced order models developed in the past (i.e. $V_W \propto 1/\sqrt{x}$). The analysis of the boundary layer flow with transpiration allowed identifying a particular blowing distribution (sawtooth wall velocity distribution) able to reduce by about 37% the coolant mass with respect to the other transpiration strategies analyzed. This result highlights the potential thermal-management implications of this concept applied to hypersonic vehicles. The analysis of the results obtained by using the high-fidelity fluid dynamics solver LAURA and GASP confirmed the capability of the reduced order model developed in Chap. 2 (AERO-Code) to capture the flow physics of the laminar boundary-layer flow over flat plates.

• The use of the variable transpiration with sawtooth wall velocity distribution has been also verified to reduce the wall heat flux at the stagnation point of a blunt body by 8% with respect to the uniform transpiration. The material’s thermal analysis for the flat plate shows the other beneficial effect that derives from the use of the variable transpiration. In fact, the sawtooth wall velocity distribution generates a pressure drop across the material’s thickness
that is 6% lower with respect to the uniform transpiration. The limited decrement of the pressure drop becomes fundamental to guarantee the structural integrity when higher external pressures are considered (e.g., during the descendant phase of hypersonic cruise vehicles or inside a supersonic combustion ramjet engine). The parametric analyses with respect to the thermophysical properties of the porous media show the remarkable influence of the porosity on defining the pressure drop across the material’s thickness while the material’s thermal conductivity is identified as the main parameter that determines the heat transfer between coolant and the solid material, if thermal insulator coolant fluids are considered. In this scenario, the use of the proposed reduced-order models at the initial design stage of thermal protection systems enables quick sensitivity analyses with respect to various control variables (wall velocity distribution, porosity, thermal conductivity, and material thickness). The integrated analysis performed for both the flat plate and blunted body have allowed prescribing the characteristics of the customized porous structure built by C-CAT (i.e. localized decrease of material thickness and variable porosity) with the final purpose of obtaining the variable transpiration cooling simulated numerically.

- A novel technique based on hot-film anemometry coupled to pressure measurements has been developed in order to characterize the above mentioned carbon-carbon prototype structure in terms of local blowing capability. In particular, the new concept of effective permeability, conceived as the local blowing capability of a porous structure with respect to a selected coolant fluid, has been introduced. The full-scale thermal protection system adopted in this study is representative of a complex porous structure for which the use of the average-permeability measurements obtained by standard method-
ologies would be questionable due to the structural morphology, unavoidable defectologies, non-uniform surface finishing and boundary constraints on the structure. In fact, all the aforementioned features generate highly asymmetric flow-path inside the porous structure, which cannot be captured by using standard methodologies for permeability measurements, that can create concentrated mechanical loads and hot-spots on the specimen surface.

- The statistical analysis of the number and dimensions of the void elements inside the porous lattice, coupled to the use of the theory of fluid flow through perforated plates, has been used to calculate the optimum distance of the sensing film from the wall and to define the minimum area to be probed by the hot-film. This fundamental parameters have been used to perform the mass-flux measurements on selected regions over three planes cut longitudinal to the symmetry axis of the cone. The resulting linear curve-fit, characteristic of the Darcy’s law, correlated fairly well the results obtained by using five air flow-rates and allowed defining the carbon-carbon prototype mask as a semipervious structure, based on the values obtained for the local effective-permeability.

- The local effective-permeability map built for the cone emphasizes the importance of having a methodology for the local characterization of full-scale components for thermal protection systems. Indeed, a pronounced asymmetric blowing capability of the cone has been detected because of the presence of two radial delaminations which enhanced the blowing near the nose-tip region. This characteristic opens the promising scenario of investigating the possibility to voluntarily introduce intrinsic defects, at the fabrication level, able to generate an effective permeability diversified accordingly to the cooling requirements of the TPS.
• The effective-permeability map of the complex porous structure characterized provides also the correct boundary conditions for the numerical aerothermal reconstructions of the experimental campaign on transpiration cooling (§6.2).

• The flow characterization of the arc-heated wind tunnel of the University of Texas at Arlington has been investigated in this work by using ablative probes in combination with total pressure measurements. The results obtained in this investigation show the possibility of using inexpensive and rapidly machinable Teflon® probes for the flow characterization of arc plasma facilities having uncertainties comparable to those characteristics of standard metal heat-flux probes. The additional capability of detecting relevant ablation processes makes ablative probes an interesting tool to assess the uniformity of the flow downstream the nozzle exit by inspecting the footprint of the flow on the exposed surfaces. These features will serve to increase the capabilities on understanding the thermal response of candidate materials for thermal protection systems and to validate and calibrate numerical models simulating the ablation mechanism.

• An analytical work based on dimensional analysis has been conducted in parallel to the experimental campaign with the purpose of correlating the blockage factor to the parameters that characterize the ablation of Teflon® and that can be directly measured from experiments. A considerable improvement of the blockage factor for Teflon®, with respect to the correlations available in literature, has been obtained by including the pressure dependence inside the non-dimensional parameters. An exponential-law and a power-law correlation using a modified blowing factor, with respect to that one used in literature, increase the coefficient of determination of about 76% and 110%, respectively.
An additional 7% improvement for the exponential-law correlation is obtained when the results from the experimental campaign presented are included.

- The flow characterization in terms of nominal stagnation enthalpy and flow uniformity will allow contributing also to the design of the experimental campaign on variable transpiration cooling and to the interpretation of the post-test results.

All the results obtained during this research work, and briefly presented above, have the side purpose of motivating the readers to push ahead the investigation of the thermal management of reusable hypersonic vehicles by combining multidisciplinary subjects for enabling hypersonic commercial flights and for a more affordable access to space by means of single-stage-to-orbit vehicles (SSTO). Indeed, starting from the numerical simulations of the interaction between the external aerodynamics and the thermal response of porous material up to the manufacturing of the prototype porous cone, different question marks have been posed at the scientific and technological level. For example, it will be fundamental to investigate the thermal response of porous materials at different length-scales in order to fully couple the external flow field to the heat-exchange properties in the porous media. This connection between material and fluid-dynamics sciences allows, as already mentioned in Chap. 4, exploring the possibility of improving the current manufacturing capabilities of composite materials in order to obtain porous materials having customized geometrical and thermo-physical properties able to provide the blowing profiles simulated numerically.

6.2 Future Work

The experimental campaign on transpiration cooling is currently in preparation at the University of Texas at Arlington and it will have the final purpose of demon-
Figure 6.1. Test setup used for measuring the total pressure losses along the coolant line. a) Internal plenum; b) External plenum.

Figure 6.2. Total pressure losses along the coolant line.

Demonstrating the feasibility of the variable transpiration for thermal protection systems. The entire work will serve as a baseline procedure for the design of reusable thermal protection systems based on transpiration cooling starting from the preliminary design up to the test of a full-scale demonstrator. The carbon-carbon nose will be instrumented with thermocouples in the plenum chamber (Fig. 4.2-a) in order to measure both the cold-wall and coolant temperatures for selected regions. The pressure in the plenum chamber of the specimen will be measured outside the test section by using a properly sized plenum and considering the total pressure losses along the coolant line (Fig. 6.1 and Fig. 6.2).
Figure 6.3. Schematic of the test setup used for the experimental campaign on transpiration cooling.

The temperature of the external surface will be monitored by using a thermocamera system. An overall look out of the test setup used for the experimental campaign is showed in Fig. 6.3. The use of a molybdenum mirror will be necessary in order to reflect the infrared radiation from the stagnation region back to the camera sensor. A properly sized Zinc Selenide (ZnSe) window will be used for maximizing the transmission across the viewport between 1.5 $\mu m$ and 14 $\mu m$.

A parallel work for the calibration of the thermocamera has been performed in collaboration with Dr. Alavi’s research group with the purpose of defining a series of calibration curves which relate the number of counts to the temperature measurements collected on selected points of the cone’s external surface by using a thermocouple rake (Fig. 6.4-a). The four points selected for the calibration are
Figure 6.4. Test setup used for the calibration of the infrared camera. a) Thermocouples rake; b) Thermal image with cone in-focus.

those above the plenum chamber that have been used for the characterization of the porous mask in terms of local effective permeability.

Both the infrared radiation coming emitted by the cone (Fig. 6.4-b) and the one reflected by the mirror (Fig. 6.5-a) have been collected to evaluate the impact on the effective temperature measurements of the mirror’s optical properties, direction and distance of the sensor from the radiative source. A thermocouple will also be placed inside the mirror to take into account for the radiation emitted by the mirror itself. The calibration curves obtained for the selected points on the surface (Fig. 6.5-b) include the effects of emissivity variation with temperature, $\epsilon(T)$.

The variation of emissivity with direction has been neglected because the same orientation of the camera with respect to the mirror, that has been used for the calibration procedure, will be maintained for the experiment on transpiration cooling (Fig. 6.3). Also the emissivity variation with wavelength will be neglected because of the use of a narrow bandwidth filter which minimize the effect of the shock layer radiation on the surface temperature measurements. Additionally, a preliminary analysis of the optical properties of the plasma flow generated by the arc heater allows classifying it as low-density plasma [123] based on the maximum density ratios.

166
obtainable at the nozzle exit ($\rho_e/\rho_{tot,e} < 0.3$). For these conditions, the emissivity of the air plasma in the infrared wavelength region is expected to be lower than $\epsilon_p = 0.1$ for total temperatures ranging from $3000 \text{ } K \leq T_{tot} \leq 8000 \text{ } K$ [123, 124, 125]. A simplified calculation for the change of plasma emissivity with plume thickness can be approached once assumed that the plasma behaves as a grey body (emissivity independent of the wavelength) [125]. In these conditions, the corrected Stefan-Boltzmann formula for emission of plasma in continuous spectrum can be used (Eq. 6.1) [125]:

$$J = [1 - e^{-k_w t}] \sigma T^4$$

(6.1)

Where, $J$ is the irradiance while $k_w$ and $t$ are the absorption coefficient and the plasma thickness, respectively. The absorption coefficient, known also as opacity, can be retrieved from reference values obtained for air plasma at different pressures and temperatures [125]. For plasma pressures $P_p = (1 \div 10) \text{ atm}$ and a reference temperature $T_p = 8000 \text{ } K$, the absorption coefficient is $k_w = (0.1 \div 1.005)$ and the
resulting total emissivity is $\epsilon = (0.002 \div 0.049)$ for plasma columns having radius spanning from 1 $in$ to 2 $in$.

A preliminary test by using a graphite disk as test sample will be also conducted for the nominal test conditions selected for the final experiments, and that have already been explored for the flow characterization reported in Chap. 5, with the purpose of defining the camera’s exposure time and aperture needed to avoid over-saturation effects during the experiment.
REFERENCES


sion, Philadelphia, PA, Air Force Flight Dynamics Laboratory Technical Re-

[26] A. Brune, S. Hosder, S. Gulli, and L. Maddalena, “Variable transpiration cool-
ing effectiveness in laminar and turbulent flows for hypersonic vehicles,” under


Francis, 2005.

[29] A. Hueppe, “Heat transfer modeling at an interface between a porous medium


[31] S. Gulli, L. Maddalena, C. McKelvey, A. Brown, Y. Nikishkov, and A. Makeev,
“Permeability measurements of complex porous structures for reusable thermal
protection systems (invited paper),” in 19th AIAA International Space Planes
and Hypersonic Systems and Technologies Conference, 2014, paper AIAA 2014-
2804.

[32] W. H. Gree and P. Sincebaugh, “Nondestructive evaluation of complex com-
posites using advanced computed tomography (CT) imaging,” General Electric
Company Re-entry and Environmental Systems Division, Philadelphia, PA,

[33] H. Bale, M. Blacklock, M. Bagley, D. B. Marshall, B. N. Cox, and
O. Ritchie, “Characterizing three-dimensional textile ceramic composites using
synchrotron x-ray micro-computed-tomography,” J. Am. Ceram. Soc., vol. 95,


[54] Z. Shi and X. Wang, “Comparison of Darcy’s law, the Brinkman equation, the modified N-S equation and the pure diffusion equation in PEM fuel cell modeling,” in *Proceeding of the COMSOL Conference*, 2007.


[84] S. S. Tompkins, J. N., Moss, C. M. Pittman, and L. M. Howser, “Numerical analysis of transient response of ablating axisymmetric bodies including the ef-


BIOGRAPHICAL STATEMENT

Stefano Gulli received his Bachelor Degree in Aerospace Engineering at the University of Rome “LaSapienza” where he performed analytical and experimental characterization of the carbon-epoxy composite materials used in all three stages of the solid rocket “VEGA”. He received his Master Degree in Astronautical Engineering at the same university in July 2009 under the guidance of Professor Claudio Bruno and Dr. Antonella Ingenito. His thesis work for the Master Degree was focused on the preliminary sizing of a hypersonic $M = 8$ commercial airliner for the European program “LAPCAT II”. During his Master Degree studies he also participated to the test and post-test analysis on a full-scale reusable thermal protection system under the supervision of Professor Mario Marchetti. The experimental campaign was performed in the 80 MW plasma wind tunnel at “CIRA”. He joined Professor Luca Maddalena’s research group at the University of Texas at Arlington in 2011, where he started his Ph.D. dissertation research on reusable thermal protection systems based on transpiration cooling.