A computationally efficient feature-based jet model for prediction of film-cooling flows

Author(s):
Burdet, André

Publication Date:
2005

Permanent Link:
https://doi.org/10.3929/ethz-a-005084097

Rights / License:
In Copyright - Non-Commercial Use Permitted
A computationally efficient feature-based jet model for prediction of film-cooling flows

A dissertation submitted to the
SWISS FEDERAL INSTITUTE OF TECHNOLOGY ZÜRICH

for the degree of
Doctor of Technical Sciences

presented by
André BURDET
Dipl.-Ing. Mec. EPFL
born 19.01.1977
citizen of Ursins-Orzens (VD), Switzerland and France

accepted on the recommendation of
Prof. Dr. R. S. Abhari, examiner
Prof. Dr. L. Kleiser, co-examiner
Dr. R. Bergholz, co-examiner

Zürich, 2005
Acknowledgments

Acknowledging all the people that helped me in the course of this research is maybe the most challenging task I encountered. I will certainly forget some of you. In fact, if you are reading this page, chances are that you are one of them. Thank you.

I would like to thank sincerely Prof. Reza Abhari for his constant guidance throughout all these years. He brought me his technical expertise and favorable working conditions. His management skills pushed me to give the best of myself.

I am very grateful to Prof. Leonhard Kleiser for accepting the role of co­examiner and for his suggestions and corrections concerning this thesis.

I would like to express my gratitude to Dr. Robert Bergholz for his support during all these years. He wisely brought me the industrial point of view underlying this thesis.

I would like to acknowledge the support of this research work by GE Aircraft Engines through their University Strategic Alliance program. Specifically the interaction with Dr. Fred Buck have been very helpful in the formulation of this thesis.

I had the chance to have access to some very precious sources of knowledge and experience. I am very grateful to Dr. Martin Rose for all his "disturbing" comments and questions. He pushed me to tackle the shadowy parts of my research. The constant support of Dr. Djamel Lakehal during the course of this work is very much appreciated. In good and bad technical and personal time, he always gave me good advises. I would also like to thank Dr. Anestis Kalfas for having opened my mind with brio to new frontiers in turbomachinery and also in other domains. The technical and social exchange with Dr. Joel Schlienger is greatly appreciated.
I would like to express my gratitude to all the members of the LSM for their friendliness and support. Marlene Hegner has always perfectly helped me in all sort of issues. I express my gratitude to the "Lab's French connection", that is Bob Mischo and Michel Mansour, for having created a very friendly atmosphere in good but also in stressful times. In the course of this work, the experimental data and comments provided by Stefan Bernsdorf are appreciated.

I would like to thank the following persons for their constant friendship throughout my doctoral time: Stefan Lendi, Patrice Carroz, Stéphane and Nathalie Amiguet, Stéphan Viennet, Alexandre Froidevaux, Leonardo Graf.

Finally, I would like to thank the people that count so much in my heart: my grandmother, Madeleine Neuville-Monnet, my parents, Pierre and Elisabeth Burdet, and the most beautiful woman a man can dream of, Magalie Blanc. Thank you so much for all your support.
Seite Leer / Blank leaf
Seite Leer /
Blank leaf
Abstract

The major contribution of this thesis is to propose a film cooling jet model that can be used in a numerical code (Computational Fluid Dynamics - CFD code), so that the prediction of the flow in a film-cooled turbine is facilitated in terms of effort and computational time.

The numerical code utilized in the course of this research is based on a Finite Volume Method, coupled with a Ni-Lax-Wendroff algorithm. The Reynolds-Averaged Navier-Stokes (RANS) equations are solved in connection with an algebraic, isotropic turbulence model (Baldwin-Lomax). The numerical code is typical of those used to predict flows in gas turbines.

The film cooling jet model is based on the macro flow features occurring just downstream of the injection hole. The modeling is based upon a large panel of experimental observations and utilized the governing equations of fluid mechanics. A numerical method, so-called Immersed Boundary Method (IBM), is introduced in order to include the model in the computational mesh. This method is entirely local. This induces a limited cost for the upgrade of the current numerical code. Meanwhile, it preserves an accurate representation of the jet model in the computational mesh. The coefficients of the model are calibrated with the aid of experimental measurements taken for a large spectrum of flow regimes, representative of engine conditions. The computational performances, when using the model in a numerical code, are evaluated as a function of the mesh density. More precisely, an axial and a lateral mesh density of \( N_X = 4 - 7 \) and \( N_Y = 7 - 11 \) grid nodes per hole diameter is shown to be optimal. For these mesh densities, the marginal gain of computational accuracy becomes low (\( C_Q \approx 1\% \)) and the loss of computational time (overhead \( \sigma \approx 1\% \)) compared to a flow without a jet is minimized.

In order to validate the use of the jet model, several flow cases of a steady coolant jet are presented in this thesis. First of all, the prediction of the aerodynamic structures of the jet downstream of the injection site, using the model, is reasonably
accurate. In reference to several published papers, it is proposed that the major discrepancies of the prediction are due to the isotropic treatment of the turbulent field inside the jet. The prediction capability of the model is compared with two other injection strategies. It is shown that the model produces results that are at least as accurate as the most complex injection strategy, while being 6 times faster. The prediction of the adiabatic film cooling effectiveness (typical design value) is found in agreement with experimental measurements taken from the literature. It is also shown that the model can be used for jet moderately laterally inclined. Coolant jets often pulsate in a turbine. The model is used in several flow cases of a typical pulsating, so-called quasi-steady regime. In this case, the overhead is shown to be only about $\sigma \approx 4\%$. The prediction of the unsteady flow field conforms the assumptions used. Eventually, it is shown that the model can be used to predict the transonic flow through a film-cooled turbine blade passage. The mesh size for a film-cooled turbine blade having 67 holes requires $2.13 \cdot 10^6$ grid nodes. The overhead is found to be about $\sigma \approx 0.1\%$ per hole. The prediction of the design values, such as blade loading, adiabatic film cooling effectiveness and heat transfer rate at blade surface is comparable to the experimental data.
Resumé

La contribution majeure de cette thèse est de proposer un modèle de jet de refroidissement qui peut être utilisé dans un code de calcul numérique (Computational Fluid Dynamics - CFD code) de telle sorte que la prédiction de l’écoulement dans une turbine refroidie par film est facilitée en termes d’effort et de temps de calcul.

Le code de calcul numérique utilisé au cours de cette recherche est basé sur une méthode au Volumes Finis, couplée avec un algorithme Ni-Lax-Wendroff. On résout les équations de Navier-Stokes moyennées (RANS) avec un modèle de turbulence algébrique et isotrope (Baldwin-Lomax). Ce code de calcul est typique de ceux utilisés pour prédire les écoulements dans les turbines à gaz.

Le modèle de jet de refroidissement est basé sur les structures d’écoulement macroscopiques se produisant juste à l’aval du trou d’injection. On base la modélisation sur un large panel d’observations expérimentales et en utilisant les équations régissant la mécanique des fluides. On propose une méthode numérique pour l’implémentation du modèle, appelée Immerged Boundary Method (IBM). Celle-ci permet d’inclure le jet tri-dimensionnel dans un maillage de calcul d’une manière entièrement locale. Ceci induit un coût de développement minimal du code de calcul utilisé, tout en préservant une représentation précise du modèle du jet. Les coefficients du modèle sont calibrés à l’aide de mesures expérimentales prises dans un large spectre de régimes d’écoulements, représentatifs de ceux régnant dans une turbine. Les performances de calcul lors de l’utilisation du modèle dans un code numérique sont évaluées en fonction de la densité du maillage. En particulier, on montre qu’un maillage ayant $N_X = 4 - 7$ et $N_Y = 7 - 11$ noeuds de grille par diamètre du trou d’injection, dans la direction axiale et latérale du jet, est optimal. Pour ces densités de maillage, le gain marginal de précision de calcul devient faible ($C_Q \approx 1\%$) et la perte de temps de calcul ($\sigma \approx 1\%$) par rapport à un problème d’écoulement sans jet est minimisé.

Différent cas d’écoulements contenant un jet stationnaire de refroidissement sont
présentés dans cette thèse afin de valider l'utilisation du modèle de jet. On montre tout d'abord que la prédiction des structures aérodynamiques du jet après le site d'injection, à l'aide du modèle, est raisonnablement précise. En référence à d'autres recherches publiées, on estime que les anomalies majeures de la prédiction sont dues au traitement isotrope du champ de turbulence à l'intérieur du jet. La capacité de prédiction du modèle est comparée à deux autres méthodes d'injection. On montre que le modèle produit des résultats au moins aussi précis que la méthode la plus complexe, tout en étant 6 fois plus rapide. La prédiction de l'efficacité adiabatique de refroidissement par film (valeur typique de design) est trouvée en accord avec des mesures expérimentales prises dans la littérature. On montre que le modèle peut être aussi utilisé pour prédire des jets modérément inclinés latéralement. Les jets de refroidissement ont souvent un caractère pulsant dans une turbine. Le modèle est utilisé dans plusieurs cas d'écoulements d'un régime pulsatif typique, dit quasi-stationnaire. Dans ce cas-là, la perte de temps de calcul est évaluée à seulement $\sigma \approx 4\%$. La prédiction de l'écoulement instationnaire est conforme aux hypothèses utilisées. Finalement, on démontre que le modèle peut être utilisé pour prédire l'écoulement transsonique à travers un passage d'aubes de turbine refroidies par film. La taille du maillage utilisé pour calculer l'écoulement à travers une passage d'aubes ayant 67 trous d'injection est de $2.13 \cdot 10^6$ noëuds de grille. On obtient une perte de temps de calcul de seulement $\sigma \approx 0.1\%$ par trou d'injection. La prédiction des valeurs de design, telles que la différence de pression sur l'aube, l'efficacité adiabatique de refroidissement par film et le taux de transfert de chaleur à la paroi de l'aube est comparable aux valeurs expérimentales.
Contents

List of Symbols XXI

1 Introduction 1
  1.1 Film-cooling in support of the improvement of energy conversion systems 1
  1.2 Modeling multi-scale flows to improve design 6
  1.3 Literature Review 10
  1.4 Research objectives 15
  1.5 Thesis outline 17

2 Numerical Method 21
  2.1 Governing equations 21
     2.1.1 The three-dimensional Reynolds-Averaged Navier-Stokes equations 21
     2.1.2 Conservative form 23
     2.1.3 Non-dimensionalization 25
     2.1.4 Turbulence model 26
  2.2 Ni Lax-Wendroff algorithm 29
     2.2.1 Time discretization 29
     2.2.2 Space discretization 30
     2.2.3 Mesh singularity 35
  2.3 Robustness of the algorithm: stability and smoothing 36
     2.3.1 Stability analysis 36
     2.3.2 Numerical smoothing 37
     2.3.3 Near wall smoothing 38
  2.4 Boundary conditions 38
     2.4.1 Inlet/outlet boundary conditions 38
     2.4.2 Wall boundary conditions 41
     2.4.3 Periodic boundary conditions 42
  2.5 Conclusion 42
### 3 Film Cooling Jet Model

3.1 Assumptions and structure of the jet model

3.1.1 Modeling approach

3.1.2 Geometrical parameters

3.1.3 Flow regimes

3.1.4 Flow structures included in the jet model

3.2 Positioning of the plane of injection: jet trajectory

3.2.1 Jet intrinsic frame of reference

3.2.2 Jet trajectory model

3.3 Principle of superposition: mixing and wake

3.3.1 Principle of superposition

3.3.2 Mixing functions

3.3.3 Flow profiles inputs

3.4 Non-mixed flow profile modeling

3.4.1 Penetration of the coolant jet

3.4.2 Coolant jet secondary flows - CVP structure

3.4.3 Freestream flow profile

3.5 Model overview

---

### 4 Numerical Implementation of the 3D Film-Cooling Jet

4.1 Implementation Strategy

4.1.1 Immersion of the 3D film-cooling jet in the computational mesh

4.1.2 Choice of the numerical method

4.2 Immersed Boundary Method

4.2.1 The forcing function

4.2.2 Immersed Boundary Method procedure

4.2.3 Immersion of the jet boundary conditions

4.3 Toroidal shape of the jet

4.3.1 Geometrical Characteristics of the 3D near-hole jet

4.3.2 The toroidal shape

4.3.3 3D film-cooling box

4.4 The full CFD-embedded film-cooling jet model

4.4.1 Closure of the model: input data to be fed

4.4.2 Probing the computational mesh

4.4.3 Full model overview
5 Experimentally-aided Model Calibration 95
5.1 Experimental setup .................................. 95
5.1.1 Test facility ........................................ 95
5.1.2 Measurement technique: PIV ...................... 98
5.1.3 Investigated flow regimes ......................... 100
5.2 Calibration of the film-cooling jet model .......... 101
5.2.1 Assumptions made for the calibration .......... 102
5.2.2 Calibration process and results .................. 105
5.3 Calibration for the conservation of mass .......... 108
5.3.1 Discharge Coefficient ............................. 108
5.3.2 Cross section of the toroidal jet body .......... 108

6 Performance Issues 111
6.1 Computational domain ................................ 111
6.1.1 Geometry ............................................ 111
6.1.2 Boundary conditions ............................... 113
6.1.3 Computational mesh ............................... 113
6.2 Computational history ................................ 114
6.2.1 Computational convergence ....................... 114
6.2.2 Computational time ................................ 117
6.3 Grid-independent solution ............................ 120
6.3.1 Mass Flow Conservation ......................... 120
6.3.2 Vortex circulation ................................ 123
6.3.3 Adiabatic film-cooling effectiveness ............ 125
6.4 Toward an engineering solution ...................... 127
6.4.1 Accuracy versus computational time ............ 127
6.4.2 Making use of the model for film-cooled turbine flow problems .................................. 132
6.5 Conclusion ............................................. 133

7 Steady Aerodynamic Validation 135
7.1 Introduction .......................................... 135
7.1.1 Strategy for validation ............................ 135
7.1.2 Overview of jet flow features .................... 136
7.2 Comparison to experimental data: velocity field .... 137
7.2.1 Flow regions of the coolant jet ................... 137
7.2.2 Influence of the streamwise injection angle $\alpha_0$ and momentum flux ratio $IR$ ............ 141
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.2.3</td>
<td>Agreements and discrepancies of the CFD prediction</td>
<td>144</td>
</tr>
<tr>
<td>7.3</td>
<td>Comparison to experimental data: vorticity field</td>
<td>149</td>
</tr>
<tr>
<td>7.3.1</td>
<td>Main issues of the comparison</td>
<td>149</td>
</tr>
<tr>
<td>7.3.2</td>
<td>Analysis of the measured and predicted vorticity field</td>
<td>150</td>
</tr>
<tr>
<td>7.4</td>
<td>Comparison with other numerical strategies</td>
<td>156</td>
</tr>
<tr>
<td>7.4.1</td>
<td>Brief description of the other numerical techniques</td>
<td>156</td>
</tr>
<tr>
<td>7.4.2</td>
<td>Static pressure field</td>
<td>157</td>
</tr>
<tr>
<td>7.4.3</td>
<td>Jet penetration and spreading</td>
<td>159</td>
</tr>
<tr>
<td>7.4.4</td>
<td>Drawbacks of the different injection strategies</td>
<td>162</td>
</tr>
<tr>
<td>8</td>
<td><strong>Steady Heat Transfer Validation</strong></td>
<td>167</td>
</tr>
<tr>
<td>8.1</td>
<td>Introduction</td>
<td>167</td>
</tr>
<tr>
<td>8.1.1</td>
<td>Thermal field versus heat transfer level prediction</td>
<td>167</td>
</tr>
<tr>
<td>8.1.2</td>
<td>Validation procedure</td>
<td>168</td>
</tr>
<tr>
<td>8.2</td>
<td>Streamwise injection</td>
<td>168</td>
</tr>
<tr>
<td>8.2.1</td>
<td>Test case definition</td>
<td>168</td>
</tr>
<tr>
<td>8.2.2</td>
<td>Predictive capability of the model - standard cases</td>
<td>169</td>
</tr>
<tr>
<td>8.3</td>
<td>Lateral injection</td>
<td>176</td>
</tr>
<tr>
<td>8.3.1</td>
<td>Test case definition</td>
<td>176</td>
</tr>
<tr>
<td>8.3.2</td>
<td>Predictive capability of the model - extreme cases</td>
<td>177</td>
</tr>
<tr>
<td>8.4</td>
<td>Conclusion</td>
<td>186</td>
</tr>
<tr>
<td>9</td>
<td><strong>Toward the Use of the Model in an Unsteady Flow Environment</strong></td>
<td>187</td>
</tr>
<tr>
<td>9.1</td>
<td>Introduction to pulsating jet in turbines</td>
<td>187</td>
</tr>
<tr>
<td>9.1.1</td>
<td>Impact of flow unsteadiness in high-pressure turbines</td>
<td>187</td>
</tr>
<tr>
<td>9.1.2</td>
<td>Relevant flow regimes of a pulsating jet</td>
<td>191</td>
</tr>
<tr>
<td>9.1.3</td>
<td>Goals of the study</td>
<td>193</td>
</tr>
<tr>
<td>9.2</td>
<td>Making use of the model for unsteady flows</td>
<td>193</td>
</tr>
<tr>
<td>9.2.1</td>
<td>Instantaneous blowing ratio model</td>
<td>193</td>
</tr>
<tr>
<td>9.2.2</td>
<td>Computational issues</td>
<td>196</td>
</tr>
<tr>
<td>9.3</td>
<td>Quasi-steady pulsating jet results</td>
<td>199</td>
</tr>
<tr>
<td>9.3.1</td>
<td>Computational history</td>
<td>199</td>
</tr>
<tr>
<td>9.3.2</td>
<td>Analysis of the computed flow field</td>
<td>203</td>
</tr>
<tr>
<td>9.4</td>
<td>Conclusion</td>
<td>207</td>
</tr>
</tbody>
</table>
L.3  Definition of loss coefficients ........................................... 309
L.4  Superposition of rows at pressure side surface with set 3 and set 4a  310
L.5  Superposition of rows at pressure side surface with set 3 and set 4ab 312
Seite Leer / Blank leaf
# List of Figures

1.1 Schematic of the GE90-94B turbofan engine. Its diameter is about 3.4 [m] and its length is about 7.3 [m]. It can deliver a thrust of 416 [kN] at sea level. *HP* denotes High Pressure, *LP* denotes Low Pressure. *C* denotes Compressor and *T* denotes Turbine. .......................... 2
1.2 Simplified example of the thermodynamic cycle of a gas turbine. .......... 3
1.3 Schematics of different air-based cooling systems. ......................... 5
1.4 Example of a film-cooled high-pressure turbine. Adapted from Hill and Peterson [44]. ................................................... 7
1.5 Example of the complexity of a 3D grid needed to solve simultaneously the flow through a film-cooled turbine passage as well as in the holes and plenum chamber (Heidmann et al. [42]). .................. 13

2.1 Stencil of the numerical algorithm ........................................ 31
2.2 Cells (*A – H*) surrounding grid node *G*N ................................ 32

3.1 Location of the near hole region within a film-cooled turbine blade. 44
3.2 Location of the plane of injection in the near hole region. ............... 45
3.3 Dimensions of cylindrical film-cooling holes. ............................ 46
3.4 Near hole coolant jet flow features. ........................................ 49
3.5 Creation and evolution of the coolant flow vortex ring. .................. 50
3.6 Short and long range entrainment (mixing). ................................ 51
3.7 Sketch of the different flow zones just downstream of the hole exit. 52
3.8 Wake upright vortices and horseshoe vortex. ................................ 53
3.9 Jet intrinsic frame of reference. ............................................. 54
3.10 Jet - solid curved cylinder analogy and forces applied to it. .......... 55
3.11 Idealized mixing of two streams. ......................................... 57
3.12 Cross view of the mixing and wake zones. ................................ 58
3.13 Model of the expansion of coolant through a hole. ....................... 61
3.14 Cross view of the CVP and their mirror-image set. ...................... 63
3.15 Vortex stretching. ........................................................... 64
3.16 Control Volume (CV) in the near hole region. ............................. 65
List of Figures

3.17 Idealized coolant velocity profile in the hole. 66
3.18 Model of the freestream flow boundary layer. 67
3.19 Detailed overview of the film cooling jet model. 69

4.1 Inclusion of the near-hole 3D jet body. 72
4.2 Pressure field in the near hole region. 73
4.3 Different methods to immerse a body within a computational grid. 75
4.4 The included surface as an immersed boundary condition. 78
4.5 Computational stencil used for the implicit Immersed Boundary Method. 80
4.6 A three-dimensional piece of the jet boundary in the near hole region. 84
4.7 A representation of the jet body in the near hole region: a tube with a curved axis. 86
4.8 The penetration of the jet body in the near hole region, from a cross section point of view. 87
4.9 Description of the toroidal three-dimensional surface mimicking the jet body near the hole. 88
4.10 Three-dimensional film-cooling box. 90
4.11 Upstream plane (UP) and pressure plane at wall (PW) for probing the near-hole freestream boundary layer. 92
4.12 Detailed overview of the full CFD-embedded film cooling jet model. 94

5.1 Schematic of the test facility, from Bernsdorf [13]. 96
5.2 Schematic of the test section, from Bernsdorf [13]. 98
5.3 PIV system mounted on a 2-axis traversing system. 99
5.4 Evolution of the main flow features of a film-cooling jet as a function of the momentum flux ratio. 106

6.1 Geometry of the computational domain. 112
6.2 Immersion of the toroidal jet surface; representation of the surface point $X_s$ for the mesh B1 (top left - I), C2 (top right - II), D3 (bottom left - III) and E4 (bottom right - IV). 116
6.3 Root Mean Square residual histories for different mesh densities: as a function of axial mesh density, based on a fixed lateral mesh density ($N_Y = 9$) (left -I) and as a function of lateral mesh density, based on a fixed axial mesh density ($N_X = 5$) (right -II). 117
6.4 Total computational time as a function of the mesh density. 118
6.5 Overhead in using the model as a function of the mesh density. 119
List of Figures XI

6.6 Global mass flow error history for a different mesh densities (left - I) and Global mass flow error found in all mesh density cases results (right - II). ........................................... 121
6.7 Local mass flow error for all mesh densities. ......................................................... 122
6.8 Axial evolution of circulation $\Gamma'_{xz}$ for different axial mesh densities, based on a lateral mesh density of $N_y = 9$ (left - I) and for different lateral mesh densities, based on an axial mesh density of $N_X = 5$ (right - II). ........................................... 124
6.9 Axial evolution of laterally averaged film-cooling effectiveness $\bar{\eta}$ for different axial mesh densities, based on a lateral mesh density of $N_y = 9$ (left - I) and for different lateral mesh densities, based on a axial mesh density of $N_X = 5$ (right - II). ........................................... 126
6.10 Marginal gain coefficient $C_T$ as a function of the axial mesh density for a constant lateral mesh density (left - I) and as a function of the lateral mesh density for a constant axial mesh density (right - II). 130
6.11 Marginal gain coefficient $C_\eta$ as a function of the axial mesh density for a constant lateral mesh density (left - I) and as a function of the lateral mesh density for a constant axial mesh density (right - II). 131

7.1 Numerical prediction, using the film-cooling model, of the flow field near the hole exit. On the flat plate $(Z/d = 0.0)$, contours of normalized static pressure $\Delta P_s$ are represented. Normalized streamwise vorticity $\omega'_x$ is shown at three different cross sections $(X/d = 2.0, 7.0, 12.0)$. The freestream streamlines ribbons are represented in yellow. ......................................................... 137
7.2 Contours of normalized streamwise velocity $U/U_f$, superimposed with freestream streamlines, in the center plane $Y/d = 0.0$, for a streamwise injection angle $\alpha_0 = 30^\circ$. ........................................... 139
7.3 Contours of normalized streamwise velocity $U/U_f$, superimposed with freestream streamlines, in the center plane $Y/d = 0.0$, for a streamwise injection angle of $\alpha_0 = 50^\circ$. ........................................... 140
7.4 Profile of streamwise velocity $(U/U_f)$ at different axial locations $(X/d = 2.0, 4.0, 6.0, 14.0)$, in the center plane $Y/d = 0.0$, for a streamwise injection angle of $\alpha_0 = 30^\circ$ (top) and $\alpha_0 = 50^\circ$ (bottom). 142
7.5 Profile of vertical velocity $(W/U_f)$ at different axial locations $(X/d = 2.0, 4.0, 6.0, 14.0)$, in the center plane $Y/d = 0.0$, for a streamwise injection angle of $\alpha_0 = 30^\circ$ (top) and $\alpha_0 = 50^\circ$ (bottom). 143
7.6 Contours of normalized streamwise velocity \((U/Uf)\), superimposed with cross velocity vectors \((V/Uf, W/Uf)\), in the cross section \(X/d = 4.0\), for a streamwise injection angle of \(\alpha_0 = 30^\circ\) .... 146

7.7 Contours of normalized streamwise velocity \((U/Uf)\), superimposed with cross velocity vectors \((V/Uf, W/Uf)\), in the center section \(X/d = 4.0\), for a streamwise injection angle of \(\alpha_0 = 50^\circ\) .... 147

7.8 Contours of normalized streamwise vorticity \((\omega_X^*)\) (top) and normalized vertical vorticity \((\omega_Z^*)\) (bottom) for a streamwise injection angle of \(\alpha_0 = 30^\circ\) and momentum flux ratio of \(IR = 4.0\) .... 150

7.9 Contours of normalized streamwise vorticity \((\omega_X^*)\) (top) and normalized vertical vorticity \((\omega_Z^*)\) (bottom) for a streamwise injection angle of \(\alpha_0 = 50^\circ\) and momentum flux ratio of \(IR = 4.0\) .... 151

7.10 Contours of predicted normalized streamwise vorticity \((\omega_X^*)\), in the cross sections \(X/d = -1.0\) and \(X/d = 0.0\), for a streamwise injection angle of \(\alpha_0 = 30^\circ\) .... 152

7.11 Normalized streamwise vorticity \(\omega_{xz}^*\) lateral profile in the normal plane \(Z/d = 0.4\), for an injection angles \(\alpha_0 = 30^\circ\) (left) and \(\alpha_0 = 50^\circ\) (right) .... 153

7.12 Circulation \(\Gamma_{xz}\) .... 154

7.13 Contours of predicted normalized static pressure \(\Delta P_s\) for different injection strategies, with a streamwise injection angle \(\alpha_0 = 30^\circ\). On the bottom right, axial profile of \(\Delta P_s\) on the center plane \(Y/d = 0.0\) is represented .... 158

7.14 Comparison of measured and predicted contours of normalized streamwise velocity \(U/Uf\), at the center plane \(Y/d = 0.0\) .... 160

7.15 Comparison of measured and predicted profiles of streamwise velocity \((U/Uf)\) at different axial locations \(X/d = 2.0, 4.0, 6.0\) and \(14.0\), in the center plane \(Y/d = 0.0\), for different numerical injection strategies .... 162

7.16 Comparison of measured and predicted contours of streamwise velocity \((U/Uf)\), superimposed with cross velocity vectors \((V/Uf, W/Uf)\), in the cross section \(X/d = 4.0\), for different numerical injection strategies .... 163

7.17 Contours of normalized absolute velocity \(U_a/Uf\), at the horizontal plane \(Z/d = 0.0\) for the full injection (FI - left) and wall injection (WI - right) strategies, with a streamwise injection angle \(\alpha_0 = 30^\circ\) .... 164

7.18 Normalized circulation \(\Gamma_{xz}^*\) for different injection strategies .... 165
7.19 Illustration of the complexity of the FI simulation. Mach number in the center plane $Y/d = 0.0$ (left) and secondary flow vector $(U_b/U_f, U_t/U_f)$ at the horizontal plane $Z/d = -0.35$ (right). 166

8.1 Predicted contours of the normalized total temperature $\theta$, at the cross section $X/d = 4.0$. $BR = 0.5$ on the left and $BR = 1.5$ on the right. 170

8.2 Measured and predicted contours adiabatic film-cooling effectiveness $\eta$ on the flat plate surface ($Z/d = 0.0$) for a low blowing ratio ($BR = 0.5$, top) and high blowing ratio ($BR = 1.5$, bottom). 171

8.3 Measured and predicted laterally-averaged adiabatic film-cooling effectiveness $\bar{\eta}$ for a low and medium blowing ratio ($BR = 0.5, 1.5$). 173

8.4 Laterally-averaged predictive error $\epsilon_{\Delta T}$ of the surface metal temperature for a low and medium blowing ratio ($BR = 0.5, 1.5$). 174

8.5 Measured and predicted centerline ($Y/d = 0.0$, left - I) and lateral ($X/d = 8.0$, right - II) adiabatic film-cooling effectiveness $\eta$. 175

8.6 Predicted freestream streamlines near hole exit, for a blowing ratio of $BR = 1.0$ and lateral injection angle of $\beta_0 = 60^\circ$. 179

8.7 Measured and predicted contours of non-dimensionalized temperature $\theta$ at the cross section $X/d = 5.0$ for a lateral injection angle of $\beta_0 = 0^\circ$ and a blowing ratio of $BR = 0.4$. 180

8.8 Measured and predicted contours of non-dimensionalized temperature $\theta$ at the cross section $X/d = 5.0$ for a lateral injection angle of $\beta_0 = 30^\circ$ and a blowing ratio of $BR = 0.9$. 180

8.9 Measured and predicted contours of non-dimensionalized temperature $\theta$ at the cross section $X/d = 5.0$ for a lateral injection angle of $\beta_0 = 60^\circ$ and a blowing ratio of $BR = 1.9$. 181

8.10 Measured and predicted contours of adiabatic film-cooling effectiveness $\eta$ on the flat plate surface ($Z/d = 0.0$) for a lateral injection angle of $\beta_0 = 0^\circ$ and a blowing ratio of $BR = 0.4$. 183

8.11 Measured and predicted contours of adiabatic film-cooling effectiveness $\eta$ on the flat plate surface ($Z/d = 0.0$) for a lateral injection angle of $\beta_0 = 30^\circ$ and a blowing ratio of $BR = 0.9$. 184

8.12 Measured and predicted contours of adiabatic film-cooling effectiveness $\eta$ on the flat plate surface ($Z/d = 0.0$) for a lateral injection angle of $\beta_0 = 60^\circ$ and a blowing ratio of $BR = 1.9$. 184
8.13 Measured and predicted levels of laterally-averaged adiabatic film-cooling effectiveness $\bar{\eta}$ for a lateral injection angle of $\beta_0 = 0^\circ$ (top left), $\beta_0 = 30^\circ$ (top right), $\beta_0 = 60^\circ$ (bottom left), $\beta_0 = 0^\circ$ (bottom right). ............................................................... 185

9.1 Graphical model, derived from unsteady CFD, of the midspan rotor surface pressure. Adapted from Abhari and Epstein [1]. ........ 189

9.2 Theoretical variation of the blowing ratio $BR$ as a function of the near-hole static pressure $P_s$, for quasi-steady freestream and coolant flow condition. It is given in terms of non-dimensionalized blowing ratio $BR^* = \left( \frac{BR - BR}{BR} \right) / BR$ and non-dimensionalized static pressure $P_s^* = \left( \frac{P_s - P_s}{P_s} \right) / P_s$. .................................................. 195

9.3 Evolution of the computed $BR$, modeled $BR_{1,DR}$ blowing ratios and the near-hole static pressure $P_s$ as a function of time $t/\tau_j$. On the left, test case $U2$ and on the right, test case $U1$. ................. 200

9.4 Evolution of the freestream momentum $\bar{\rho}\bar{U}^f$, coolant momentum $\bar{\rho}\bar{U}^c$ and near-hole static pressure $P_s$ as a function of time $t/\tau_j$. On the left, test case $U2$ and on the right, test case $U1$. ................. 201

9.5 Evolution of the computed $BR$, modeled $BR$ blowing ratios (left), freestream momentum $\bar{\rho}\bar{U}^f$, coolant momentum $\bar{\rho}\bar{U}^c$ (right) and near-hole static pressure $P_s$ as a function of time $t/\tau_j$. ................. 203

9.6 Axial evolution of the boundary layer streamwise velocity $\hat{U} / \bar{U}^f$ for the test case $U2$ ($BR = 0.5$). ................................................................. 204

9.7 Axial evolution of the boundary layer streamwise velocity $\hat{U} / \bar{U}^f$ for the test case $U1$ ($BR = 2.13$). ................................................................. 204

9.8 Axial evolution of the boundary layer streamwise velocity $\hat{U} / \bar{U}^f$ for the test case $U3$ ($BR = 2.13$). ................................................................. 205

9.9 Laterally-averaged film-cooling effectiveness $\bar{\eta}$ for the test case $U2$ ($BR = 0.5$). ................................................................. 206

9.10 Laterally-averaged film-cooling effectiveness $\bar{\eta}$ for the test case $U1$ ($F_j = 250$ [Hz], left) and for the test case $U3$ ($F_j = 1000$ [Hz], right). ................................................................. 206

10.1 Sketch of the linear turbine cascade geometry of Ashworth [6].... 211

10.2 Film-cooling holes configurations (adapted from Rigby et al. [74]). 212
10.3 Computational domain mesh, in the $XY$-plane at hub (2D grid - left) and in the $XZ$-plane at the pressure side surface and periodicity plane (right). .......................................................... 215
10.4 Mesh near cooling holes, for set 1. Before adaption on the left and after adaption on the right. .......................................................... 217
10.5 Computational history (Root-Mean Square residuals ($DQ_{rms}$)) of the film-cooled linear turbine cascade (adiabatic wall) with set 1, set 3, set 4ab and $BR = 1.0$. ......................................................... 218
10.6 Overhead $\sigma$ as a function of the number of holes $N_h$ found in the linear film-cooled turbine cascade test case. .......................................................... 219
10.7 Static pressure coefficient $C_{ps}$ at midspan section ($Z/C_{AC} = 0.718$). .......................................................... 221
10.8 Nusselt number $Nu$ at the blade surface, superimposed with the surface flow streamlines. .......................................................... 222
10.9 Measured (EXP) and predicted (CFD) isentropic Mach number $M_{is}$ at midspan (left) and Nusselt number $Nu_0$ (right), for the uncooled blade. .......................................................... 223
10.10 Measured (EXP) and predicted (CFD) isentropic Mach number $M_{is}$ at midspan, for two blowing ratios ($BR = 1.0, 1.5$). Injection at set 1, set 3 and set 4ab. .......................................................... 224
10.11 Predicted contours of adiabatic film-cooling effectiveness $\eta$ at the suction side surface for a blowing ratio of $BR = 1.0$ (left) and $BR = 1.5$ (right). .......................................................... 226
10.12 Measured (EXP) and predicted (CFD) adiabatic film-cooling effectiveness $\eta$ at midspan suction side, for two blowing ratios ($BR = 1.0, 1.5$). Injection at set 1. .......................................................... 227
10.13 Measured (EXP) and predicted (CFD) Nusselt number $Nu$ at midspan, for two blowing ratios ($BR = 1.0, 1.5$). Injection at set 1 and set 3. .......................................................... 228
10.14 Pressure loss coefficient $\gamma_p$. .......................................................... 230
10.15 Enthalpy loss coefficient $\gamma_H$. .......................................................... 231
10.16 Predicted contours of adiabatic film-cooling effectiveness $\eta$ at the pressure side surface for a blowing ratio of $BR = 1.0$. Injection at set 3 and set 4a (left) and injection at set 3 only, superposed with injection at set 4a only (right). .......................................................... 234
10.17 Predicted contours of normalized total temperature $\theta$ superimposed with the coolant secondary flow vectors $U_{2nd}^c$, near the pressure side surface, for a blowing ratio of $BR = 1.0$. Injection at set 3 and set 4a. ................................................. 236

10.18 Predicted contours of normalized total temperature $\theta$ superimposed with the coolant secondary flow vectors $U_{2nd}^c$, near the pressure side surface, for a blowing ratio of $BR = 1.0$. Injection at set 4a only. .................................................. 237

10.19 Predicted contours of normalized total temperature $\theta$ superimposed with the coolant secondary flow vectors $U_{2nd}^c$, near the pressure side surface, for a blowing ratio of $BR = 1.0$. Injection at set 3 and set 4a. .................................................. 238

10.20 Predicted contours of normalized total temperature $\theta$ superimposed with the coolant secondary flow vectors $U_{2nd}^c$, near the pressure side surface, for a blowing ratio of $BR = 1.0$. Injection at set 4a only. .................................................. 239

10.21 Schematic of the influence of the upstream jets upon downstream jets. .................................................. 240

A.1 Description of the toroidal three-dimensional surface mimicking the jet body near the hole. .................................................. 266

A.2 Variation of the two axis $A$ and $B$ of the jet elliptic cross section ellipse. .................................................. 268

A.3 Normal vectors to the toroidal surface at $X_{le}$ and $X_{wi}$ to find the center $X_{tc}$ of the toroidal shape .................................................. 270

B.1 Three-dimensional film-cooling box. .................................................. 273

B.2 Regions and boolean. .................................................. 274

B.3 Two-dimensional plane where to search the initial guess of the surface point pertaining to the toroidal shape. .................................................. 277

B.4 Pseudo-mirror point in case of mirror point outside of the computational mesh. .................................................. 280

B.5 Ghost node identified as a corner node. .................................................. 281

E.1 Comparison of the measured (left) [75] and modeled (right) coolant jet flow field on the plane of injection. .................................................. 292
List of Figures

J.1 Arrangement of the hole exit grid nodes at the flat plate surface ($Z/d = 0.0$). .................................................. 299

K.1 Dimensions of the computational domain used in the FI strategy, in [mm]. ......................................................... 302
K.2 Center plane cut of the computational mesh used in the FI strategy. ................................................................. 303
K.3 Three-dimensional view of the computational mesh in the near hole region. ....................................................... 304
K.4 Root Mean Square (RMS) residuals for the initial guess computation (left) and final computation (right). ............... 305

L.1 Predicted adiabatic film-cooling effectiveness $\eta$ at the pressure side surface for a blowing ratio of $BR = 1.0$. Injection at set 3 and set 4a (left) and injection at set 3 only (right). ................... 310
L.2 Predicted adiabatic film-cooling effectiveness $\eta$ at the pressure side surface for a blowing ratio of $BR = 1.5$. Injection at set 3 and set 4a (left) and injection at set 3 only superposed with injection at set 4a only (right). .......................................................... 311
L.3 Predicted adiabatic film-cooling effectiveness $\eta$ at the pressure side surface for a blowing ratio of $BR = 1.0$. Injection at set 3 and set 4ab (left) and Injection at set 3 only superposed with injection at set 4ab only (right). .................................................. 312
L.4 Predicted adiabatic film-cooling effectiveness $\eta$ at the pressure side surface for a blowing ratio of $BR = 1.5$. Injection at set 3 and set 4ab (left) and Injection at set 3 only superposed with injection at set 4ab only (right). .................. 313
Seite Leer / Blank leaf
## List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Input flow variables</td>
<td>60</td>
</tr>
<tr>
<td>4.1</td>
<td>Comparison of numerical methods to include the film-cooling jet model in the computational mesh</td>
<td>77</td>
</tr>
<tr>
<td>4.2</td>
<td>Input data to the model that need to be probed in the computational mesh</td>
<td>91</td>
</tr>
<tr>
<td>5.1</td>
<td>Hole dimensions for the two available sets</td>
<td>98</td>
</tr>
<tr>
<td>5.2</td>
<td>Freestream flow parameters</td>
<td>100</td>
</tr>
<tr>
<td>5.3</td>
<td>Coolant to freestream flow parameters for a steady jet, Bernsdorf’s experiment [13]</td>
<td>101</td>
</tr>
<tr>
<td>5.4</td>
<td>Film-cooling jet model coefficients to be tuned</td>
<td>101</td>
</tr>
<tr>
<td>6.1</td>
<td>Geometrical size of the computational domain</td>
<td>112</td>
</tr>
<tr>
<td>6.2</td>
<td>Identity code, axial $N_X$ and lateral $N_Y$ mesh densities and total number of grid nodes</td>
<td>114</td>
</tr>
<tr>
<td>6.3</td>
<td>Qualitative classification of the overhead level as a function of the mesh density</td>
<td>120</td>
</tr>
<tr>
<td>7.1</td>
<td>Comparison of the different numerical strategies to inject the coolant jet</td>
<td>157</td>
</tr>
<tr>
<td>8.1</td>
<td>Scaling of the flow conditions made for the computation</td>
<td>178</td>
</tr>
<tr>
<td>9.1</td>
<td>Different modes of turbine unsteadiness</td>
<td>188</td>
</tr>
<tr>
<td>9.2</td>
<td>Flow regimes investigated for the quasi-steady pulsating jet problem</td>
<td>197</td>
</tr>
<tr>
<td>9.3</td>
<td>Pulsating jet: computational time to reach a periodic solution</td>
<td>198</td>
</tr>
<tr>
<td>9.4</td>
<td>Overhead obtained for the computed pulsating jet cases</td>
<td>198</td>
</tr>
<tr>
<td>10.1</td>
<td>Geometrical dimensions of the linear turbine cascade of Ashworth [6]</td>
<td>210</td>
</tr>
<tr>
<td>Table</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>-------</td>
<td>-----------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>10.2</td>
<td>Experimental flow condition of the linear turbine cascade of Rigby et al. [74]</td>
<td>211</td>
</tr>
<tr>
<td>10.3</td>
<td>Coolant flow conditions.</td>
<td>213</td>
</tr>
<tr>
<td>A.1</td>
<td>Unknown parameters to be used as a function $T(\chi, \xi, \eta)$ which defines the toroidal surface.</td>
<td>269</td>
</tr>
<tr>
<td>A.2</td>
<td>Geometrical characteristics of the toroidal jet surface that are known a priori.</td>
<td>269</td>
</tr>
<tr>
<td>F.1</td>
<td>Calibrated functional dependencies of the model coefficient for $\alpha_0 = 30^\circ$</td>
<td>293</td>
</tr>
<tr>
<td>G.1</td>
<td>Calibrated functional dependencies of the model coefficient for $\alpha_0 = 50^\circ$</td>
<td>294</td>
</tr>
<tr>
<td>H.1</td>
<td>Calibrated functional dependencies of the coefficients of the small ellipse of the toroidal shape, for $\alpha_0 = 30^\circ$.</td>
<td>295</td>
</tr>
<tr>
<td>H.2</td>
<td>Calibrated functional dependencies of the coefficients of the small ellipse of the toroidal shape, for $\alpha_0 = 50^\circ$.</td>
<td>295</td>
</tr>
<tr>
<td>L.1</td>
<td>Percentage of injected coolant mass flow as a function of the incoming freestream mass flow.</td>
<td>307</td>
</tr>
<tr>
<td>L.2</td>
<td>Global mass flow error $\epsilon_m^G$</td>
<td>308</td>
</tr>
</tbody>
</table>
# List of Symbols

## Latin letters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>matrix coefficient</td>
</tr>
<tr>
<td>$A$</td>
<td>vertical ellipse axis [m]</td>
</tr>
<tr>
<td>$A_1$</td>
<td>lower semi-axis of small ellipse (toroidal shape) [m]</td>
</tr>
<tr>
<td>$A_2$</td>
<td>upper semi-axis of small ellipse (toroidal shape) [m]</td>
</tr>
<tr>
<td>$A_m$</td>
<td>mean vertical axis of small ellipse (toroidal shape) [m]</td>
</tr>
<tr>
<td>$A_{m1}$</td>
<td>lower semi-axis of mixing zone [m]</td>
</tr>
<tr>
<td>$A_{m2}$</td>
<td>upper semi-axis of mixing zone [m]</td>
</tr>
<tr>
<td>$A_{w1}$</td>
<td>lower semi-axis of wake zone [m]</td>
</tr>
<tr>
<td>$A_{w2}$</td>
<td>upper semi-axis of wake zone [m]</td>
</tr>
<tr>
<td>$A_0^+$</td>
<td>mixing length damping coefficient</td>
</tr>
<tr>
<td>$a$</td>
<td>speed of sound [m/s]</td>
</tr>
<tr>
<td>$B$</td>
<td>body force vector</td>
</tr>
<tr>
<td>$B$</td>
<td>lateral ellipse axis [m]</td>
</tr>
<tr>
<td>$B_1$</td>
<td>right semi-axis of small ellipse (toroidal shape) [m]</td>
</tr>
<tr>
<td>$B_2$</td>
<td>left semi-axis of small ellipse (toroidal shape) [m]</td>
</tr>
<tr>
<td>$B_{f cm}$</td>
<td>3D film-cooling box</td>
</tr>
<tr>
<td>$B_{m1}$</td>
<td>right semi-axis of mixing zone [m]</td>
</tr>
<tr>
<td>$B_{m2}$</td>
<td>left semi-axis of mixing zone [m]</td>
</tr>
<tr>
<td>$B_{w1}$</td>
<td>right semi-axis of wake zone [m]</td>
</tr>
<tr>
<td>$B_{w2}$</td>
<td>left semi-axis of wake zone [m]</td>
</tr>
<tr>
<td>$b$</td>
<td>bi-tangential to wall unity vector</td>
</tr>
<tr>
<td>$b$</td>
<td>bi-tangential to wall direction</td>
</tr>
<tr>
<td>$C$</td>
<td>boolean function, set cell character</td>
</tr>
<tr>
<td>$C_{AC}$</td>
<td>blade axial chord [m]</td>
</tr>
<tr>
<td>$C_{cp}$</td>
<td>Baldwin-Lomax model coefficient</td>
</tr>
<tr>
<td>$C_d$</td>
<td>hole discharge coefficient</td>
</tr>
<tr>
<td>$C_h$</td>
<td>jet width growing coefficient</td>
</tr>
<tr>
<td>$C_{kleb}$</td>
<td>Klebanoff coefficient</td>
</tr>
</tbody>
</table>
\( C_{n} \)  
flow resistance coefficient

\( C_{ps} \)  
static pressure coefficient \( \frac{p_{s} - p_{out}}{p_{in} - p_{out}} \)

\( C_{Q} \)  
marginal gain coefficient

\( C_{U} \)  
velocity mixing coefficient

\( C_{T} \)  
temperature mixing coefficient

\( C_{TC} \)  
blade true chord \([\text{m}]\)

\( C_{wk} \)  
wake coefficient

\( C_{xs} \)  
convergence criterion (Newton-Raphson)

\( C_{\lambda} \)  
convergence criterion (Newton-Raphson)

\( c_{p} \)  
specific heat at constant pressure \([\text{J/kgK}]\)

\( c_{v} \)  
specific heat at constant volume \([\text{J/kgK}]\)

\( DQ \)  
residual

\( d \)  
hole diameter \([\text{m}]\)

\( d_{f} \)  
wall damping vortex function parameter \([\text{m}]\)

\( dA \)  
amplitude of the variation of the semi axis \( A \) \([\text{m}]\)

\( dB \)  
amplitude of the variation of the semi axis \( B \) \([\text{m}]\)

\( dC \)  
infinitesimal centrifugal force \([\text{N}]\)

\( dl \)  
infinitesimal jet length \([\text{m}]\)

\( dN \)  
infinitesimal flow resistance force \([\text{N}]\)

\( dA \)  
amplitude of the variation of the semi axis \( \Lambda \) \([\text{m}]\)

\( E \)  
total energy \([\text{J/kg}]\)

\( e_{i} \)  
unity vector in \( i^{th} \) direction

\( F \)  
axial flux vector

\( F' \)  
discretized forcing vector

\( F_{kleb} \)  
Klebanoff intermittency function

\( F_{wk} \)  
wake function

\( f \)  
forcing function vector

\( f_{BL} \)  
blending function

\( f_{l} \)  
near-wall damping function of vortex \( l \)

\( f_{c} \)  
blending parameter for corner nodes

\( G \)  
lateral flux vector

\( G_{N} \)  
grid node

\( H \)  
vertical flux vector

\( H \)  
total enthalpy \([\text{J/kg}]\)

\( H'f \)  
freestream boundary layer shape factor

\( H_{HT} \)  
blade hub to tip height \([\text{m}]\)

\( H_{s} \)  
static enthalpy
$h$  
bulk width of the jet  

$h_0$  
bulk width of the jet at hole exit  

$k$  
turbulent kinetic energy  

$k_1$  
jet trajectory coefficient  

$l$  
length of the hole  

$l$  
total length of the jet trajectory  

$l_{\text{mix}}$  
mixing length  

$\bar{u}_w$  
wake intensity velocity deficit  

$\bar{T}_w$  
wake intensity temperature deficit  

$\dot{m}$  
mass flow rate  

$\dot{m}_{\text{ideal}}$  
ideal jet mass flow rate  

$\dot{m}_{\text{actual}}$  
actual jet mass flow rate  

$N$  
number of grid points  

$N_h$  
number of cooling holes  

$N_X$  
number of grid nodes per hole diameter (X-dir)  

$N_Y$  
number of grid nodes per hole diameter (Y-dir)  

$N_v$  
number of vertices in a cell  

$N_{2D}$  
total number of grid nodes in a 2D plane  

$N_{3D}$  
total number of grid nodes  

$n$  
normal to wall direction  

$n$  
normal to wall unity vector  

$\mathbb{P}$  
boolean function (plane of injection)  

$P$  
static pressure  

$P$  
plane of injection function  

$P_{BB}$  
pitch blade to blade distance  

$P_s$  
near-hole static pressure  

$P_T$  
total pressure  

$Q$  
state vector  

$Q$  
state value  

$q'_i$  
$ith$ heat flux component  

$q$  
sum of all weighting functions  

$R$  
maximum radius  

$R_g$  
gas constant  

$R_m$  
mean radius of the big circle of the toroidal shape  

$r$  
radius  

$S$  
any cartesian direction  

$S$  
any surface  

$S/S_{\text{max}}$  
fraction of wetted surface
\( S_i \)  
\( i^{th} \) surface vector component \([m^2]\)

\( s \)  
hole to hole pitch distance \([m]\)

\( T \)  
boolean function (toroidal surface)

\( T \)  
transformation matrix

\( T^{-1} \)  
inverse transformation matrix

\( T \)  
static temperature \([K]\)

\( T \)  
function of toroidal shape

\( T_T \)  
total temperature \([K]\)

\( t \)  
tangential to wall unity vector

\( t \)  
tangential to wall direction

\( t \)  
time \([s]\)

\( U \)  
absolute magnitude of velocity \([m/s]\)

\( u \)  
velocity vector \([m/s]\)

\( u \)  
axial velocity \([m/s]\)

\( u_{diff} \)  
velocity velocity \([m/s]\)

\( u_{i,j} \) or \( k \)  
\( i^{th}, j^{th} \) or \( k^{th} \) cartesian velocity component \([m/s]\)

\( u_r \)  
wall friction velocity \([m/s]\)

\( u_x \)  
\((i, j = 1)\) cartesian velocity component \([m/s]\)

\( u_y \)  
\((i, j = 2)\) cartesian velocity component \([m/s]\)

\( u_z \)  
\((i, j = 3)\) cartesian velocity component \([m/s]\)

\( V \)  
volume \([m^3]\)

\( v \)  
lateral velocity \([m/s]\)

\( w \)  
vertical velocity \([m/s]\)

\( w_n \)  
weighting function at node \( n \) \((n = 1..N)\)

\( X \)  
cartesian location vector of a point \([m]\)

\( X' \)  
cartesian location vector of a pseudo-point \([m]\)

\( X \)  
axial position \([m]\)

\( (X_{jc}, Y_{jc}, Z_{jc}) \)  
jet center cartesian coordinates \([m]\)

\( x \)  
\((i, j = 1)\) cartesian coordinate \([m]\)

\( x_{i,j} \) or \( k \)  
\( i^{th}, j^{th} \) or \( k^{th} \) cartesian coordinate \([m]\)

\( Y \)  
lateral position \([m]\)

\( Y (\xi, \chi) \)  
toroidal shape subfunction

\( Y_H \)  
enthalpy loss coefficient

\( Y_p \)  
pressure loss coefficient \([m]\)

\( y \)  
\((i, j = 2)\) cartesian coordinate \([m]\)

\( Z \)  
vertical position \([m]\)

\( Z^+ \)  
non-dimensional (stretched) wall distance \([m]\)

\( z \)  
\((i, j = 3)\) cartesian coordinate \([m]\)
Non-dimensional numbers

\( Re \)  Reynolds number
\( M \)  Mach number
\( Pr \)  Prandtl number
\( Nu \)  Nusselt number
\( DR \)  Density ratio
\( BR \)  Blowing ratio
\( IR \)  Momentum flux ratio

Greek letters

\( \alpha_0 \)  streamwise hole injection angle  \[ \text{[rad]} \]
\( \alpha \)  local streamwise jet angle  \[ \text{[rad]} \]
\( \alpha_{BL} \)  Baldwin-Lomax model coefficient
\( \beta_0 \)  lateral (compound) hole injection angle  \[ \text{[rad]} \]
\( \beta \)  local lateral jet angle  \[ \text{[rad]} \]
\( \Gamma \)  circulation  \[ \text{[m}^2\text{/s]} \]
\( \Gamma^*_l \)  circulation coefficient of vortex \( l \)  \[ \text{[m}^2\text{/s]} \]
\( \gamma \)  isentropic exponent
\( \Delta \epsilon \)  wake deficit function
\( \delta^{0.99} \)  boundary layer height  \[ \text{[m]} \]
\( \delta_{ij} \)  Kronecker symbol
\( \epsilon \)  mixing fraction
\( \epsilon_T \)  temperature mixing fraction
\( \epsilon_U \)  velocity mixing fraction
\( \epsilon \)  mixing fraction
\( \epsilon_m \)  mass flow error  \[ \text{[%]} \]
\( \eta \)  wall adiabatic film-cooling effectiveness
\( \kappa \)  gas conductivity  \[ \text{[W/mK]} \]
\( \kappa_{vK} \)  von-Karman constant
\( \Lambda \)  eigenvalue matrix
\( \Lambda_j \)  nearest wall point to vortex \( j \)  \( (j = 1, 2) \)
\( \Lambda \)  semi-axes of small ellipse (toroidal shape)
\( \Lambda_m \)  mean semi-axes of small ellipse (toroidal shape)
\( \lambda \)  non-dimensional parameter
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>μ</td>
<td>molecular (dynamic) viscosity</td>
<td>[kg/ms]</td>
</tr>
<tr>
<td>ν</td>
<td>kinematic viscosity</td>
<td>[m²/s]</td>
</tr>
<tr>
<td>ν₂</td>
<td>2&lt;sup&gt;nd&lt;/sup&gt; order numerical (artificial) viscosity</td>
<td>[m²/s]</td>
</tr>
<tr>
<td>ν₄</td>
<td>4&lt;sup&gt;nd&lt;/sup&gt; order numerical (artificial) viscosity</td>
<td>[m²/s]</td>
</tr>
<tr>
<td>Ω&lt;sub&gt;ji&lt;/sub&gt;</td>
<td>position of image vortex ji (j = 1, 2)</td>
<td>[m]</td>
</tr>
<tr>
<td>Ω&lt;sub&gt;jr&lt;/sub&gt;</td>
<td>position of real vortex jr (j = 1, 2)</td>
<td>[m]</td>
</tr>
<tr>
<td>Ω</td>
<td>reduced frequency</td>
<td></td>
</tr>
<tr>
<td>Ω(ξ, χ)</td>
<td>toroidal shape sub-function</td>
<td></td>
</tr>
<tr>
<td>ω</td>
<td>vorticity vector</td>
<td>[1/s]</td>
</tr>
<tr>
<td>ω*</td>
<td>non-dimensionalized vorticity</td>
<td></td>
</tr>
<tr>
<td>Φ</td>
<td>vector of linearized characteristics perturbation</td>
<td></td>
</tr>
<tr>
<td>φ</td>
<td>any flow quantity</td>
<td></td>
</tr>
<tr>
<td>φ&lt;sub&gt;ω&lt;/sub&gt;</td>
<td>absolute vorticity flux</td>
<td>[m²/s]</td>
</tr>
<tr>
<td>ρ</td>
<td>density</td>
<td>[kg/m³]</td>
</tr>
<tr>
<td>σ</td>
<td>computational overhead</td>
<td>[%]</td>
</tr>
<tr>
<td>σ&lt;sub&gt;K&lt;/sub&gt;</td>
<td>turbulent diffusion coefficient</td>
<td></td>
</tr>
<tr>
<td>σ₁</td>
<td>stretching magnitude of vortex l</td>
<td>[1/s]</td>
</tr>
<tr>
<td>τ&lt;sub&gt;ij&lt;/sub&gt;</td>
<td>shear stress component</td>
<td>[kg/ms²]</td>
</tr>
<tr>
<td>θ</td>
<td>any angle</td>
<td>[rad]</td>
</tr>
<tr>
<td>θ</td>
<td>non-dimensionalized temperature</td>
<td></td>
</tr>
<tr>
<td>θ&lt;sup&gt;f&lt;/sup&gt;</td>
<td>freestream boundary layer momentum thickness</td>
<td>[m]</td>
</tr>
<tr>
<td>θ&lt;sub&gt;p&lt;/sub&gt;</td>
<td>pitch angle</td>
<td>[rad]</td>
</tr>
<tr>
<td>(ξ, χ, η)</td>
<td>coordinates in the jet intrinsic frame of reference</td>
<td>[m]</td>
</tr>
</tbody>
</table>

**Subscripts**

- <sub>aw</sub>: adiabatic wall
- <sub>b</sub>: (for i,j or k = 3) bi-tangential to wall
- <sub>fc</sub>: with film-cooling
- <sub>ho</sub>: intersection point of the big circle and hole exit plane
- <sub>I</sub>: inviscid
- <sub>i</sub>: indices of the cartesian coordinates
- <sub>i</sub>: inner turbulent layer
- <sub>inl</sub>: inlet of the computational domain
- <sub>J</sub>: pertain to jet
- <sub>j</sub>: indices of the cartesian coordinates
$jc$  jet center at plane of injection
$ji$  pertain to image vortex $i$  ($j = 1, 2$)
$jr$  pertain to real vortex $r$  ($j = 1, 2$)
$jw$  pertain to wall point $w$  ($j = 1, 2$)
IBC  immersed boundary condition at jet surface
j  indices of the cartesian coordinates
le  hole leading edge
lm  intersection big circle and hole LE radius
m  mixing zone
m  mirror point
max  maximum value
min  minimum value
noc  no film-cooling
n  (for $i,j$ or $k = 1$) normal to wall
n  any grid node surrounding a mirror point
o  outer turbulent layer
out  outlet of the computational domain
p  ghost node
$\phi$  pertain to any flow quantity
ref  reference value
rec  recovery
rms  root-mean square
s  jet surface point
t  (for $i,j$ or $k = 2$) tangential to wall
tc  center of toroidal shape big circle
V  viscous
w  wake zone
wall  at wall
wi  jet windward side upper limit at plane of injection
x  (for $i,j$ or $k = 1$) cartesian component
y  (for $i,j$ or $k = 2$) cartesian component
z  (for $i,j$ or $k = 3$) cartesian component
$\theta$  tangential component
$\xi$  (for $i,j$ or $k = 1$) jet frame of reference
$\chi$  (for $i,j$ or $k = 2$) jet frame of reference
$\eta$  (for $i,j$ or $k = 3$) jet frame of reference
Superscripts

0 initial guess
a pertain to sound property
c pertain to coolant property
f pertain to freestream property
G global
J pertain to jet property
L local (jet region)
L laminar
l pertain to vortex l (l = 1r, 2r, 1i, 2i)
n time step
n power exponent
p power exponent
T turbulent

Abreviations

CFD Computational Fluid Dynamics
CFL Courant-Friedrichs-Levy
CGCC Cartesian Grid Cut Cell
CPU Central Processing Unit
CVP Counter-rotating Vortex Pair
DNS Direct Numerical Simulation
EXP Experimental
FCM Film Cooling Model
FI Full Injection
FVM Finite Volume Method
GGR Global Grid Reshaping
IBM Immersed Boundary Method
JICF Jet In Cross Flow
LES Large Eddy Simulation
NGV Nozzle Guide Vane
OG Overlapping Grid
RANS Reynolds-Averaged Navier-Stokes
TET Turbine Entry Temperature
WI Wall Injection
Chapter 1

Introduction

1.1 Film-cooling in support of the improvement of energy conversion systems

The need for energy conversion devices at a large scale

With the paddle of the 21st century, energy supply is more than ever a major technical, economical and political stake. Since the second world war, the continuous growth of the world population and economy, coupled recently with the entrance in the free market economy of important developing countries, have significantly increased the energy consumption of today and of the foreseeable future. In this context, energy conversion technologies are a fundamental part of the new global village. The energy conversion technologies are fortunately diverse, regarding their type of energetic transformation process, their total capacity of energy production and their exergetic value. The diversification of energy production techniques allows to face foreseen and unforeseen local or global events that change a human society. In this respect, the development of renewable energy technologies (wind, solar, hydraulic, biomass, geothermal, etc.) are highly greeted. Meanwhile, the conversion of primary fuels such as oil, natural gas, coal and nuclear into electrical, thermal or mechanical energy is still by far the major mean that can sustain the energy supply in regards of the rapidly growing world energy demand. Large, medium and micro fossil-fuel-based power plants, for electrical and heating systems, as well as combustion-based engines for transportation systems, have still means to be improved and to be used at a large scale.

Gas turbine: a major energy conversion device

One of the main characteristics required for an energy conversion engine is to deliver usable power in a continuous way. To this effect, turbomachines are the most
1.1. FILM-COOLING IN SUPPORT OF THE IMPROVEMENT OF ENERGY CONVERSION SYSTEMS

Efficient and broadly used engines conceived by humans. Turbomachines are essentially composed of rows of rotating (rotor) and non-rotating (stator) blades. The work exchange resulting from the dynamic interaction of the fluid flowing through a turbomachine (in particular through the rotating blades) is the key element of this type of energy conversion device. The idea of this engine is very ancient. It goes back to the Roman civilization where peasants were using "paddle-type" water wheels for grinding grain. During the industrial revolution in the 19th century, the use of turbomachines started to spread, favored by new disposabilities of fossil fuels at a large scale. There are many different types of turbomachines, see Lakshminarayana [53]. They can be broadly classified in three groups, that are namely hydraulic machines (water-based), steam turbines (nuclear-fission and coal-fired based) and gas turbine (natural gas and oil-fired based). Gas turbines are widely used for transportation systems, such as aircraft engines (see Fig. 1.1), where the propulsive work created by the exhaust gas is the main objective, see Hill and Peterson [44]. There is also a large use of gas turbines for producing electricity at different power output scales (typically from 100 [kW] to 100 [MW]). Gas turbines are basically composed of three main devices, see Fig. 1.2.

![Schematic of the GE90-94B turbofan engine.](image)

*Figure 1.1: Schematic of the GE90-94B turbofan engine. Its diameter is about 3.4 [m] and its length is about 7.3 [m]. It can deliver a thrust of 416 [kN] at sea level. HP denotes High Pressure, LP denotes Low Pressure. C denotes Compressor and T denotes Turbine.*
working gas, such as air, is first sucked into a compressor where it is pressurized. Then, the pressurized gas is guided through a combustion chamber where the fuel
(primary energetic source) is burnt so that the temperature (enthalpy) of the working gas is increased. Eventually, the hot (high enthalpy) gas enters a turbine where it is expanded. This is where it exchanges its energetic content by making the rotor blade disks rotate. The rotative motion of the turbine disks entrains the compressor disks. For aircraft engines, the remaining energetic content of the working gas at the exhaust of the turbine is used in terms of propulsive work.

The need of high gas temperature for better thermal efficiency

The engineering of such a sophisticated energy conversion device is a key to improve its efficiency. Improving the efficiency of an engine leads to lower economical and environmental costs. In particular, it is wished to have the Specific Fuel Consumption (SFC - amount of fuel burnt per unit thrust for an aircraft engine) to be as low as possible. To meet this objective, a high thermal efficiency (fraction of thermal energy actually transformed in mechanical work) is required. One of the most efficient ways to improve the thermal efficiency is to increase the Turbine Entry Temperature (TET - mean stagnation temperature of the working gas at the outlet of the Nozzle Guide Vane (NGV) of the turbine). As an example, an increase of the TET by 100 [K] rises the thermal efficiency of about 5 - 7%, which is quite substantial. In 1939, one of the first industrial gas turbine built had a TET of about 830 [K]. During the next three decades, the TET has approximatively increased by 10 [K/year] to reach a value of about 1200 [K] in the late 1960’s [76]. Although large efforts have been made to get better thermal, mechanical and corrosion resistivity of blade materials at high temperature, there is no such material that can withstand a TET of more than 1200 – 1400 [K] long enough, in terms of the standard blade lifetime needed for commercial use.

Internal versus external air cooling

Since materials technology reached a dead-end stage in the 1960’s for blade thermal management\(^1\), air-cooled turbines have been commercially introduced at that time. Using diverse cooling techniques of the turbine parts, the TET has been increased up to more than 2000 [K] in some of the most advanced propulsive systems existing in the 2000’s. As such, the cooling of turbine material parts located just at the outlet of the combustion chamber is one of the few key elements (maybe

\(^1\)Nowadays, materials technology is still a large research topic in turbomachinery, especially for blade mechanical and thermal stresses, manufacturing, etc...
with engine control and combustion) that adds most of the economical and environmental value of modern gas turbine design. Many types of cooling systems have been developed, tested and utilized. For example water cooling, steam cooling, liquid metal cooling and air cooling. In aerospace applications, air cooling is by far the most used technique. This thesis deals with air-based cooling systems. All the existing air cooling techniques necessitate to lay out cold air within the turbine parts. For this purpose, cold air is first bled from the compressor or less often from the combustion chamber outer annulus (up to 20% of the working gas mass flow can be withdrawn for cooling). Then, it is bypassed through internal channels and directed toward the turbine NGV and downstream rotors. Air-based cooling systems can broadly be classified in two different types, see LeGrives [55] and Fig. 1.3.

- **Internal cooling**
  - *Convection cooling*: cold air flows through internal passages of the blade material and is ejected at the blade trailing edge or blade tip. The blade material is cooled due to the forced convection occurring between the coolant and internal walls.

![Schematics of different air-based cooling systems.](image)

*Figure 1.3: Schematics of different air-based cooling systems.*
1.2. Modeling multi-scale flows to improve design

- Impingement cooling: cold air is injected through several inserts inside the blade. It eventually hits the internal blade walls. A reduction of the blade wall temperature is obtained.

- External cooling
  - Film cooling (full coverage): on the turbine blade surface (and also on other turbine components), small holes are drilled, of a diameter typically of 0.5 – 1.0 [mm] (100 times smaller than the blade chord), see Fig. 1.4. Through these small cooling holes, cold air is injected into the mainflow, in the form of a coolant jet. The goal is that the coolant jet forms a thin layer shielding the blade surface from the hot gas.
  - Transpiration cooling: as this is the most recent cooling technique, it is still under development. The idea is to have a porous blade material. The coolant passes through the small pores to create a thin cold layer onto the blade external surface.

Historically, internal cooling techniques were the first to be applied. However, internal cooling is less efficient than external cooling. Indeed, in internal cooling, the external side of the blade is only moderately cooled. Internal cooling can be used for a Turbine Entry Temperature (TET) of no more than 1600 [K]. External cooling, especially film cooling, is now largely used for turbine cooling. The most advanced film cooling technique allows the external blade walls to withstand a TET of around 2000 [K]. In a film cooling system, the better the film-cooling holes are shaped and placed in the blade surface, the better the heat protection of the blade is, so that the TET can be increased. Thus, the design of film-cooled turbine blades sets the entire turbo-engine thermal efficiency.

1.2 Modeling multi-scale flows to improve design

Designing film-cooled turbine blade

The design of turbine cooling systems can either focus on the internal coolant channels and/or the cooling holes at the blade surface. This thesis focuses on the flow physics occurring nearby the cooling holes. It is therefore assumed that the flow conditions of the coolant within the internal channels are known. The design of a cooling hole is a large topic in itself. There is a large number of different hole geometries that have been investigated, see for instance Hyams and Leylck
1.2. MODELING MULTI-SCALE FLOWS TO IMPROVE DESIGN

The simplest hole geometry is an inclined cylindrical tube. Shaping the hole length and profile has shown to be beneficial in some cases but there are still no standard rules. This thesis focuses only on cylindrical holes. In some situations, more than 300 holes per blade can be used in order to ensure an efficient cooling of the blade walls. However, increasing the total coolant mass flow leads to an increase of the losses induced by the cooling system, namely the mass flow deficit at high enthalpy (fluid extracted from the compressor) and mixing losses due to the
coolant jet-mainstream interaction. The designer has therefore to carefully balance the aero thermodynamics of the flow through the film-cooled turbine passage, often highly loaded and transonic, to maximize thermal efficiency. The measure of a good thermal protection of blade material by film-cooling can be done by several means. One of the most used value is the adiabatic film-cooling effectiveness $\eta$. It scales the difference of temperature experienced at the surface of a film-cooled versus a non-film-cooled blade. The blade wall is considered adiabatic in this case.

$$\eta = \frac{T_{aw} - T_{rec}}{T_c - T_{rec}}$$

(1.1)

where $T_{aw}$ is the temperature obtained at the adiabatic wall with the presence of a film-cooling jet, $T_{rec}$ is the recovery temperature\(^2\) at the same wall without film-cooling and $T_c$ is the coolant total temperature before emerging out of the hole. Another important value is the heat transfer rate $q'_w$ occurring at the film-cooled isothermal blade wall.

$$q'_w = -\kappa \left[ \frac{\partial T}{\partial n} \right]_{wall}$$

(1.2)

where $\kappa$ is the gas conductivity and $n$ is the normal direction of the blade wall surface. The interaction between the coolant jet and freestream flow is commonly scaled by two ratios. There is first the Density Ratio $DR$ which scales the coolant density $\rho_c$ to the freestream fluid density $\rho_f$.

$$DR = \frac{\rho_c}{\rho_f}$$

(1.3)

This ratio gives a sense of the temperature difference between the coolant and the freestream flow. The second scaling is given by the Blowing Ratio $BR$ which scales the coolant to freestream mass flux.

$$BR = \frac{\rho_c U_c}{\rho_f U_f}$$

(1.4)

where $U$ represents the flow velocity. A high blowing ratio ($BR \gg 1.0$) means a strong coolant jet and a low blowing ratio ($BR \ll 1.0$) means a weak coolant jet.

---

\(^2\)The recovery temperature is always smaller or ideally equal to the nearby total freestream temperature. Indeed, the deceleration of the freestream flow in the boundary layer experienced friction losses (ideally it should be an isentropic process).
1.2. MODELING MULTI-SCALE FLOWS TO IMPROVE DESIGN

In general, the designer needs to adjust the freestream and coolant flow conditions in order to get a density and blowing ratio that lead to the highest film-cooling effectiveness and the lowest heat transfer rate. In the meantime, the cost function, that is the absolute loss of work exchange through the turbine, has to be minimized.

**Sensitivity of metal temperature**

The fatigue of the blade material strongly depends on the surface metal temperature. The designer should ensure that the surface metal temperature is kept below a certain value (with a security factor) during the life of the turbine. When a novel film-cooled turbine blade is commercialized, it should be ensured that the tools utilized for its design, model or predict film-cooling effectiveness and/or surface heat transfer within a specified range of accuracy. Indeed, as an example, an error of $\Delta \eta = 0.1$ in the modeling or prediction of the film cooling effectiveness leads to an error of $\Delta T = 0.1 (T_T^c - T_{rec})$. Knowing that the difference of temperature between the coolant and the freestream can be much higher than 200 [K], this leads to an error of at least 20 [K] in the modeling or prediction of the surface metal temperature. At these high temperatures, an increase of 20 [K] of the surface metal temperature can lead a reduction of 50% of the blade lifetime. This shows how critical it is to have accurate tools for the design of film-cooled turbine parts.

**Computational Fluid Dynamics (CFD) in support of design**

Overall, in order to perform a good aerothermodynamics design optimization, a parametric study of the different flow conditions and geometrical dimensions typical of a film-cooled turbine blade has to be carried out. Thus, the designer needs a design tool that is, at the same time:

- **Accurate, to ensure a relevant design result.**
- **Computationally efficient, to shrink the turnover time, hence design cost.**

It is the intention in this thesis to propose a novel numerical tool that can be used to design film-cooled turbine blades and also to investigate flows through them in a reasonable time scale. It takes the form of a film-cooling jet model that can be embedded in a standard Computational Fluid Dynamic (CFD) code. In fact, as computer Central Processing Unit (CPU) power is almost doubling every 18 months, CFD becomes a powerful tool on which the design and optimization of film-cooled turbine blades can be based. Numerical methods that solve the Navier-Stokes equations of fluid motion are nowadays accepted as tools for investigating
the fluid dynamics occurring in turbine flows. However, depending on the flow problem to be solved, there is still room for improvement of numerical methods and physical models to gain accuracy and reduce computational costs.

**Film-cooling: a multi-scale flow phenomenon**

In order to create an efficient numerical tool for computing, investigating and designing film-cooled turbine blades, it is first essential to understand the fluid dynamics underlying film-cooling flow in turbines (see Chapter 3 for an in-depth review of such flows). Film-cooling jets are typical of a broadly studied flow, that is Jet In Cross Flow (JICF). The jet in cross flow phenomenon in turbines is multi-scale by nature. The coolant jet, coming out of the hole, gradually mixes with the freestream (hot gas) flow with a characteristic length scale of the hole diameter. In contrast, the aerodynamics of the turbine passage flow, e.g. passage vortex, has a much larger length scale. These scales may differ by up to two orders of magnitude. Thus, in order to carry out a relevant design, it is necessary to understand and model the fluid dynamics occurring at these two flow scales. This is a major challenge since the interaction of several flow scales leads to very complex flow phenomena, turbulence in fluid being the best example. A relevant numerical prediction of such flows needs therefore to be able to reveal and/or model accurately the smallest length scale, that is the length scale of the coolant jet.

### 1.3 Literature Review

Many researchers have numerically studied film-cooling jets using different numerical methods. A chronological literature review of few selected relevant studies, going from film-cooled flat plate to film-cooled turbine blade configuration is done below. Indeed, it is intended to give the reader a comprehensive review on the state of the art of the best numerical approaches when dealing with film-cooling jet flows. Light will ultimately be shed on the approach chosen in this thesis for the modeling of a film-cooling jet.

**Previous numerical approaches for solving film-cooling flow: film-cooled flat plate configurations**

Bergeles et al. [11] used a partially parabolic scheme with an anisotropic coefficient associated with the turbulent stress term to compute an isolated jet. The coolant velocity was specified at hole exit and the grid contained about $10^4$ grid
1.3. LITERATURE REVIEW

The accuracy of the prediction for blowing ratio higher than 0.5 was found to be strongly linked to the type of velocity profile specified at hole exit, particularly as the jet starts to lift off. The conclusion is that extending the computational domain inside the coolant hole itself may improve the accuracy of the prediction. Demuren et al. [25] performed a large series of computations of a jet issuing normally to the surface with different grid refinement, to have up to 15 grid nodes per hole diameter. It was concluded that the grid spacing near the hole needs to be very fine to accurately resolve the jet bending at hole exit, which has a strong impact on the downstream coolant flow evolution. Leylek and Zerkle [56] presented a full three-dimensional Reynolds-Averaged Navier-Stokes (RANS) computation of a film cooling jet in a flat plate. The plenum chamber, coolant hole and cross-flow region were modeled. The computation was carried out using a wall function procedure for the near wall treatment, requiring a grid of $2 \cdot 10^5$ nodes in total. About 15 grid nodes per hole diameter in streamwise and lateral direction were used. It was found that the coolant flow inside the pipe is already complex and has a strong influence on the downstream evolution of the jet and on the film cooling effectiveness. Furthermore, the velocity profile at hole exit is a strong function of the blowing ratio. In the continuation of this work, Walters and Leylek [90] did the same type of computation but they increased the mesh size up to $6.2 \cdot 10^5$ grid nodes. The mesh was essentially densified in the boundary layer region in order to use a two-layer, $k - \epsilon$ type of turbulence model. Using this strategy, the computational cost is significantly increased (no quantification is given) but the near-hole flow field prediction greatly differs compared to a calculation using wall function. In particular, the flow field at the hole exit exhibits a much more complicated flow field, in particular with a tiny recirculating zone at the trailing edge of the hole. This results in a better prediction of film-cooling effectiveness downstream of the hole. More recently, Lakehal [52] presented a comparative study of different RANS-based turbulence modeling strategies for solving a film-cooling jet issuing from a hole inclined at an angle of $35^\circ$ in a flat plate. The mesh contained about $4.3 \cdot 10^5$ grid nodes, with more than 150 cells within the hole exit area. The computational mesh included both the plenum chamber and hole. The conclusion of the study is that the right modeling of the anisotropic turbulent mixing between the coolant jet and freestream is essential to accurately predict the lateral spreading of the jet. Meanwhile, it is worth using a highly complex turbulent model only if the mesh is carefully refined in the region of the jet. In the last 10 years, few researchers (see for instance Yuan et al. [94], Wegner et al. [92], Muldoon and Acharya [69]) have started to use Large-Eddy Simulation (LES) and even Direct Numerical Simulation (DNS) methods to investigate film-cooling jets.
These methods are very helpful to gain insight of the fluid dynamics occurring in such flows, and also to support turbulence model development. Meanwhile, due to the fact that the unsteady flow field must be computed, the computational cost is still much greater than for RANS-based methods. More flow structures, at smaller scales, are revealed using these methods but the gain in accuracy versus computational cost in the prediction of flow quantities relevant for the designer, such as film-cooling effectiveness and/or heat transfer level, is still uncertain.

**Previous numerical approaches for solving film-cooling flow: film-cooled turbine blade configurations**

Garg and Gaugler [32] presented a study of the effect of the coolant velocity and temperature distribution at hole exit on the heat transfer coefficient on three different film-cooled turbine blades (namely the C3X vane, VKI rotor and the ACE rotor). For each turbine case, a mesh was used covering only the mid-span region. This mesh had about $1 \times 10^6$ grid nodes, with about 20 cells per hole. The calculation did not start inside the cooling hole, instead the velocity and temperature profile at hole exit was defined using a 6\textsuperscript{th} order polynomial function ensuring mass flow and energy conservation. It was found that the predicted heat transfer coefficient depends strongly on the imposed velocity and temperature profile at hole exit. Therefore, a proper definition of the boundary condition at hole exit is of primary importance to have a good prediction, but it is a complex function of many variables. Garg and Abhari [33] did a comparative study of measured and predicted Nusselt number at the surface of a rotating turbine blade (ACE). The computational mesh included the entire blade span. It contained about $2.28 \times 10^6$ grid nodes in total and each of the 93 cooling holes was covered by 10 to 20 computational cells. The plenum chamber and hole were not included in the computational mesh. The film injection was specified through a uniform velocity and temperature profile at the location of hole exit. On the pressure surface, an underprediction of the Nusselt number was found and in the suction surface, a reasonably good prediction of the Nusselt number was obtained. More recently, Garg and Rigby [34] and Heidmann et al. [42] presented a numerical investigation of a film-cooled turbine blade where the hole and plenum chamber were gridded. The complexity of the meshing was underlined, since a multitude of different grid blocks (up to 2300!) needs to be connected together to fully model the interior and exterior of the film-cooled blade, see Fig. 1.5. Due to the complexity of the problem, in terms of mesh size and topology, only the midspan section was computed, with symmetry boundary condition at the bottom and top side of the computational domain. For
1.3. LITERATURE REVIEW

Figure 1.5: Example of the complexity of a 3D grid needed to solve simultaneously the flow through a film-cooled turbine passage as well as in the holes and plenum chamber (Heidmann et al. [42]).

only computing 2 – 3 holes per row (in the midspan section), a mesh size up to $1.6 \cdot 10^6$ was used [42]. Although very detailed qualitative predictions of the flow field were found, it is still unclear if the gain in accuracy for predicting wall heat transfer is relevant, compared to a numerical approach that does not include the plenum and hole.

The need for a film-cooling model

It is evident from the above cited literature that the meshing of the film-cooling hole and plenum is a major strategic decision. Due to the multi-scale nature of the flow to be investigated, a balance has to be struck between accuracy on one hand and grid size and run time on the other. In particular, with hundreds of holes used on some turbine components, grids of hundreds of millions of nodes may be required. Furthermore, the complexity arising from the meshing can be a real bottleneck. In order to solve these issues, various authors have proposed the application of film-cooling models within CFD. A literature review of former research carried out in this area is provided below.
1.3. LITERATURE REVIEW

Film-cooling model in two-dimensional codes

Crawford et al. [22] proposed an injection jet model that can be implemented in a two-dimensional boundary layer code. Essentially, the coolant fluxes are derived by a one-dimensional momentum and energy balance and a lateral augmentation of the eddy viscosity is given to account for the lateral mixing. They have shown good prediction of Stanton number but only for blowing ratio below 0.8. Schöning and Rodi [81] presented an injection model that can also be used in a two-dimensional boundary layer code. The strategy was to specify a new boundary layer profile just after the near hole elliptic region. The profile is given using a one-dimensional momentum balance of the incoming boundary layer and coolant fluxes. In order to reproduce the lateral entrainment of the hot fluid, which enhances the mixing, a dispersion model term was added to the equations solved, acting as a source term. Haas et al. [40] extended this model to take account of the density ratio between the hot and cold fluid. The model showed a good prediction capability in a two-dimensional environment, with one row of holes. Tafti and Yavuzkurt [87] developed another two-dimensional injection model in which the coolant fluid trajectory and penetration profile were based on semi-empirical results and observations. The three-dimensional mixing is accounted for through an "entrainment fraction", correlated to experimental observation. The model was successful in predicting film cooling in a two-dimensional multirow environment. Abhari [2] extended the model to be used in a two-dimensional environment by specifying a plane jet after hole exit with CFD. The entrainment fraction was modified to take the effect of density ratio. The model was shown to be well suited for the prediction of heat transfer on film-cooled turbine blades, except in the region immediately downstream of the injection rows. Kulisa et al. [51] used a two-dimensional boundary layer code coupled iteratively with a three-dimensional approach to compute the coolant jet. The three-dimensional jet behavior was described using an integral formulation of the equations of conservation over the jet cross-section. The resultant jet flow field was incorporated in the two-dimensional boundary layer computational procedure using a source term approach. Good prediction of the coolant jet evolution was shown but only in the center plane; the freestream fluid entrainment was not incorporated, i.e. no model of the counter-rotating pair was proposed. It was emphasized that agreement with experimental observation is strongly dependent on the jet behavior near hole exit. Very recently, Forest et al. [30] proposed a three-dimensional mixing model that includes the effect of the counter-rotating vortex pair through a source term, to be coupled to a two-dimensional boundary layer code. The model coefficients were cleverly tuned.
1.4. RESEARCH OBJECTIVES

to experimental data obtained in a turbine blade cascade. The prediction of heat transfer on the suction side surface of a blade was reasonably good, however a large discrepancy was found on the pressure side surface.

Film-cooling model in three-dimensional RANS codes

Very few attempts have been made to include a fully three-dimensional film-cooling jet model within a CFD code. As the only attempt known by the author, Dalhander et al. [23] proposed a three-dimensional source term approach to take into account the effect of the coolant fluid on the boundary layer flow. The introduction of the source term is not only done at the injection site but also downstream to control the evolution of the mixing process. The injection model ensures mass flow and energy conservation but the downstream source terms showed to be very sensitive to the flow case simulated. The direct control of the flow a long distance downstream of the injection site may be very tedious. Using this method, significant deviations of the predicted film-cooling effectiveness compared to experimental data were found. In addition, the three-dimensional coolant jet flow field was not revealed.

1.4 Research objectives

The main objective of the current research can be summarized by the following question:

- Given a flow through an energy conversion engine having two main flow scales, how could we model them in order to be able to predict the consequence of their interaction upon the global performance of the engine in a reasonable time scale?

Obviously, this thesis is certainly not solving once and for all the above question, but it tentatively tries to give a relevant strategy leading to a solution. In connection to the above question, the particular flow case to be studied is a collection of film-cooling jets issuing through a turbine blade. The goal is to propose a novel three-dimensional jet model that can be used by a designer using a CFD code for his task. Incidentally, the model should also be usable for further research concerning more generally jets in turbine flows. To the author’s knowledge, this is the first time that such a model is proposed. The potential of LES and DNS methods is great for the accurate resolution of such flows. Meanwhile, in terms of daily design
of film-cooled turbine blade, there is still a huge gap to be covered before some robust and fast methodologies are established when using them. It has therefore been chosen to base the model upon a RANS approach. This means that the model should reproduce the macro flow features pertaining to a jet in cross flow near the hole exit. It has been shown in the literature review that a relevant numerical prediction of a film-cooling jet necessitates the complex meshing of the plenum chamber, hole and blade passage. An order of magnitude of $10^6$ grid nodes are needed to accurately resolve the fluid dynamics of such a flow. This is why the model development is performed at the hole exit. Thus, it is wished to alleviate the problem of meshing inside the hole and internal channels. In connection to this, it is intended to propose a numerical technique that allows immersing the film-cooling jet model inside any RANS-based CFD code. The numerical immersion technique should allow to use the jet model in a mesh that is analog to those standard ones used for prediction of flows through blade passages. In order to ensure an accurate modeling, a calibration of the model coefficients with three-dimensional experimental data should be pursued. An in-depth performance analysis of the CFD-embedded film-cooling model, in terms of computational accuracy versus computational costs, is a must to show the relevancy of the proposed methodology. In addition, the model should be validated against a wide range of test cases, from film-cooled flat plate to film-cooled turbine blade configurations. These validation steps should, in parallel, serve to gain knowledge of the flow physics occurring in film-cooling jets. In summary, the detailed research objectives are:

1. Modeling of a film-cooling jet near the hole exit.
2. Development of a numerical technique to immerse the model in a CFD algorithm.
3. Calibration of the model coefficients using experimental data.
4. Analysis of the computational performance of the model.
5. Validation of the model in different geometries, from film-cooled flat plate to film-cooled turbine (including blind test cases).
6. Gain insight of the effect of hole geometry, location and coolant to freestream flow regime upon thermal protection.

A prerequisite to this research is to have a RANS-based CFD code at hand. To this purpose, a three-dimensional unsteady RANS code (so-called MULTI3 [15])...
1.5. THESIS OUTLINE

has been first developed and brought into operation. The code is based on the Ni-Lax-Wendroff algorithm coupled to a Finite-Volume Method (FVM) for space discretization. This is essentially an explicit, time-marching, second order accurate algorithm, typical of codes used for computing turbomachinery flows. The algebraic, two-layer, isotropic Baldwin-Lomax turbulence model is included in the code for the treatment of the turbulent field. Thus, the present numerical work is strictly done in the context of isotropic turbulence.

1.5 Thesis outline

• The technical and scientific reasons that lead to this thesis are given in Chapter 1. In particular, the need to have a numerical tool for predicting film-cooling jets in turbines in a reasonable time scale is explained. The literature review leads to the modeling strategy chosen. In this respect, a thoughtful research procedure is discussed at the end of the Chapter.

• Chapter 2 introduces the theoretical background for building a standard turbomachinery CFD code. The numerical issues linked to it, namely the governing equations of motion to be solved, computational algorithm, turbulence modeling, boundary conditions and code stability are shown.

• The development of a novel film-cooling jet model is described in Chapter 3. The selected flow features to be modeled are discussed, on the basis of experimental observations available in the literature. The mathematical derivations that express the modeling of a film-cooling jet are shown. Eventually, an overview of the film-cooling jet model is given in a flow chart.

• The numerical immersion of the model in a CFD code is explained in Chapter 4. It first discusses the strategy chosen, namely the Immersed Boundary Method (IBM). The general methodology of the numerical immersion is described and its application to the flow studied in this thesis is shown. The inputs needed for the model are explained. Eventually, an overview of the full CFD-embedded film-cooling jet is given in a flow chart.

• Chapter 5 is centered around the calibration of the model coefficient. For this purpose, the film-cooled flat plate experiment of Bernsdorf [13] is utilized. The calibration process is explained. A broad range of flow regimes is included in the model calibration. Thus, it is shown that the model can be used for different flow and geometrical configurations.
1.5. THESIS OUTLINE

• The model performance, in a steady state sense, is evaluated in Chapter 6. It is first shown that the use of the model in a CFD code leads to a stable and converging solution. The resulting mass flow error is quantified to be small enough. A study of the computational accuracy versus computational cost is pursued in order to propose an optimal use of the CFD-embedded film-cooling jet model.

• Chapter 7 deals with the steady aerodynamic validation of the model. The predicted velocity and vorticity fields for different flow regimes are compared to accurate experimental data. Discrepancies in the prediction are discussed. The effect of the hole injection angle and blowing ratio is investigated. Eventually, a comparison of the predictive capability of the model versus two other numerical injection strategies is shown in order to assess the merit of the modeling strategy chosen.

• In Chapter 8, a steady heat transfer validation of the model is performed. Experimental data found in the literature serve as reference (blind test cases). The discrepancies in the prediction of film-cooling effectiveness are explained, in particular the isotropic treatment of the turbulence. The model is also tested for coolant flow cases having a high lateral injection component. This study allows to propose further developments of the model to increase its range of application.

• As flows in turbine are inherently unsteady, in a periodic mode, it is shown in Chapter 9 that the model can also be used when the flow field surrounding the hole is pulsating. The type of unsteadiness generally occurring in turbines is first discussed. Based on this analysis, a step toward using the model in such flow situations is made. The computational cost is shown to be low when used in an unsteady mode.

• In order to show that the model can effectively be used to predict flows through film-cooled turbine blade passages, Chapter 10 presents a computation in such a configuration. Up to 67 cooling holes are taken into account in the prediction of the flow field of a film-cooled transonic turbine cascade. The computational cost is low, compared to a calculation without any cooling. The effects of the blowing ratio and the spatial location of the cooling rows of holes are discussed.

• Chapter 11 concludes by pointing to the key results obtained in this thesis. In
addition, an outlook for further model development and utilization is eventually given.
Seite Leer /
Blank leaf
This Chapter introduces the numerical method used to solve the governing equations of fluid motion. Based on the numerical method presented herein, a CFD code, so-called MULTI3 (see Burdet and Lakehal [15]), has been coded. This CFD solver serves as the main tool for the numerical implementation and validation of the film-cooling jet model presented in the next Chapters. The numerical algorithm employs a Ni-Lax-Wendroff approach in the context of the Finite-Volume Method (FVM). The stability analysis and artificial smoothing of the numerical scheme is discussed in this Chapter, as well as the handling of the diverse boundary conditions to be imposed.

2.1 Governing equations

2.1.1 The three-dimensional Reynolds-Averaged Navier-Stokes equations

In this thesis, the three-dimensional compressible Reynolds-Averaged Navier-Stokes (RANS)\(^1\) equations are numerically solved in order to study flows through gas turbines. Any flow quantity \(\phi\) can be decomposed into two parts, by two means. For density \(\rho\) and pressure \(P\), the following Reynolds decomposition is used

\[
\phi = \bar{\phi} + \phi'
\]

(2.1)

where \(\bar{\phi}\) is the mean averaged value and \(\phi'\) is the fluctuation. For the velocity and energy fields, the following Favre decomposition applies

\[
\phi = \tilde{\phi} + \phi''
\]

(2.2)

\(^1\)Since the compressibility effect needs to be taken into account, fluctuation of density and temperature must be taken into account. Hence, a Favre-averaging procedure (see Wilcox [93]) is performed for the velocity and energy fields.
2.1. GOVERNING EQUATIONS

where $\tilde{\phi}$ is the mean mass-averaged value and $\phi''$ is the fluctuation. The averaging reads

$$
\tilde{\phi} = \lim_{T \to \infty} \frac{1}{T} \int_{t}^{t+T} \phi(\zeta) d\zeta
$$

(2.3)

$$
\bar{\phi} = \frac{1}{\rho} \lim_{T \to \infty} \frac{1}{T} \int_{t}^{t+T} \rho(\zeta) \phi(\zeta) d\zeta
$$

(2.4)

The following relations apply [93]

$$
\bar{\phi}'' = 0 ; \quad \rho \phi'' = 0
$$

(2.5)

In the following, each flow quantity is expressed in terms of its averaged value. The RANS equations model the fundamental laws of mass conservation (Eq. 2.6), momentum conservation (Eq. 2.7) and energy conservation (Eq. 2.8).

$$
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \tilde{u}_i)}{\partial x_i} = 0
$$

(2.6)

$$
\frac{\partial \rho \tilde{u}_i}{\partial t} + \frac{\partial (\rho \tilde{u}_i \tilde{u}_j)}{\partial x_j} = - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \tau_{ij}^L - \rho u''_i u''_j \right) + B_i
$$

(2.7)

$$
\frac{\partial}{\partial t} \left( \rho \tilde{E} + \frac{\rho u''_i u''_i}{2} \right) + \frac{\partial}{\partial x_j} \left( \rho \tilde{u}_i \tilde{H} + \tilde{u}_j \frac{\rho u''_i u''_j}{2} \right) = \frac{\partial}{\partial x_j} \left[ \tilde{u}_i \left( \tau_{ij}^L - \rho u''_i u''_j \right) \right]
$$

$$
- \frac{\partial}{\partial x_j} \left[ \tilde{q}_j^L - \tau_{ij}^L u''_i + \rho u''_j H'' + \rho u''_j \frac{1}{2} u''_i u''_i \right]
$$

(2.8)

where $\tilde{u}_i$ are the cartesian components of the velocity vector ($i = x, y, z$), $\tilde{E}$ is the total energy per unit mass, $\tilde{H}$ is the total enthalpy per unit mass, $\tau_{ij}^L$ are the cartesian components of the laminar stress tensor, $\tilde{q}_i^L$ are the cartesian components of the laminar heat flux vector and $B_i$ are the cartesian components of a body force acting on the fluid. From now on, signs of averaging are omitted for clarity. The turbulent kinetic energy, denoted by $\rho k$, is defined as

$$
\rho k = \frac{1}{2} \rho u''_i u''_i
$$

(2.9)
The turbulent kinetic energy, up to the hypersonic range, can be considered negligible compared to the pressure \( P \) [93]. As flows of interest in this study are much lower than hypersonic range, the contribution of the turbulent kinetic energy is neglected. The laminar stress tensor \( \tau^L_{ij} \) and the Reynolds (turbulent) stress tensor \( \tau^T_{ij} = -\rho u^\prime_i u^\prime_j \) are defined through the Boussinesq model (turbulent kinetic energy is neglected).

\[
\tau_{ij} = \tau^L_{ij} + \tau^T_{ij} = 2 \left( \mu^L + \mu^T \right) \left[ \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{1}{3} \left( \frac{\partial u_k}{\partial x_k} \right) \delta_{ij} \right]
\]

(2.10)

where \( \mu^L \) is the laminar molecular viscosity and \( \mu^T \) is the eddy (turbulent) viscosity. The eddy viscosity \( \mu^T \) models the effect of the dissipation of the smallest flow structures. The laminar molecular viscosity \( \mu^L \) is found using the Sutherland's law. It is valid up to a temperature of 1500 [K].

\[
\mu^L = \frac{C_1 T^{\frac{3}{2}}}{C_2 + T}
\]

(2.11)

where \( T \) is the static temperature, \( C_1 = 1.458 \cdot 10^{-6} \) [kg/ms/\( \sqrt{K} \)] and \( C_2 = 110.4 \) [K]. To model the laminar heat flux vector \( q^L_i \), as well as the turbulent heat flux vector \( q^T_i = \rho u^\prime_j H^\prime \), Fourier's law is employed.

\[
q_i = q^L_i + q^T_i = - \left( \frac{\mu^L}{Pr^L} + \frac{\mu^T}{Pr^T} \right) \frac{\partial (c_p T)}{\partial x_i}
\]

(2.12)

where \( c_p \) is the specific heat at constant pressure, \( Pr^L \) is the laminar Prandtl number and \( Pr^T \) the turbulent Prandtl number. In general, \( Pr_L = 0.71 \) and \( Pr_T = 0.9 \). The turbulent molecular diffusion and transport are commonly modeled as follows

\[
-\rho u^\prime_j \frac{1}{2} u^\prime_i u^\prime_j + \tau^T_{ij} u^\prime_i = \left( \mu^L + \frac{\mu^T}{\sigma K} \right) \frac{\partial k}{\partial x_j}
\]

(2.13)

Since the turbulent kinetic energy is neglected, the effect of the turbulent molecular diffusion and transport is also neglected in this study [93].

### 2.1.2 Conservative form

In many situations, flows through gas turbines contain both subsonic and supersonic regions, due mainly to the presence of large pressure gradients. Therefore,
shocks can occur through blade passages. In order to satisfy the Rankine-Hugoniot conditions for shocks, the equations of motion have to be solved in a conservative form.

\[ \frac{\partial Q}{\partial t} = \left[ \frac{\partial F_I}{\partial x} + \frac{\partial G_I}{\partial y} + \frac{\partial H_I}{\partial z} \right] + \left[ \frac{\partial F_V}{\partial x} + \frac{\partial G_V}{\partial y} + \frac{\partial H_V}{\partial z} \right] + B \]  \hspace{1cm} (2.14)

where \( Q \) is the state vector, \( F_I, G_I, H_I \) are the inviscid fluxes, \( F_V, G_V, H_V \) are the viscous fluxes and \( B \) is a body force.

\[ Q = \begin{bmatrix} 
\rho \\
\rho u_x \\
\rho u_y \\
\rho u_z \\
\rho E 
\end{bmatrix} \hspace{1cm} (2.15) \]

\[ F_I = \begin{bmatrix} 
\rho u_x \\
\rho u_x^2 + P \\
\rho u_x u_y \\
\rho u_x u_z \\
\rho \dot{H} u_x 
\end{bmatrix}, \quad G_I = \begin{bmatrix} 
\rho u_y \\
\rho u_y u_x \\
\rho u_y u_y + P \\
\rho u_y u_z \\
\rho \dot{H} u_y 
\end{bmatrix}, \quad H_I = \begin{bmatrix} 
\rho u_z \\
\rho u_z u_x \\
\rho u_z u_y \\
\rho u_z u_z + P \\
\rho \dot{H} u_z 
\end{bmatrix} \hspace{1cm} (2.16) \]

\[ F_V = - \begin{bmatrix} 
0 \\
\tau_{xx} \\
\tau_{xy} \\
\tau_{xz} \\
\tau_{xx} u_x + \tau_{xy} u_y + \tau_{xz} u_z - q_x 
\end{bmatrix} \hspace{1cm} (2.17) \]

\[ G_V = - \begin{bmatrix} 
0 \\
\tau_{xy} \\
\tau_{yy} \\
\tau_{yz} \\
\tau_{xy} u_x + \tau_{yy} u_y + \tau_{yz} u_z - q_y 
\end{bmatrix} \hspace{1cm} (2.18) \]

\[ H_V = - \begin{bmatrix} 
0 \\
\tau_{xz} \\
\tau_{yz} \\
\tau_{zz} \\
\tau_{xz} u_x + \tau_{yz} u_y + \tau_{zz} u_z - q_z 
\end{bmatrix} \hspace{1cm} (2.19) \]
2.1. GOVERNING EQUATIONS

To characterize the state of the working gas, the perfect gas model is used. It relates the total energy per unit mass $E$ to the pressure $P$.

$$P = (\gamma - 1) \left[ \rho E - \frac{1}{2\rho} \left( (\rho u_x)^2 + (\rho u_y)^2 + (\rho u_z)^2 \right) \right]$$  \hspace{1cm} (2.20)

where the total energy per unit mass $E$ is given by

$$E = c_v T + \frac{u_x^2 + u_y^2 + u_z^2}{2c_p}$$  \hspace{1cm} (2.21)

where $T$ is the static temperature and $c_v$ is the specific heat at constant volume. The isentropic exponent $\gamma$ reads

$$\gamma = \frac{c_p}{c_v}$$  \hspace{1cm} (2.22)

For air at ambient temperature, it is of the order of $\gamma = 1.4$. In the following, the isentropic exponent is assumed to be constant. The specific heat at constant pressure $c_p$ is related to the gas constant $R_g$.

$$c_p = \frac{\gamma}{\gamma - 1} R_g$$  \hspace{1cm} (2.23)

If not explicitly specified, the working gas is air. The gas constant value for air that is used is 287 [J/kgK]. The total enthalpy per unit mass $H$ is a function of the total energy per unit mass $E$ and temperature $T$.

$$H = E + R_g T$$  \hspace{1cm} (2.24)

2.1.3 Non-dimensionalization

The RANS equations to be solved are used in practice in a non-dimensionalized form. This is due to computer limitation to represent a number. This allows all computed state variables to be about the same order of magnitude. The reference values are the dominant length of the problem $L_{ref}$ (e.g. blade axial chord for flow problem through a turbine), the inlet stagnation density $\rho_{ref}$ and the inlet stagnation speed of sound $a_{ref}$. Indeed, as flows through turbines significantly vary around the speed of sound $a_{ref}$, the speed of sound is a better scale of the velocity than the inlet freestream velocity, as opposed to external flow problems.

$$x^* = \frac{x}{L_{ref}} \hspace{1cm} y^* = \frac{y}{L_{ref}} \hspace{1cm} z^* = \frac{z}{L_{ref}}$$
2.1. GOVERNING EQUATIONS

2.1.4 Turbulence model

The zero-equation Baldwin-Lomax turbulence model, see Baldwin and Lomax [8], is chosen to compute the eddy viscosity $\mu^T$. The model employs a two-layer technique, in which the inner eddy viscosity $\mu^T_I$ is boundary layer affected and the outer viscosity $\mu^T_O$ scales the turbulent diffusion in the freestream. This turbulence model is isotropic, since the eddy viscosity $\mu^T$, at a given discrete location in the flow, has no preferred direction, i.e. it gets the same value for all the spatial directions. Essentially, the Baldwin-Lomax model has to be used in a so-called "profile line". The "profile line" is characterized by a line that leaves the wall surface orthogonally, in the "y-direction", going up to the freestream region. All the parameters needed for the model are computed in this profile line. Thus, this is a turbulence model that does not take into account the spatial and time history.

$$t^* = \frac{t}{L_{ref}/a_{ref}}$$

$$u_x^* = \frac{u_x}{a_{ref}} \quad u_y^* = \frac{u_y}{a_{ref}} \quad u_z^* = \frac{u_z}{a_{ref}}$$

$$\rho^* = \frac{\rho}{\rho_{ref}} \quad P^* = \frac{P}{\rho_{ref}a_{ref}^2} \quad T^* = \frac{T}{a_{ref}^2/R_g}$$

$$E^* = \frac{E}{a_{ref}^2} \quad H^* = \frac{H}{a_{ref}^2}$$

$$\mu^* = \frac{\mu}{\mu_{ref}}$$ (2.25)

Ultimately, the reference Reynolds number $Re_{ref}$ scales the viscous fluxes when treating the conservative form of the non-dimensionlized RANS equations.

$$Re_{ref} = \frac{\rho_{ref}a_{ref}L_{ref}}{\mu_{ref}}$$ (2.26)
of the turbulence. The two-layers Baldwin-Lomax turbulence model is given as follows

\[
\mu_T = \begin{cases} 
\mu_i^T & y \leq y_m \\
\mu_o^T & y > y_m
\end{cases}
\]  

(2.27)

where \( y_m \) is the distance to the wall where the character of the eddy viscosity switches. The inner eddy viscosity \( \mu_i^T \) is given by

\[
\mu_i^T = \rho \frac{l_{mix}^2}{vm} \omega
\]

(2.28)

where \( \omega \) is the absolute vorticity magnitude. \( l_{mix} \) is the characteristic mixing length

\[
l_{mix} = v \kappa \lambda (1 - e^{-x^{+}/A_0^{+}})
\]

(2.29)

where \( A_0^{+} = 26 \) and the von-Karman constant \( \kappa_{vK} \) is set to \( \kappa_{vK} = 0.41 \). The non-dimensional wall distance \( y^+ \) is given by

\[
y^+ = \frac{\rho_{wall} u_\tau y}{\mu_{wall}}
\]

(2.30)

where \( u_\tau \) is the wall friction velocity,

\[
u_\tau = \sqrt{\frac{\tau_{wall}}{\rho_{wall}}}
\]

(2.31)

where \( \tau_{wall} \) is the wall shear stress vector. It is given by the projection of the shear stress tensor \( \tau \) onto the tangential \( t \) and bi-tangential \( b \) vectors to the surface. The absolute magnitude of the vorticity \( |\omega| \) reads

\[
|\omega| = \sqrt{\left( \frac{\partial u_z}{\partial y} - \frac{\partial u_y}{\partial z} \right)^2 + \left( \frac{\partial u_x}{\partial z} - \frac{\partial u_z}{\partial x} \right)^2 + \left( \frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y} \right)^2}
\]

(2.32)

The outer eddy viscosity \( \mu_o^T \) is expressed as follows

\[
\mu_o^T = \rho \alpha_{BL} C_{cp} F_{wk} F_{kleb}
\]

(2.33)

The wake function \( F_{wk} \) allows to recognize wakes from boundary layer regions. It is found by taking the minimum value between the quantities \( y_{max} F_{max} \) and
2.1. GOVERNING EQUATIONS

$C_{wk}y_{max}u_{difu}^2 / F_{max}$ found on the observed line profile. The maximum velocity difference $u_{difu}$ is defined by subtracting the maximum absolute velocity found on the observed profile line to the minimum one (which is equal to zero if the profile line is attached to a wall). The distance $y_{max}$ is found on the profile line where $l_{mix} |\omega|$ is maximum. The $F_{max}$ factor is given by

$$F_{max} = \frac{1}{\kappa_v K} [l_{mix} |\omega|]_{y=y_{max}}$$  \hfill (2.34)

The Klebanoff intermittency function $F_{kleb}$ is given by

$$F_{kleb} = \left[ 1 + 5.5 \left( \frac{y}{\delta} \right)^6 \right]^{-1}$$  \hfill (2.35)

where $\delta = y_{max} / C_{kleb}$. The model coefficients are: $\alpha_{BL} = 0.0168$, $C_{cp} = 1.6$, $C_{wk} = 0.25$, $C_{kleb} = 0.3$. For three-dimensional flow problem, the eddy viscosity near corners is affected by the two adjacent walls. In order to take this effect into account, a blending function $f_{BL}$ is introduced to smoothly distribute the eddy viscosity. Indeed, in these corner regions, the eddy viscosity is computed twice from the two adjacent walls, that is namely $\mu_{c1}^T$ and $\mu_{c2}^T$. Eventually, the two eddy viscosities $\mu_{c1}^T$ and $\mu_{c2}^T$ that have been found are combined together through the blending factor $f_{BL}$ to get the eddy viscosity $\mu^T$.

$$\mu^T = (1 - f_{BL}) \mu_{c1}^T + f_{BL} \mu_{c2}^T$$  \hfill (2.36)

Following the work of Bassi et al. [9], the blending function $f_{BL}$ is given by

$$f = \frac{1}{y_{c2}^T} = \frac{1}{y_{c1}^T} + \frac{1}{y_{c2}^T}$$  \hfill (2.37)

To reproduce the effect of the laminar to turbulent boundary layer transition, Baldwin and Lomax [8] have proposed that the eddy viscosity is set to zero everywhere in a profile when the maximum computed value of the eddy viscosity $\mu_{max}^T$ is less than a specified value, that is

$$\mu^T = 0 \quad \text{if} \quad \mu_{max}^T < C_{tr} \mu^L$$  \hfill (2.38)

It is proposed [8] to have the transition coefficient $C_{tr}$ equal to 14.
2.2 Ni Lax-Wendroff algorithm

The numerical method used to solve the unsteady RANS equations is explained in this section. The solution procedure consists of an explicit, time marching, Finite Volume Method (FVM) that is second order accurate in space and time. For time discretization, the one-step Ni-Lax-Wendroff scheme has been chosen, originally developed by Ni [70] to solve the Euler equations. The solution algorithm is based on a central cell-vertex variable allocation. This type of scheme is typical of those used nowadays in industry. The stability criterion is analytically derived for the inviscid term and is found empirically for the viscous term. A second and a fourth order artificial smoothing are applied to the system of equations to be solved. Indeed, this central scheme yields unrealistic oscillations (odd-even wiggles). These numerical artifacts can be partially damped out by a fourth order artificial smoothing. In addition, shocks can be better predicted with the help of a second order artificial smoothing. A one-dimensional non-reflecting boundary condition strategy is selected to specify the flow conditions at inlet and outlet planes.

2.2.1 Time discretization

Knowing the value of the state vector $Q$ at time $n$, one can explicitly find its new value at time $n+1$, using a Taylor series expansion.

$$Q^{n+1} = Q^n + \left( \frac{\partial Q}{\partial t} \right)^n \Delta t + \left( \frac{\delta^2 Q}{\partial t^2} \right)^n \frac{\Delta t^2}{2} + O \left( \Delta t^3 \right) \quad (2.39)$$

The state vector $Q$ can be expressed as a function of the inviscid and viscous fluxes using the conservative form of the RANS equations (see Eq. 2.14). This allows to rewrite Eq. 2.39 as

$$Q^{n+1} - Q^n = -\Delta t \left( \frac{\partial F_I}{\partial x} + \frac{\partial G_I}{\partial y} + \frac{\partial H_I}{\partial z} - \frac{\partial F_V}{\partial x} - \frac{\partial G_V}{\partial y} - \frac{\partial H_V}{\partial z} \right) ^n -$$

$$- \frac{\Delta t^2}{2} \frac{\partial}{\partial t} \left( \frac{\partial F_I}{\partial x} + \frac{\partial G_I}{\partial y} + \frac{\partial H_I}{\partial z} - \frac{\partial F_V}{\partial x} - \frac{\partial G_V}{\partial y} - \frac{\partial H_V}{\partial z} \right)^n \quad (2.40)$$

Defining $\delta Q = Q^{n+1} - Q^n$ and rearranging the second term yields

$$\delta Q = -\Delta t \left[ \frac{\partial F_I}{\partial x} + \frac{\partial G_I}{\partial y} + \frac{\partial H_I}{\partial z} - \frac{\partial F_V}{\partial x} - \frac{\partial G_V}{\partial y} - \frac{\partial H_V}{\partial z} \right]^n$$

$$- \frac{\Delta t}{2} \left( \frac{\partial}{\partial x} \left( \frac{\partial F_I}{\partial t} \right)^n + \frac{\partial}{\partial y} \left( \Delta t \frac{\partial G_I}{\partial t} \right)^n + \frac{\partial}{\partial z} \left( \Delta t \frac{\partial H_I}{\partial t} \right)^n \right)$$

$$- \frac{\Delta t}{2} \left[ -\frac{\partial}{\partial x} \left( \Delta t \frac{\partial F_V}{\partial t} \right)^n - \frac{\partial}{\partial y} \left( \Delta t \frac{\partial G_V}{\partial t} \right)^n - \frac{\partial}{\partial z} \left( \Delta t \frac{\partial H_V}{\partial t} \right)^n \right] \quad (2.41)$$
The second-order change of the inviscid and viscous fluxes \((F, G, H)\) is introduced as

\[
\delta(F, G, H) = \Delta t \frac{\partial(F, G, H)}{\partial t}
\]  

(2.42)

This enables to write Eq. 2.41 in a more compact form.

\[
\delta Q = \delta Q_I_1 + \delta Q_I_2 + \delta Q_{V_1} + \delta Q_{V_2}
\]  

(2.43)

where the inviscid and viscous first- and second-order changes of the state vector \((\delta Q_{I_1}, \delta Q_{I_2}, \delta Q_{V_1}, \delta Q_{V_2})\) are defined as

\[
\delta Q_{I_1} = -\Delta t \left[ \frac{\partial F_I}{\partial x} + \frac{\partial G_I}{\partial y} + \frac{\partial H_I}{\partial z} \right] n
\]

\[
\delta Q_{V_1} = \Delta t \left[ \frac{\partial F_{V_1}}{\partial x} + \frac{\partial G_{V_1}}{\partial y} + \frac{\partial H_{V_1}}{\partial z} \right] n
\]

\[
\delta Q_{I_2} = -\frac{\Delta t}{2} \left[ \frac{\partial}{\partial x} (\delta F_I) n + \frac{\partial}{\partial y} (\delta G_I) n + \frac{\partial}{\partial z} (\delta H_I) n \right]
\]

\[
\delta Q_{V_2} = \frac{\Delta t}{2} \left[ \frac{\partial}{\partial x} (\delta F_{V_1}) n + \frac{\partial}{\partial y} (\delta G_{V_1}) n + \frac{\partial}{\partial z} (\delta H_{V_1}) n \right]
\]

2.2.2 Space discretization

The computational mesh is assumed to contain only hexahedral cell volumes. The state variables are stored at the intersections of the mesh lines, that are namely the grid nodes. The grid nodes \(G_N\) define the vertices of the computational cells \(C\) (there are 8 vertices per hexahedral cell), see Fig. 2.1. The FVM assumes that the state variables are constant within the volume of a pseudo-cell \(P\). The pseudo-cell \(P\) includes the near-space volume surrounding the grid node \(G_N\), in between the adjacent grid nodes. Thus, the space discretization of 2.44 at each grid node \(G_N\) employs the integration of Eq. 2.44 over the volume \(V_P\) associated to the pseudo-cell \(P\).

\[
\delta Q = \frac{1}{V_P} \left[ \int_{V_P} \delta Q_{I_1} dV + \int_{V_P} \delta Q_{I_2} dV + \int_{V_P} \delta Q_{V_1} dV + \int_{V_P} \delta Q_{V_2} dV \right]
\]  

(2.45)

Applying the Gauss theorem to Eq. 2.45 yields

\[
\delta Q = -\left( \frac{\Delta t}{V_P} \right) P \int_{S_P} (F_{I_1} n_x + G_{I_1} n_y + H_{I_1} n_z) n dS \\
+ \left( \frac{\Delta t}{V_P} \right) P \int_{S_P} (F_{V_1} n_x + G_{V_1} n_y + H_{V_1} n_z) n dS \\
- \left( \frac{\Delta t}{2 V_P} \right) P \int_{S_P} (\delta F_{I_2} n_x + \delta G_{I_2} n_y + \delta H_{I_2} n_z) n dS \\
+ \left( \frac{\Delta t}{2 V_P} \right) P \int_{S_P} (\delta F_{V_2} n_x + \delta G_{V_2} n_y + \delta H_{V_2} n_z) n dS
\]

(2.46)
where \( n_i \) are the cartesian components of the unit vector normal to the surface \( S_p \). \( \Delta t_P \) is the time step associated to the pseudo-cell \( P \) and \( S_p \) is any of the 6 boundary faces of the pseudo-cell \( P \). In reality, the dimension of the pseudo-cell \( P \) is not explicitly known. The cells that are explicitly defined are those which surround the grid node \( G_N \), namely the cells \( (A-H) \), as shown in Fig. 2.2. The major advantage of the Ni-Lax-Wendroff algorithm lies in the fact that only explicitly known information is used. Thus, instead of computing the dimension of the pseudo-cell \( P \), the dimension and fluxes through the 8 cells \( (A-F) \) surrounding the grid node \( G_N \) are computed, using Eq. 2.46. The rate of change \( \Delta Q_P \) of all the state variable at the ghost node \( G_N \) is given by the arithmetical average of the rate of change \( \Delta Q_C \) of the state variables calculated in the 8 surrounding cells \( (C = A-F) \). For example, the volume over time step ratio is explicitly defined as

\[
\left( \frac{V}{\Delta t} \right)_P = \left( \frac{1}{8} \right) \sum_{C=A}^{H} \left( \frac{V}{\Delta t} \right)_C
\]  

(2.47)
First order inviscid flux

Applying the Ni-Lax-Wendroff space discretization (e.g. Eq. 2.47) strategy to Eq. 2.46 for the first order inviscid term yields

\[
\delta Q_{I_1} = \left(\frac{1}{8}\right) \sum_{C=A}^{H} \left[ -\left(\frac{\Delta t}{V}\right) \int_{S_C} \left( F_I n_x + G_I n_y + H_I n_z \right)^{n} dS \right]
\] (2.48)

where \( S_C \) represents the entire boundary area of cell \( C \). Considering the fact that only hexahedral cells are used, Eq. 2.48 becomes

\[
\delta Q_{I_1} = \left(\frac{1}{8}\right) \sum_{C=A}^{H} \left[ -\left(\frac{\Delta t}{V}\right) \sum_{f=1}^{6} \left( F_I S_x^f + G_I S_y^f + H_I S_z^f \right)_C \right]^{n}
\] (2.49)

where \( S_i^f \) denotes the vector components \( i \) of any of the 6 boundary faces \( f \) of the hexahedral cell \( C \). The averaged fluxes \( \overline{F_I}, \overline{G_I}, \overline{H_I} \) through each face \( f \) are given by

\[
\overline{F_I} = \frac{1}{4} \left( F_I^{G_1} + F_I^{G_2} + F_I^{G_3} + F_I^{G_4} \right)
\] (2.50)
2.2. NI LAX-WENDROFF ALGORITHM

where \((G_1, G_2, G_3, G_4)\) are the four vertices of face \(f\). Although this is only shown for \(F_I\), the averaging is the same for the two other inviscid fluxes \(G_I, H_I\).

**Second order inviscid flux**

The integration of the second order fluxes needs to be done over the mean surfaces of all the 8 surrounding cells \((A-H)\). The second order fluxes are derived from the first order fluxes, known at the center of each surrounding cell \((A-H)\). Therefore, the second order inviscid fluxes of Eq. 2.46 read

\[
\delta Q_{I_2} = - \sum_{C=A}^{H} \left[ \left( \frac{\Delta t}{2V} \right) C \left( \frac{1}{4} \right) \sum_{i=1}^{3} \int_{\overline{S_{C_i}}} \left( \Delta F_I n_x + \Delta G_I n_y + \Delta H_I n_z \right) dS \right]
\]

(2.51)

where the \(\overline{S_{C_i}}\)'s represent the mean surface area of each cell \((A-H)\) in the three spatial directions. The mean surface area is defined as being the surface area of the pseudo-cell \(P\). The \((1/4)\) term comes from the fact that each mean surface area \(S_{C_i}\) is taken into account 4 times in the integration of Eq. 2.51. Using the same averaging procedure as for the first order inviscid term (e.g. Eq. 2.50), Eq. 2.51 can written as follows

\[
\delta Q_{I_2} = - \sum_{C=A}^{H} \left[ \left( \frac{\Delta t}{2V} \right) C \left( \frac{1}{4} \right) \sum_{f=1}^{3} \left( \Delta F_I \overline{S^f_x} + \Delta G_I \overline{S^f_y} + \Delta H_I \overline{S^f_z} \right) \right]
\]

(2.52)

The averaged second order inviscid fluxes can be expressed as

\[
\Delta \overline{F_I} = \left( \frac{\partial F_I}{\partial Q} \right) \Delta Q_{I_1}, \quad \Delta \overline{G_I} = \left( \frac{\partial G_I}{\partial Q} \right) \Delta Q_{I_1}, \quad \Delta \overline{H_I} = \left( \frac{\partial H_I}{\partial Q} \right) \Delta Q_{I_2}
\]

(2.53)

Eq. 2.53 yields the formula to compute the fluxes derivative in respect to \(Q\). This leads to the computation of a Jacobian matrix. This operation can be very computationally intensive. In order to substantially reduce the computational cost, the averaged second order inviscid fluxes \((\Delta \overline{F_I}, \Delta \overline{G_I}, \Delta \overline{H_I})\) can be derived in a very efficient manner. Let us first introduce the first order inviscid change of the
2.2. NIX-LAX-WENDROFF ALGORITHM

conservative variables.

\[
\Delta Q = \begin{bmatrix}
\Delta \rho \\
\Delta (\rho u_x) \\
\Delta (\rho u_y) \\
\Delta (\rho u_z) \\
\Delta (\rho E)
\end{bmatrix}
\]  

(2.54)

Using the known values of the rate of change of the conservative variables (in Eq. 2.55), the second order inviscid fluxes can be expressed as

\[
\Delta F_I = \begin{bmatrix}
\Delta (\rho u_x) \\
u_x \Delta (\rho u_x) + \rho u_x \Delta u_x + \Delta p \\
u_x \Delta (\rho u_y) + \rho u_y \Delta u_x \\
u_x \Delta (\rho u_z) + \rho u_z \Delta u_x \\
u_x (\Delta (\rho E) + \Delta p) + (\rho E + p) \Delta u_x
\end{bmatrix}
\]  

(2.55)

\[
\Delta G_I = \begin{bmatrix}
\Delta (\rho u_y) \\
u_y \Delta (\rho u_x) + \rho u_x \Delta u_y \\
u_y \Delta (\rho u_y) + \rho u_y \Delta u_y + \Delta p \\
u_y \Delta (\rho u_z) + \rho u_z \Delta u_y \\
u_y (\Delta (\rho E) + \Delta p) + (\rho E + p) \Delta u_y
\end{bmatrix}
\]  

(2.56)

\[
\Delta H_I = \begin{bmatrix}
\Delta (\rho u_z) \\
u_z \Delta (\rho u_x) + \rho u_x \Delta u_z \\
u_z \Delta (\rho u_y) + \rho u_y \Delta u_z \\
u_z \Delta (\rho u_z) + \rho u_z \Delta u_z + \Delta p \\
u_z (\Delta (\rho E) + \Delta p) + (\rho E + p) \Delta u_z
\end{bmatrix}
\]  

(2.57)

where the change of the primitive variables are obtained as follows

\[
\Delta u_x = \frac{\Delta (\rho u_x) - u_x \Delta \rho}{\Delta \rho} \\
\Delta u_y = \frac{\Delta (\rho u_y) - u_y \Delta \rho}{\Delta \rho} \\
\Delta u_z = \frac{\Delta (\rho u_z) - u_z \Delta \rho}{\Delta \rho}
\]

(2.58)

\[
\Delta p = (\gamma - 1) [\Delta (\rho E) - u_x \Delta (\rho u_x) - u_y \Delta (\rho u_y) - u_z \Delta (\rho u_z) + \\
\frac{\Delta \rho}{2} (u_x^2 + u_y^2 + u_z^2)]
\]
First order viscous flux

The same strategy is applied to the first order viscous fluxes as well as for the second order inviscid fluxes. Both express a second order derivative of the state variables.

\[ \delta Q_{V_1} = \sum_{C=A}^{H} \left[ \left( \frac{\Delta t}{2V} \right)_C \left( \frac{1}{4} \right) \sum_{f=1}^{3} \left( \bar{F}_V S_x^f + \bar{G}_V S_y^f + \bar{H}_V S_z^f \right)^n \right] \quad (2.59) \]

The derivatives contained in the first order fluxes \( \bar{F}_V, \bar{G}_V \) and \( \bar{H}_V \) are known at the cell center of each cell \( C \), using the Gauss theorem. Hence, for any flow quantity \( \phi \), its derivative in cell \( C \) is given by

\[ \left( \frac{\partial \phi}{\partial x} \right)_C = \left( \frac{1}{V} \right)_C \sum_{f=1}^{6} \phi S_C^f \quad (2.60) \]

Second order viscous flux

The computation of the second order viscous flux demands much more computational resources than for the other fluxes. Indeed, a Jacobian in the three spatial direction must be computed, for each grid node \( G_N \). However, it has been shown (Hirsch [45]) that this flux has a negligible effect on the accuracy of the whole computation, in particular for the range of Reynolds numbers encountered in turbomachinery flows. Thus, the second order viscous flux is neglected.

2.2.3 Mesh singularity

For complex geometries, several hexahedral blocks (structured collection of hexahedral cells) need to be connected together. This leads to a multiblock grid topology. In this case, it is possible that at some particular grid nodes, there are an odd number of blocks connected together. This leads to have less, or more than 8 cells surrounding this node. The numerical correction at this node location is easily implemented. The final time over volume ratio, as shown in Eq. 2.47, is corrected by the actual number of surrounding cells \( N_C \) [29]. This gives

\[ \left( \frac{V}{\Delta t} \right)_P = \left( \frac{1}{N_C} \right) \sum_{C=1}^{N_G} \left( \frac{V}{\Delta t} \right)_C \quad (2.61) \]
2.3. ROBUSTNESS OF THE ALGORITHM: STABILITY AND SMOOTHING

2.3 Robustness of the algorithm: stability and smoothing

2.3.1 Stability analysis

A standard Von Neumann stability analysis of the Ni-Lax-Wendroff scheme applied to the RANS equations cannot be carried out. This is due to the highly non-linear form of these equations. However, a first approach to the condition of stability of the scheme is to apply a Von Neumann stability analysis to the inviscid part of the governing equations of motion. To do so, the linearized form of the Euler equations is taken. For the sake of simplicity, the reader should refer to Burdet and Lakehal [15] for an in-depth information about this stability analysis. The result of this analysis gives the inviscid stability criterion, by restricting the numerical time step $\Delta t_I$ as a function of the cell velocity and geometrical dimension.

$$\Delta t_I \leq \frac{J_I}{R_I + aM_I}$$  \hspace{1cm} (2.62)

where $J_I$ is a Jacobian representing the volume of the cell. $R_I$ and $M_I$ represent different metrics of the cell. The speed of sound is represented by $a$. To get the overall stability criterion, the time step restriction $\Delta t$ has to take into account the viscous term of the RANS equations. It is commonly accepted to scale the inviscid time step $\Delta t_I$ by using a local cell Reynolds number $Re_\Delta$.

$$\Delta t \leq \frac{\Delta t_I}{1 + \frac{2}{Re_\Delta}}$$  \hspace{1cm} (2.63)

where the cell Reynolds number $Re_\Delta$ is defined as

$$Re_\Delta = \frac{\rho \min (|u_x \Delta x|, |u_y \Delta y|, |u_z \Delta z|)}{\mu}$$  \hspace{1cm} (2.64)

where $\Delta x$, $\Delta y$ and $\Delta z$ are the different length of the considered cell. The Courant-Levy-Friedrich (CFL) condition is fulfilled by scaling the time step with a number ranging from 0 to 1. In general the CFL number is set to CFL = 0.9. For steady-state computation, to accelerate the convergence rate, the time step computed in each cell is locally used (i.e. local time stepping). On the contrary, in unsteady computation, the minimum time step found in the computational domain is used in every computational cell (i.e. global time stepping). This enables to ensure that the physical time is the same everywhere in the computational domain.
2.3.2 Numerical smoothing

The Ni-Lax-Wendroff scheme is second order accurate in time and in space. The second order term plays the role of a dissipative operator, analogous to an upwind scheme. Therefore, for the simulation of a smooth flow field (i.e., no large velocity or pressure gradient within the flow field), the scheme is stable. When solving the one-dimensional wave equation, it can be proved (c.f. Hirsch [45]) that the extension of this scheme in a finite difference form introduces a third order dispersion term. A dominating lagging phase is then unavoidable when applying this scheme to the RANS equations. This phenomenon can be interpreted as follows: when a simulation is carried out with this scheme, one can observe waves of different frequencies (due to the transient solution, computer limitation to represent a number and handling of the boundary conditions) which travel at different speeds. Hence, due to the phase error caused by the dispersive term, an aliasing phenomenon resulting from the superposition of the different numerical waves can arise. A well-known mode to characterize this error is the odd-even decoupling of pressure for steady calculation. In order to partially damp out these non-physical oscillations, a fourth order operator is added to the solved RANS equations, playing the role of a background smoother. Apart from this type of dispersive error feature, another major numerical wiggling artifact can arise. The compressible form of the Euler equations allows discontinuous flow solutions which result in the Rankine-Hugoniot relations. In reality, these discontinuities are not singular so that they have a very thin thickness. This is due to the dominant role of viscosity and heat conduction in this region. Interestingly, in earlier numerical simulations, where the same type of scheme has been used (see for instance Billonet et al. [14]), it has been observed that numerically-based oscillatory modes are created near discontinuities. This results in an inaccurate prediction of shocks. A dissipative operator of second order is therefore required to obtain a smooth result near shocks. This operator has to play a dominant role in shock region whereas in other parts of the flow field, it should be minimized as much as possible. The numerical (artificial) smoothing added to the RANS equations to be solved is

\[
S = \nu_2 \left[ \frac{\partial^2 Q}{\partial x^2} + \frac{\partial^2 Q}{\partial y^2} + \frac{\partial^2 Q}{\partial z^2} \right] + \nu_4 \left[ \frac{\partial^4 Q}{\partial x^4} + \frac{\partial^4 Q}{\partial y^4} + \frac{\partial^4 Q}{\partial z^4} \right]
\] (2.65)

where the artificial viscosities \( \nu_2 \) and \( \nu_4 \) scales the second- and fourth-order numerical smoothing. For the sake of simplicity, the reader should refer to Burdet and Lakehal [15] for an in-depth information on how these artificial viscosities are defined and computed. In general, the scaling of the numerical (artificial)
smoothing utilized in most of the computations is $\nu_2 = 0.01$ and $\nu_4 = 0.002$. This introduces a relatively low numerical smoothing, compared to what is used in industry.

2.3.3 Near wall smoothing

A major inconvenience involved by the numerical smoothing is the degradation of the solution in low Mach number regions (relatively to the incoming freestream Mach number $M_f$ of the flow problem studied). These are regions where the dissipative force is strong, relatively to the convective force. This is especially true in the boundary layer region. In these low Mach number regions, the numerical smoothing shifts the real viscosity level, which results in an inaccurate modeling of the near wall diffusion. In order to damp the numerical smoothing near walls and also wakes, it is scaled by a function which is determined by the local cell Mach number $M_C$.

$$\nu_2^{scaled} = \left( \frac{M_C}{M_f} \right)^2 \nu_2$$  \hspace{1cm} (2.66)

$$\nu_4^{scaled} = \left( \frac{M_C}{M_f} \right)^2 \nu_4$$  \hspace{1cm} (2.67)

2.4 Boundary conditions

2.4.1 Inlet/outlet boundary conditions

Nature of the RANS equations

The nature of the unsteady compressible RANS equations is parabolic-hyperbolic. As turbomachinery flows are essentially internal, there are two main regions of interest. On one hand, far from walls (blade, platform, casing, etc.), a free shear flow occurs so that the inertial force is much stronger than the viscous force (high local Reynolds number). This implies that the hyperbolic nature of the RANS equations dominates. On the other hand, flow near a wall feels much more the viscous force (relatively low local Reynolds number) due to the no-slip boundary condition at wall. Hence, the parabolic part of the RANS equations becomes important here. At the inlet and outlet cross plane, the flow is mostly inviscid. Indeed, only the tiny
boundary layer zone at the platform and casing is viscosity-affected. In order to substantially simplify the treatment of the inlet/outlet boundary condition, without relevant loss on the solution accuracy, it is assumed that the Euler equations are sufficient to describe the flow behavior at the inlet and outlet planes. For subsonic, inviscid, three-dimensional flow problems, Gustafsson and Sundström [39] have shown that four inlet and one outlet flow characteristics have to be imposed for conforming the hyperbolic nature of the Euler equation. Furthermore, if a supersonic flow is encountered, then five inlet and zero outlet flow characteristics have to be imposed.

**One-dimensional non-reflecting boundary conditions**

Imposing directly the flow state variables at the inlet and outlet plane leads to get in- and out-going spurious waves through the boundaries. In order to solve this issue, the governing equations of the flow characteristics are utilized. Let us assume that a perturbation wave \( \tilde{Q} \) is traveling normally to the inlet and outlet planes of the computational domain. In this case, the one-dimensional, linearized Euler equations governing the wave propagation can be used.

\[
\frac{\partial \tilde{Q}}{\partial t} + A \frac{\partial \tilde{Q}}{\partial x} = 0
\]  
where \( \tilde{Q} = \begin{bmatrix} \tilde{\rho} \\ \tilde{u}_x - \bar{u}_x \\ \tilde{u}_y - \bar{u}_y \\ \tilde{u}_z - \bar{u}_z \\ \tilde{p} - \bar{p} \end{bmatrix} \), \( \tilde{\rho} = \begin{bmatrix} \tilde{\rho} \\ \tilde{u}_x \\ \tilde{u}_y \\ \tilde{u}_z \\ \tilde{p} \end{bmatrix} \), \( A = \begin{bmatrix} \bar{u}_x & \bar{p} & 0 & 0 & 0 \\ 0 & \bar{u}_x & 0 & 0 & 1/\bar{p} \\ 0 & 0 & \bar{u}_x & 0 & 0 \\ 0 & 0 & 0 & \bar{u}_x & 0 \\ 0 & \gamma \bar{p} & 0 & 0 & \bar{u}_x \end{bmatrix} \)  

All these vectors and matrices are defined locally at a grid node \( G_N \), lying on the inlet or outlet plane. The matrix \( \bar{A} \) of the linearized coefficients can be diagonalized.

\[
\Lambda = T^{-1} \bar{A} T = \begin{bmatrix} \bar{u}_x & 0 & 0 & 0 & 0 \\ 0 & \bar{u}_x & 0 & 0 & 0 \\ 0 & 0 & \bar{u}_x & 0 & 0 \\ 0 & 0 & 0 & \bar{u}_x + \bar{a} & 0 \\ 0 & 0 & 0 & 0 & \bar{u}_x - \bar{a} \end{bmatrix}
\]
where \(\bar{a} = \sqrt{\gamma \bar{p} / \bar{\rho}}\) is the local speed of sound. The diagonal components of the matrix \(\Lambda\), which are the eigenvalues of the system, represent the speed of the characteristic waves. A positive value indicates a wave coming inside the computational domain at the inlet plane, and outside of the computational domain at the outlet plane. A negative value reverses the previous comment. The matrices \(T\) and \(T^{-1}\) are given by

\[
T = \begin{bmatrix}
-\frac{1}{\gamma} & 0 & 0 & \frac{1}{\gamma} & 0 \\
0 & 0 & 0 & \frac{1}{\gamma} & -1 \\
0 & 1 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 \\
0 & 0 & 0 & \frac{1}{2} & \frac{1}{2}
\end{bmatrix},
\]

\[
T^{-1} = \begin{bmatrix}
-\bar{c}^2 & 0 & 0 & 0 & 0 \\
0 & 0 & \bar{p}\bar{c} & 0 & 0 \\
0 & 0 & 0 & \bar{p}\bar{c} & 0 \\
0 & \bar{p}\bar{c} & 0 & 0 & 0 \\
0 & 0 & \bar{p}\bar{c} & 0 & 0
\end{bmatrix}
\]

Injecting Eq. 2.70 in the linearized Euler equation, see Eq. 2.68, and multiplying the result by \(T^{-1}\) reads

\[
\frac{\partial \Phi}{\partial t} + \Lambda \frac{\partial \Phi}{\partial x} = 0
\]

where

\[
\Phi = T^{-1} \tilde{Q}
\]

\(\Phi\) is the vector of the linearized characteristic perturbation waves. The non-reflecting boundary condition, following the definition of Engquist and Majda [27], implies that local perturbations propagated along the incoming characteristics have to vanish. Hence, for a subsonic flow, the non-reflecting boundary condition reads

\[
\Phi_{i, \text{inlet}} = \phi_{i, \text{outlet}} = 0
\]

In practice, the characteristics \(\Phi_i\) are first computed using Eq. 2.73. The perturbation vector \(\tilde{Q}\) is given by the value of state variables \(Q_{i, BC}\) that needs to be specified at the inlet and outlet planes, minus the value of the state variables \(Q_{i,n}\) computed at time \(n\). Using the condition given by Eq. 2.74, the characteristics are driven to 0 using a Newton-Raphson procedure [29]. Then, the value of the state variables
2.4. BOUNDARY CONDITIONS

\( Q_i^{n+1} \) at time \( n + 1 \) are computed back by using the inverse relation of Eq. 2.73.

In general, the total pressure \( P_i^f \), total temperature \( T_i^f \) as well as the tangential and radial flow angles at the inlet plane are specified. At the outlet plane, the static pressure \( P \) is specified.

2.4.2 Wall boundary conditions

The no-slip boundary condition applies onto every wall pertaining to the computational domain. For a stationary wall, it takes the form of a Dirichlet type of boundary condition for the velocity vector.

\[
\mathbf{u}_{wall} = 0
\]  

(2.75)

The pressure boundary condition at a wall is simplified to a level where the normal shear stress force is much smaller than the pressure gradient. Thus, using the momentum equation (see Eq. 2.7) and the fact that walls are non-porous, the pressure boundary condition takes the form of a Neumann type of boundary condition.

\[
\left( \frac{\partial p}{\partial n} \right)_{wall} = 0
\]  

(2.76)

The energy field at walls is determined by imposing either an adiabatic or an isothermal boundary condition. The adiabatic boundary condition says that walls are completely isolated from any external heat source, leading to a Neumann type of boundary condition for the temperature.

\[
\left( \frac{\partial T}{\partial n} \right)_{wall} = 0
\]  

(2.77)

The isothermal boundary condition implies that the walls are heated such as they have a constant, predetermined temperature. This is essentially a Dirichlet type of boundary condition.

\[
T_{wall} = T_{wanted}
\]  

(2.78)

A Dirichlet boundary condition is very easily implemented in the Ni-Lax-Wendroff scheme. Indeed, state variables are stored at the location of the grid nodes (cell-vertex algorithm). Thus, state variables at grid nodes lying on a wall are simply set to the specified boundary conditions. A Neumann boundary condition necessitates the extrapolation of the state variables at wall using one or two grid nodes inside the domain.
2.4.3 Periodic boundary conditions

For a given blade cascade, it is often not necessary to explicitly compute all the blade passages. Indeed, if the turbomachine operates in its stable operating margin, it is preferable to compute only one passage. It substantially saves computational resources and efforts. This induces that it is necessary to specify a periodic boundary condition on the sides of the computational domain. As the algorithm used is cell-vertex based, grid nodes lying on the two periodic planes are obviously duplicated in both periodic sides \( s_1 \) and \( s_2 \). In each periodic side, they receive only one half of the total flux contribution to update the corresponding state vector. Thus, in a linear case, the final rate of change of the state vector in side \( s_1 \) is given by the addition of the flux contribution received in side \( s_1 \) and \( s_2 \).

\[
\delta Q_{s_1, s_2}^{n+1}_{\text{final}} = \delta Q_{s_1}^{n+1} + \delta Q_{s_2}^{n+1}
\]  

(2.79)

The order of accuracy of the algorithm at the periodic planes is fully conserved with this approach.

2.5 Conclusion

The numerical method used for solving the RANS equations of fluid motion, explained in the previous sections, has been utilized to build a CFD code, namely MULT13. The numerical method employed is typical of industrial CFD code. This code has been validated against several typical flow problems encountered in turbomachinery. In particular, this code has shown a reasonable predictive capability of flows through a compressor linear cascade, a turbine linear cascade (for both see [15]) and an annular turbine cascade [16]. The odd-even wiggling numerical artifact has been recognized as the major drawback when using this type of central scheme.
Chapter 3

Film Cooling Jet Model

3.1 Assumptions and structure of the jet model

3.1.1 Modeling approach

Near-hole region

The path of a coolant flow within a film-cooled turbine blade is characterized by three main regions, see Fig. 3.1. First of all, it is dispatched toward the hole of injection through several internal channels. Just before flowing through the hole, it dwells in a plenum chamber. In a second step, the coolant emerges from the hole and makes contact with the freestream fluid. In this zone, so-called "near hole region", localized near the hole exit, the coolant starts to mix with the hot fluid but at very small scales. However, its trajectory rapidly turns toward the freestream flow direction. In a third step, the coolant flow mixes in a large scale with the hot freestream fluid. This mixing process is turbulent and unsteady. A non-constant blowing of the coolant jet can occur as a function of the surrounding freestream flow conditions.

The modeling of the coolant flow has to be located in one or several of these zones. The first zone, namely the plenum chamber, does not comply with one of the main restrictions, which says the flow inside the hole should not be solved, see Chapter 1. The third zone contains a highly unsteady and turbulent flow field, where the mixing has already taken place in a large scale. Then, this implies more a development of a turbulence model than anything else. Chances of success are small, and certainly the model would be applicable only for a small range of cooling parameters. The near hole region is then the ideal candidate for the modeling; it does not require to solve the flow inside the pipe and large scale mixing has not taken place yet. The near hole region is sketched in Fig. 3.2.
Figure 3.1: Location of the near hole region within a film-cooled turbine blade.

**Plane of injection**

The modeling of the near-hole coolant jet is based on a semi-empirical approach. On the one hand, the main flow features are identified and then modeled using the governing equations of fluid dynamics. On the other hand, the coefficients deriving from the proposed model are carefully tuned with the aid of experimental data\(^1\) describing the near hole three-dimensional flow field. As a consequence, the modeling approach is to experimentally anchor the near hole coolant jet model. More precisely, the model is only applicable in a two-dimensional plane normal to the direction of the coolant jet, so-called "plane of injection". It is located just downstream of the hole exit, as shown in Fig. 3.2. This is where the resultant flow field, derived from the model, is injected into the CFD algorithm. Between

---

\(^1\)The experimental data of Bernsdorf et al. [12] are used, see Chapter 5 for more details.
the hole exit and the plane of injection, the jet is strongly bent, which provokes a fundamental change in its own structure within this tiny region. By modeling the coolant jet at a location where it is almost completely re-organized but not fully mixed with the hot fluid yet, we alleviate the need of numerically solving the near-hole, complex flow region where the jet bends. Indeed, the accurate but fastidious resolution of this tiny region is essential to ensure the good prediction of the downstream jet evolution.

**Reynolds-Averaged Navier-Stokes (RANS) modeling**

The model aims to provide the designer with a tool that is computationally efficient and accurate for film-cooled blade optimization. Nowadays and in the foreseeable future, RANS solvers will stand the state of the art for 3D blade design. Therefore, the model is specifically dedicated for use in a CFD code solving the RANS equations. Thus, the model is simplified to a level that is resolved in RANS computation. The modeling of any flow structure is done by only considering its mean flow behavior.

3.1.2 Geometrical parameters

A typical cylindrical film cooling holes arrangement is shown in Fig. 3.3. It describes the main geometrical parameters, such as the hole diameter \( d \), the hole
3.1. ASSUMPTIONS AND STRUCTURE OF THE JET MODEL

Figure 3.3: Dimensions of cylindrical film-cooling holes.

length per hole diameter $l/d$, the pitch distance between two holes per hole diameter $s/d$, as well as the streamwise angle of injection $\alpha_0$ and the lateral angle of injection $\beta_0$. The jet in crossflow phenomenon can produce a broad range of flow structures downstream of the injection site, depending on the geometrical arrangement of the holes, and also on the geometry of the hole itself. The validation of the proposed model covers cylindrical hole, inclined in the streamwise direction up to $60^\circ$ and up to $90^\circ$ in the lateral direction. These hole inclinations are typical of the pressure and suction sides of a turbine blade. Showerhead (leading edge cooling) and endwall cooling are not considered in this study since they have a streamwise injection angle of $90^\circ$: the coolant jet flow pattern can be then very different. Nowadays, shaped holes are more and more the rule in advanced cooling system. The current model is developed so that an inclusion of shaped holes can easily be done. Furthermore, long holes are assumed, i.e. the length of the hole is much greater than its diameter.
3.1.3 Flow regimes

**Compressibility**

The coolant jet and freestream flow are considered compressible. Furthermore, it is hypothesized that the Mach number, based on the coolant jet fluid properties, is less or equal to one (choked condition). A transonic regime within the near hole region is not considered in the model.

**Turbulence**

The local Reynolds number $Re_d$, based on the hole diameter and coolant fluid flow properties is given by

$$Re_d = \frac{\rho c U c d}{\mu c}$$

In all cases, it is assumed that $Re_d > 5 \cdot 10^3$ so that the coolant flow is fully turbulent, see Schlichting and Gersten [80]. As film-cooled blades are generally designed for high-pressure turbines, they are standing just downstream of the exit of the combustor chamber. At the exit of a gas turbine combustor, freestream turbulence is typically ranging between 10% to 20%. This means that the freestream flow is highly turbulent. The effect of the freestream turbulence on the coolant jet can be important but concerns especially the mixing occurring some distance downstream of the injection site. The effect of freestream turbulence is not taken into account in the modeling of the jet at the plane of injection. Indeed, it seems that it has little influence upon the jet in the near hole region, see for example results of Mayhem et al. [61]. In addition, it is hypothesized that the freestream boundary layer is fully turbulent when approaching the hole so that transition processes are neglected.

**Unsteadiness**

In reality, coolant jets at film-cooled turbine blade walls exhibit a pulsating behavior (see Abhari and Epstein [1]). In particular, rotor blades, and more generally second stage blades, work in a highly unsteady flow environment. Periodically, there are upstream wakes and secondary flows convecting and diffusing through the downstream blade passages and also potential interactions and shock systems occurring in between the blade rows. This causes an unsteady behavior of the coolant jet blowing. In fact, the strength of the jet blowing is very dependent on
the recent history of near hole flow conditions. A measure of the near hole jet unsteadiness can be determined by the near hole reduced frequency $\Omega^d$

$$\Omega^d = \frac{F_j d}{U_c}$$

(3.2)

Where $F_j$ is the jet pulsating frequency. The pulsating frequency is very often triggered by the wake passing frequency \textit{i.e.} by the rotor speed. In general, this near hole reduced frequency $\Omega^d$ is much less than 1. This means that, in the near-hole region only, it can be hypothesized that the jet has a quasi-steady behavior. Thus, at each time step, the jet can be modeled using a steady approach. Note here that it does not mean that the jet is not pulsating, it only means that the coolant fluid particles between the hole exit and the plane of injection are convecting much faster than the time taken by the jet to pulsate half a period.

**Coolant to freestream flow ratios**

The density ratio $DR$ is generally used to quantify the thermal difference between the hot and the cold fluid.

$$DR = \frac{\rho_c}{\rho_f}$$

(3.3)

For film-cooling applied in turbomachinery, the density ratio is typically ranging from 1.0 (no temperature difference) to 2.0 (high temperature difference). The massflux and momentum of the coolant jet relative to the freestream flow is often quantified by the blowing $BR$ and momentum flux ratio $IR$.

$$BR = \frac{\rho_c U_c}{\rho_f U_f} \quad IR = \frac{\rho_c U_c^2}{\rho_f U_f^2}$$

(3.4)

In general, the blowing ratio of a pressure and suction side film-cooling is included between 0.5 and 3.0. A blowing ratio of 0.5 corresponds to a coolant jet tending to stick to the wall (wall jet) whereas a blowing ratio of 3.0 indicates a jet detached from the wall, penetrating beyond the freestream boundary layer, see for instance Leylek and Zerkle [56].

---

For example, a turbine having a rotor speed of 6000 rpm and 40 rotor blades (which gives a blade passing frequency of 4000 [Hz]), and also a cooling hole diameter of 1 [mm] and a coolant velocity of 200 [m/s] leads to a near hole reduced frequency of $\Omega^d = 2 \cdot 10^{-2}$
3.1.4 Flow structures included in the jet model

The coolant jet behavior in the near hole region is highly three-dimensional and contains several flow scales. The main, large scale flow features are shown in Fig. 3.4.

![Diagram of jet flow features](image)

*Figure 3.4: Near hole coolant jet flow features.*

**Near hole jet behavior and trajectory**

It is commonly accepted that the coolant jet behavior is quantitatively very close to that of a solid body near the hole exit. Moussa *et al.* [68] have shown that the incoming freestream flow suffers from a deceleration near the leading edge of the hole similar to the one felt around a cylinder in crossflow. Thus, the coolant jet provokes a blockage effect onto the external freestream flow. This phenomenon
induces a pressure gradient between the windward side and the lee side of the jet, which in return bends rapidly the trajectory of the jet fluid toward the freestream flow direction. The trajectory of the jet is therefore a major phenomenon to be modeled.

**Counter-rotating vortex pair - convective entrainment**

There is a considerable amount of dynamical adjustments within the jet flow between the hole exit and few diameters downstream. Moussa et al. [68] have proposed that the major change in the jet flow is the stretching and tilting of the vorticity created within the boundary layer of the coolant flow inside the hole. This vorticity can be originally seen as a vorticity ring inside the hole, as shown in Fig. 3.5. Indeed, the evolution of this ring leads rapidly to concentrate a high percent-

![Center view Cross view](image)

*Figure 3.5: Creation and evolution of the coolant flow vortex ring.*

age of vorticity into its sides. Then, the two sides of the vorticity ring fall in line with the dominant streamwise velocity, to form a so-called Beltrami flow. This leads to the appearance of a Counter-rotating Vortex Pair (CVP) structure. Following the work of Moussa et al. [68], about 75% of the total mean vorticity existing inside the coolant boundary layer in the hole is redistributed to the CVP. Fric and Roshko [31] have shown that at one hole diameter downstream of the hole center, the CVP structure is already present. Further downstream, the CVP structure supports an entrainment of the freestream fluid toward the core of the jet. In particular, the orientation of the CVP rotation brings hot freestream fluid underneath the jet which induces a dramatic reduction of the thermal protection offered by the film cooling jet. As a consequence, the CVP structure needs to be modeled in order to
3.1. ASSUMPTIONS AND STRUCTURE OF THE JET MODEL

set the right convective entrainment. Note here that as the strength of the CVP and its shape are very sensitive to the geometry of the hole, the above-cited assumption of a long hole is essential.

Mixing - diffusive entrainment

Near the hole exit, there is almost no convective mixing due to the CVP structure. Morton and Ibbetson [67] have suggested that at the leading edge of the jet, there is only a short-range turbulent mixing, as shown in Fig. 3.6. The remaining vorticity

![Center view](image)

*Figure 3.6: Short and long range entrainment (mixing).*

embedded within the coolant fluid in the hole distributes along the jet boundary, showing up as turbulent eddies. This induces a local and quantitatively small entrainment of the freestream fluid within the jet. This diffusive entrainment mode is dominant near the hole exit but reduces relatively to the CVP structure further downstream. Surprisingly, this diffusion process leads to a type of mixing function for velocity and temperature between the two fluid that is largely insensitive to the type of coolant injection investigated, see for instance Abramovich [3].

Wake

The freestream flow is naturally deflected by the coolant jet when coming into the near hole region. In analogy to a flow around a cylinder, it is expected that a wake forms behind the jet, in its lee side. Andreopoulos and Rodi [5] have shown that this wake zone is characterized by a velocity deficit in the lower part of the jet, near the wall. In this wake region, the gradient of streamwise velocity is relatively
small compared to the one observed in the mixing zone, as shown in Fig. 3.7. In connection to the formation of the wake, it can be expected that a von Karman vortex street is shedding, as it is observed in the wake of a cylinder. Although small vortical structures in the wake have been observed by many researchers, they are not related to a von Karman vortex street. Indeed, in studying this phenomena with smoke visualization, Fric and Roshko [31] quoted that "The flow around the jet looks nearly potential; the near wake streaklines are closed. Conversely, the flow around the cylinder separates, thereby opening the near wake". This means that vorticity coming from the freestream can be neither created nor shed at the coolant jet surface. In fact, the flow around the jet has essentially a potential character. Morton and Ibbetson [67] concluded that the wake is only existing in the lee side of the jet. This wake is much more a zone of retarding fluid due to the action of the CVP structure in the freestream boundary layer. Thus, this "near wall velocity damping zone" needs to be accounted for in the modeling of the velocity field.

Figure 3.7: Sketch of the different flow zones just downstream of the hole exit.
3.2. POSITIONING OF THE PLANE OF INJECTION: JET TRAJECTORY

Other flow structures

Although the wake does not exhibit a von Karman vortex street, small upright wake vortices have been observed, see Morton and Ibbetson [67], as sketched in Fig. 3.8. It is suggested that these vortical structures have their origin in the interaction between the wake boundary layer and the wall. The consequence in the evolution of the coolant jet is unclear but shows to be negligible in a RANS equations framework. Furthermore, Andreopoulos and Rodi [5] have noted the presence of a horseshoe vortex at the two sides of the jet, near the wall, see Fig. 3.8. This horseshoe vortex is created by the deflection of the vorticity lines embedded in the freestream boundary layer. However, they have reckoned that the circulation of these vortices is much smaller than the one of the CVP, thus negligible in the modeling approach proposed in this thesis. It is worth noting that, in the case of the existence of a complex vortical structure inside the plenum chamber, some extra vorticity can be ingested into the hole. This may lead to the appearance of extra vortices at hole exit. As no clear picture of this type of flow has been universally found, it is not considered in the present jet model.

3.2 Positioning of the plane of injection: jet trajectory

3.2.1 Jet intrinsic frame of reference

The jet flow field is modeled at the plane of injection, located downstream of the hole, as shown in Fig. 3.9. The centre of the jet at the plane of injection \((X, Y, Z)_{je}\)
3.2. POSITIONING OF THE PLANE OF INJECTION: JET
TRAJECTORY

is known by searching the location \((X, Y, Z)\) of the intersection between the three-dimensional jet line trajectory and the plane of injection itself. The full model is developed using the intrinsic local coordinate of the jet \((\chi, \xi, \eta)\) which lies on the plane of injection. The origin of these local coordinates is located in the centre of jet. The first coordinate \(\chi\) is lying on the unity vector \(e_\chi\) normal to the plane of injection. The two coordinates \(\xi\) and \(\eta\) are linked to the two unity vectors \(e_\xi\) and \(e_\eta\) describing the surface of the plane of injection. In order to transform the general coordinates \((X, Y, Z)\) into the local coordinates \((\chi, \xi, \eta)\) system, a translation and three rotations have to be achieved. First, the translation from the origin of the general cartesian coordinate system to the center of the jet \((X, Y, Z)_{jc}\) is made. Then a positive rotation of 90° in the axial direction needs to be done to switch the orientation of the coordinate system. Finally, the two rotations of angle \(\alpha\) and \(\beta\) are carried out.

\[
\begin{bmatrix}
\chi \\
\xi \\
\eta
\end{bmatrix}
= \begin{bmatrix}
\cos \beta & 0 & -\sin \beta \\
0 & 1 & 0 \\
\sin \beta & 0 & \cos \beta
\end{bmatrix}
\begin{bmatrix}
\cos \alpha & -\sin \alpha & 0 \\
\sin \alpha & \cos \alpha & 0 \\
0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
1 & 0 & 0 \\
0 & 1 & 0 \\
0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
X - X_{jc} \\
Y - Y_{jc} \\
Z - Z_{jc}
\end{bmatrix}
\]

\[
\begin{bmatrix}
\chi \\
\xi \\
\eta
\end{bmatrix}
= \begin{bmatrix}
\cos \alpha \cos \beta & \sin \beta & -\sin \alpha \cos \beta \\
\sin \alpha & 0 & \cos \alpha \\
\cos \alpha \sin \beta & -\cos \beta & -\sin \alpha \sin \beta
\end{bmatrix}
\begin{bmatrix}
X - X_{jc} \\
Y - Y_{jc} \\
Z - Z_{jc}
\end{bmatrix}
\]

\[ (3.5) \]
3.2. POSITIONING OF THE PLANE OF INJECTION: JET TRAJECTORY

where $\alpha$ and $\beta$ are the local angles of the jet trajectory in respect to the hole exit surface.

3.2.2 Jet trajectory model

The coolant jet trajectory model proposed by Abramovich [3] is selected for application here. This is the only sub part of the current film cooling jet model that has not entirely been developed by the author. The jet trajectory model is developed for a circular jet in a crossflow. This model is used to compute both the lateral $Y/d$ and the streamwise $Z/d$ deflections of the jet. The model is based on an two-dimensional analysis of the dynamical forces which apply to a fluid element of length $dl$ representing the jet, as shown in Fig. 3.10. The jet is approximated as being a solid curved cylinder. Two forces balance. The jet is bent due to a force $dN$ created by the flow pressure (momentum) of the incoming freestream flow. This flow pressure force is directed toward the freestream flow direction and is scaled by a flow resistance coefficient $C_n$ which represents the action of the jet surface opposing the freestream flow.

$$dN = C_n \cdot \frac{\rho f (u_f \sin \theta)^2}{2} hdl$$

(3.6)

where $h$ is the bulk width of the jet. Furthermore, as the jet bends, a centrifugal force $dC$ is applied. It tends to push the curved axis of the jet outboard.

$$dC = \rho c \frac{u_c^2}{r} S_n dl$$

(3.7)
3.2. POSITIONING OF THE PLANE OF INJECTION: JET TRAJECTORY

where \( r \) is the local radius of curvature of the jet axis. \( S_n \) represents the cross-sectional area of the jet.

\[
S_n \approx \frac{\pi h^2}{4}
\]  

(3.8)

The forces balance as follows

\[
dN = -dC
\]  

(3.9)

The jet cross-sectional area \( S_n \), and also its width \( h \), are in fact increasing when convecting downstream, due to the diffusion and lateral spreading process. It is hypothesized that the spreading rate of the jet is constant. This means that

\[
h = h_0 + c_h l
\]  

(3.10)

where \( l \) represents the total length of the jet trajectory. According to its experiments, for circular inline holes, Abramovich [3] has suggested that \( h_0 = 2.25d \) and \( c_h = 0.22 \). The streamwise and lateral jet trajectory \( S/d \), in terms of hole diameter \( d \), can therefore be found by integrating Eq. 3.9 [3].

\[
\left( \frac{S}{d} \right) = \sqrt{39k_1 \cdot \ln \left[ \frac{10 + \left( \frac{X}{d} \right) + \sqrt{\left( \frac{X}{d} \right)^2 + 20 \left( \frac{X}{d} \right) + 7k_1 \cot^2 \theta_0}}{10 + \sqrt{7k_1 \cot \theta_0}} \right]}
\]  

(3.11)

where \( \theta_0 = \alpha_0 \) or \( \beta_0 \) whether \( S = Z \) or \( Y \). The coefficient \( k_1 \) reads

\[
k_1 = \frac{\pi \cdot IR \cdot \sin \theta_0}{4C_n}
\]  

(3.12)

The flow pressure coefficient \( C_n \) can vary as a function of the streamwise injection angle \( \alpha_0 \), the lateral injection angle \( \beta_0 \) and the momentum flux ratio \( IR \). This flow pressure coefficient needs to be tuned\(^3\), see Chapter 5. The axial position \((X/d)_{jc}\), in other terms the axial location where the plane of injection lies, is given by the user. Once the jet lateral \((Y/d)_{jc}\) and vertical \((Z/d)_{jc}\) positions at the plane of injection are found using Eq. 3.11, one needs to know the local lateral \( \beta \) and streamwise \( \alpha \) inclination angles of the jet. This is done by differentiating Eq. 3.11.

\[
\frac{\partial Z}{\partial X} = \sqrt{\frac{39k_1}{(\frac{X}{d})^2 + 20 \left( \frac{X}{d} \right) + 7k_1 \cot^2 \theta_0}}
\]  

(3.13)

\(^3\)There are two flow pressure coefficients to be tuned, one for the streamwise trajectory \( C_{n_x} \) and the other one for the lateral trajectory \( C_{n_y} \).
Hence, the local inclination angle $\theta$ is found.

$$\theta = \arctan \left( \frac{\partial Z}{\partial X} \right)$$

(3.14)

### 3.3 Principle of superposition: mixing and wake

#### 3.3.1 Principle of superposition

The penetration of the coolant jet into the freestream boundary layer induces the merging of two different streams. The velocity and thermal flow profiles of the cold and hot fluids start mixing. This mixing process can be idealized in a two-dimensional form as sketched in Fig 3.11. In addition to the mixing process, the presence of the wall and the freestream boundary layer provokes the appearance of a wake in the lee side of the jet, as described in section 3.1.4. The wake has the characteristics of damping the jet velocity magnitude in the streamwise direction\(^4\). Furthermore, the action of the CVP structure may already have induced, at the plane of injection, a higher thermal mixing in the wake region than anywhere else in the coolant to freestream mixing zone. The wake zone can be identified as an extra mixing region. Thus, in order to reconstruct the velocity and thermal fields on the plane of injection, the coolant to freestream mixing and the wake need to be accounted for, as shown in Fig. 3.12. There are therefore two mixing zones. The first one, denoted by $m$ is between the coolant ($c$) and the freestream ($f$) fluid. The second one, denoted by $w$ is located in the wake zone. It is assumed

---

\(^4\)see for example the experimental measurements in Chapter 5
that the mean values of the velocity $u$ and total temperature $T_T$ of the coolant and freestream fluid are known. Hence, to reconstruct the overall flow field on the plane of injection, one needs to link the two fluid flow profiles through the two mixing zones. The principle of superposition is used here. That is, for any flow quantity $\phi$ ($\phi$ can be either the velocity or temperature),

$$\phi = (\epsilon_\phi + \Delta \epsilon_\phi) \phi^c + (1 - \epsilon_\phi) \phi^f$$  \hspace{1cm} (3.15)

where $\epsilon_\phi$ is the mixing function between the coolant and freestream fluid. $\Delta \epsilon_\phi$ is the wake flow damping (mixing) function located near the wall. The mixing function $\epsilon_\phi$ varies from 0 (only freestream fluid) to 1 (only coolant) whereas the wake flow damping function $\Delta \epsilon_\phi$ is free, depending on the wake intensity.

### 3.3.2 Mixing functions

**Jet boundary shape**

The mixing functions $\epsilon_\phi$ and $\Delta \epsilon_\phi$ need to be characterized. First of all, it is assumed that these functions do not change, whatever the angle $\zeta$ is. Hence, they
3.3. PRINCIPLE OF SUPERPOSITION: MIXING AND WAKE

depend only on the radial direction \( r \). Many experiments ([3], [68]) have shown that the shape of the mixing zone is very similar to the shape of a kidney. To define the jet boundaries near the hole exit, Coehlo and Hunt [21] have proposed an inviscid three-dimensional vortex sheet model in which the kidney shape of the jet is clearly derived. However, because the RANS equations are the framework of the model, the jet mixing and wake boundaries are approximated as an ellipse\(^5\), as shown in Fig. 3.12. The two axes of the ellipse \((A_{m,w}, B_{m,w})\) are set as model coefficients. Thus, for any local position \((\xi, \eta)\) on the \textit{plane of injection}, the selected mixing zone \((m \text{ or } w)\) local radius \( r_{m,w} \) is given by

\[
r_{m,w} = \sqrt{\left[\frac{\xi - \xi_{m,w}}{A_{m,w}}\right]^2 + \left[\frac{\eta - \eta_{m,w}}{B_{m,w}}\right]^2}
\]

where \((\xi_{m,w}, \eta_{m,w})\) is the selected mixing zone center, given by the trajectory model.

Mixing functions of two different streams

Many researchers have studied streamwise profiles of the velocity and temperature fields in the mixing region of two different streams. Example of such flows are co-flowing jets, wake, cross-flowing jets, counter-flowing jets, etc. Abramovich [3] gathered a broad range of studies, allowing thus to synthesize a large spectrum of experimental observations. This data collection leads to a remarkable observation: the velocity and temperature field appears to always follow the same type of profile, whatever the type of the two-stream mixing is. Furthermore, Schlichting and Gersten [80] have derived the boundary layer equation for axisymmetric free shear flows and wall jets, showing that the velocity field in the mixing zone can be theoretically always described using a power law (the exponent may change depending on the type of flow observed). The jet case in this study, namely a jet in crossflow near a wall, is a combination of a wall jet and an axisymmetric free shear flow. It is therefore assumed that the mixing function can take the form of a power law or a hyperbolic function, pretty close to a power law. For the jet velocity, the mixing function \( \epsilon_U \) takes the form of a power law.

\[
\epsilon_U = \left[1 - C_U \cdot (r_m)^{\frac{2}{3}}\right]^2
\]

\(^5\)Note that the combination of the mixing function \( \epsilon_\phi \) and wake function \( \Delta \epsilon_\phi \) allows to get a kidney type of shape for the reconstructed flow profile at the \textit{plane of injection}, see Chapter 5
3.3. PRINCIPLE OF SUPERPOSITION: MIXING AND WAKE

The jet to freestream temperature mixing function $\epsilon_T$ is more precisely described by an hyperbolic function. Indeed, the mixing zone might in certain cases (high blowing ratio and small streamwise injection angle) be very thin, in which an hyperbolic function is more flexible than a power law function.

$$\epsilon_T = 0.5 \cdot \left[ 1 - \left( \frac{\tanh(C_T \cdot (2r_m - 1))}{\tanh(C_T)} \right) \right]$$ (3.18)

where $C_U$ and $C_T$ are set so that when the radius $r_m = 1$, the highest rate of mixing occurs. The wake deficit function $\Delta \epsilon_\phi$ takes the same form as the mixing function. Note here that the imposition of the wake function allows to retrieve a jet mixing zone as a kidney shape, see Chapter 5 and Appendix D.

3.3.3 Flow profiles inputs

It is assumed that the non-mixed freestream and coolant flow levels are known. Thus, the velocity $u$ and total temperature level $T_T$ of the freestream and coolant flow are inputs to the model. The modeling approach to treat the mixing of the fluid, using the principle of superposition, requires to know the non-mixed flow profile of the two flows, see Eq. 3.15. In table 3.1 the flow profiles inputs are recapitulated; they must be known for the reconstruction of the overall flow field on the plane of injection.

<table>
<thead>
<tr>
<th>Flow variable</th>
<th>Freestream</th>
<th>Coolant</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u$</td>
<td>$u^f$</td>
<td>$u^c$</td>
</tr>
<tr>
<td>$T_T$</td>
<td>$T_{T}^f$</td>
<td>$T_{T}^c$</td>
</tr>
<tr>
<td>$\rho$</td>
<td>$\rho^f$</td>
<td>$\rho^c$</td>
</tr>
</tbody>
</table>

*Table 3.1: Input flow variables.*

---

6This means that the first derivative of the mixing function is maximal when $r_m = 1$
3.4 Non-mixed flow profile modeling

3.4.1 Penetration of the coolant jet

Inlet coolant total flow values

It is assumed that the non-mixed streamwise coolant velocity $u^c_x$ and temperature $T^c_T$ profiles are constant on the plane of injection. Furthermore, the streamwise coolant velocity is assumed equal to the coolant flow velocity $u^c$ inside the hole. In reality, it is not possible to know (measure) directly the velocity and temperature inside the hole. The hole size, whose typical value ranges from 0.5 to 2.0 [mm], is too small for the present measurement technology. In order to solve this problem, it is common to specify the total pressure $P^C_T$ and temperature $T^C_T$ of the coolant inside the plenum. These flow quantities are easily linked to blade design: this is what is normally known as boundary conditions when designing a film-cooled turbine blade. Although the coolant temperature $T^C_T$ given in the plenum can be identified as the coolant temperature on the plane of injection, the coolant streamwise velocity $u^c_x$ and/or coolant massflow cannot be derived directly. Indeed the coolant flow is expanded through the hole because of the existence of a gradient of static pressure between the plenum and the hole exit.

Isentropic Expansion

In order to derive the streamwise coolant velocity $u^c_x$, the expansion of the coolant fluid though the hole is modeled, as shown in Fig. 3.13. To expand the coolant jet, a negative gradient of static pressure has to exist between the plenum chamber and the hole exit. To this purpose, it is assumed that the static pressure $P_s$ in the near
hole region is known and that it is lower than in the plenum chamber. In reality, the cooling system is obviously designed as such. The static pressure $P_s$ can be inferred from measurements done in the freestream region and/or given as input. Once the static pressure $P_s$ is known, the ideal massflow $\dot{m}_{\text{ideal}}$ through the hole can be computed using an isentropic expansion.

$$\dot{m}_{\text{ideal}} = P_T^c \left( \frac{P_s^c}{P_T^c} \right)^{\frac{2\gamma}{2\gamma - 1}} \sqrt{\frac{2\gamma}{(\gamma - 1) R_{\text{gas}} T_T^c}} \left[ \left( \frac{P_T^c}{P_s^c} \right)^{\frac{2\gamma}{\gamma}} - 1 \right]$$ (3.19)

Discharge coefficient

It is obvious that losses are occurring through the hole, due to the friction and the three-dimensional aspect of the coolant flow. This leads to an actual massflow $\dot{m}_{\text{actual}}$ lower that the ideal massflow $\dot{m}_{\text{ideal}}$. In other terms, the ratio of the two massflows is denoted as the discharge coefficient $C_d$ of the hole.

$$C_d = \frac{\dot{m}_{\text{actual}}}{\dot{m}_{\text{ideal}}} \Leftrightarrow \dot{m}_{\text{actual}} = C_d \cdot \dot{m}_{\text{ideal}}$$ (3.20)

The discharge coefficient varies as a function of many parameters, such as the geometrical dimensions of the hole as well as the coolant to freestream flow ratios. In this study, the correlation given by Gritsch et al. [38] for computing the discharge coefficient $C_D$ is used, see Chapter 5 for more details. The coolant velocity $u_{c}^\xi$ is therefore easily found.

$$u_{c}^\xi = \frac{4 \dot{m}_{\text{actual}}}{\pi d^2}$$ (3.21)

3.4.2 Coolant jet secondary flows - CVP structure

Overall strategy for the CVP model: vortex mirror image

The jet secondary flow field results from the CVP structure. The secondary flow model is applied on the plane of injection using the local coordinates $(\xi, \eta)$. The impact of the wall upon the CVP structure is modeled through the addition of a mirror-image set $ji(j = 1, 2)$ of the two vortices $jr$, as shown in Fig. 3.14. Indeed, the presence of the wall induces an inviscid lateral push which tends to bring together the two vortices. The positions $(\xi_{1r}, \eta_{1r})$ and $(\xi_{2r}, \eta_{2r})$ of the two
3.4. NON-MIXED FLOW PROFILE MODELING

Figure 3.14: Cross view of the CVP and their mirror-image set.

vortices $\Omega_{jr}$ are model coefficients. The position $(\xi_{ji}, \eta_{ji})$ of the image vortex $\Omega_{ji}$ is computed as follows

$$\Omega_{ji} (\xi_{ji}, \eta_{ji}) = 3 \cdot \Omega_{jr} (\xi_{jr}, \eta_{jr}) - 2 \cdot \Lambda_j (\xi_{jw}, \eta_{jw})$$

(3.22)

where $\Lambda_j$ is the position of the wall point nearest vortex $\Omega_{ji}$.

Vortex model

Each vortex is assumed to be of an Hamel-Oseen type [71]. This vortex is derived from an exact solution of the incompressible vorticity equation.

$$\frac{D\omega}{Dt} = \underbrace{(\omega \cdot \nabla u)}_{\text{Evolution}} + \underbrace{\nu \nabla \omega}_{\text{Diffusion}}$$

(3.23)

It is assumed that, on the plane of injection, any vortex $l(l = 1r, 2r, 1i, 2i)$ is axisymmetric and quasi-steady. Furthermore, the vortex $l$ only moves toward the streamwise direction $e_x$, its tilting having already taken place. Thus, the vortex $l$ experiences only a constant stretching $\sigma_l$ in the streamwise direction $e_x$ while being diffused, see Fig 3.15. Plugging these assumptions in Eq. 3.23 leads to the
3.4. NON-MIXED FLOW PROFILE MODELING

following solution

\[ \omega^l_\infty (\xi, \eta) = \frac{\Gamma_l}{\pi} e^{\frac{\sigma_l r_l^2}{4\nu}} \] (3.24)

where \( \Gamma_l \) is the circulation of the vortex \( l \). This a model coefficient. The radius \( r_l \) is given by

\[ r_l = \sqrt{(\xi - \xi_l)^2 + (\eta - \eta_l)^2} \] (3.25)

The tangential velocity magnitude \( u^l_\theta (\xi, \eta) \) is therefore known, using the fact that \( \omega = \nabla \times u \).

\[ u^l_\theta (\xi, \eta) = \frac{\Gamma_l}{2\pi r_l} \left[ 1 - e^{\frac{\sigma_l r_l^2}{4\nu}} \right] \] (3.26)

The resulting two secondary velocity components \( u_\xi \) and \( u_\eta \) due to the CVP structure are found by summing all the velocity components given for each vortex.

\[ u_\xi (\xi, \eta) = \sum_{l=1}^{2i} u^l_\theta (\xi, \eta) \cdot \left( \frac{\eta - \eta_l}{r_l} \right) \cdot f_l \]

\[ u_\eta (\xi, \eta) = \sum_{l=1}^{2i} u^l_\theta (\xi, \eta) \cdot \left( \frac{\xi - \xi_l}{r_l} \right) \cdot f_l \] (3.27)

where \( f_l \) is a damping factor. Indeed, near the wall, due to the no-slip boundary condition, the velocity of the CVP is effectively dropping quickly down to zero. The damping factor reads

\[ f_l = 1 - e^{-\left( \frac{\xi - \xi_l}{d_f} \right)} \] (3.28)

where \( d_f \) represents the half distance from the wall where the damping function dies. After some testing, this coefficient is set to 0.025.
Vortex circulation scaling

The vortex circulation $\Gamma_l$ is scaled by analyzing the conservation of the net absolute vorticity flux $\phi_\omega$ over a control volume (CV) covering the near hole region [68], see Fig 3.16. The net absolute vorticity flux $\phi_\omega$ reads

$$\phi_\omega = \int_{CV} |\omega| (u \cdot n) \, dV = 0 \tag{3.29}$$

The net absolute vorticity flux is equal to zero because a quasi-steady flow in the near hole region is assumed. The Gauss theorem tells that the net flux of a scalar quantity within a CV is equal to the sum of fluxes passing through the boundaries of the selected CV. This means that the net absolute vorticity flux is equal to the addition of the absolute vorticity flux coming from the coolant $\phi_\omega^c$ and the freestream $\phi_\omega^f$ minus the vorticity flux going out of the CV, namely the absolute vorticity $\phi_\omega^\prime$, embedded within the jet. It is assumed here that all fluxes are orthogonal to the CV boundaries.

$$\phi_\omega = \sum_k \int_{S_k} |\omega| (u \cdot n) \, dS = \phi_\omega^f + \phi_\omega^c - \phi_\omega^\prime = 0 \tag{3.30}$$

where the index $k$ represents any CV sub-boundary face (such as in the hole, at freestream inlet, etc..). Now, let us consider that the inlet coolant velocity has an axisymmetric parabolic profile pointing toward the normal direction of the CV boundary surface in the hole, see Fig. 3.17. The coolant velocity profile in the hole is idealized by the following relation

$$u_z (r) = 2u^c \left[ 1 - 4 \left( \frac{r}{d} \right)^2 \right] \tag{3.31}$$
3.4. NON-MIXED FLOW PROFILE MODELING

Figure 3.17: Idealized coolant velocity profile in the hole.

So that only the vorticity component is directed toward the tangential direction

\[ \omega_r (r) = \frac{16u^c r}{d^2} \quad (3.32) \]

Hence, the coolant absolute vorticity flux can be scaled by putting Eqs. 3.31 and 3.32 in Eq. 3.30 to give

\[ \phi_\omega = \left( \frac{16}{15} \right) \pi u^c d \quad (3.33) \]

This total amount of vorticity is distributed within the coolant boundary layer in the hole, forming a vortex ring. Once this vortex ring is going out of the hole, its total amount of vorticity is mainly re-distributed equally to its two lateral sides forming the CVP structure. This means that each half vortex ring contains the following amount of vorticity.

\[ \phi_\omega^{\text{half vortex ring}} = \left( \frac{8}{15} \right) \pi u^c d \quad (3.34) \]

The circulation \( \Gamma_l \) of each vortex of the CVP structure is assumed to have almost the entire amount of the absolute vorticity embedded within the vortex ring.

\[ \Gamma_l = \frac{\phi_\omega^{\text{half vortex ring}}}{u^c} = \Gamma_l^* \left( \frac{8}{15} \right) \pi u^c d \quad (3.35) \]

where \( \Gamma_l^* \) is a model coefficient to be tuned. This model coefficient allows the user to control precisely the circulation of the vortices. This is a very important feature since the CVP structure sets the entrainment rate of the freestream flow downstream of the injection site, thus the film-cooling effectiveness and heat transfer rate.
3.4. NON-MIXED FLOW PROFILE MODELING

Vortex radius scaling

The radius $a_l$ of the vortex $l$ is defined as follows.

$$a_l = \sqrt{\frac{\sigma_l}{4\nu}}$$  \hspace{1cm} (3.36)

As the scaling of the stretching $\sigma_l$ of the vortex $l$ is uncertain, the vortex radius $a_l$ is set as a model coefficient.

3.4.3 Freestream flow profile

The velocity and temperature profiles of the non-mixed freestream flow are given by the user. In fact, the complete flow field before the hole has to be known. In practice, the flow field of the incoming freestream fluid is easily inferred from experimental data (they are inlet boundary conditions) or given by the design. Due to the presence of the wall, the freestream flow field cannot be assumed to be constant. Indeed, the freestream boundary layer needs to be included, as shown in Fig. 3.18. The freestream boundary layer is modeled using a power law, that is

$$\phi^f(z) = \phi_{0.99}^f \left( \frac{z}{\delta_{0.99}} \right)^{\frac{1}{7}}$$  \hspace{1cm} (3.37)

where $\delta_{0.99}$ is the boundary layer thickness. It is the location where the velocity $u_{0.99}$ is equal to $99\%$ of the freestream velocity $u^f$. As a consequence, the boundary layer thickness $\delta_{0.99}$ must also be given by the user. In general, this quantity

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{figure3.18.png}
\caption{Model of the freestream flow boundary layer.}
\end{figure}
is known \textit{a priori}. Although it is a poor variable to measure the boundary layer characteristics in comparison to the displacement thickness $\delta^*$ and the momentum thickness $\theta$, it has been selected because there is a well calibrated model to compute it in a computational mesh, see Chapter 4. The transformation of $u_f(z)$ to the jet coordinate system $(\chi, \xi, \eta)$ is performed using Eq. 3.5.

### 3.5 Model overview

The detailed overview of the model is given in Fig. 3.19. Briefly, the model consists of three main steps. The first step is to incorporate the boundary conditions into the model. That is the hole geometry, the incoming freestream flow, the coolant total flow properties in the plenum chamber and the streamwise position of the \textit{plane of injection}. In a second step, the non-mixed jet flow profile is built, using the penetration (expansion) and mirror-image vortex pair model. At the same time, the mixing functions are defined. It is crucial in this step to ensure that the model coefficients are well calibrated, see Chapter 5. In a third step, the non-mixed flow profile of both the freestream and coolant flow are joined together using the principle of superposition. Eventually, the final flow profile on the \textit{plane of injection} is known.
3.5. MODEL OVERVIEW

Figure 3.19: Detailed overview of the film cooling jet model.
3.5. MODEL OVERVIEW

Seite Leer / Blank leaf
Chapter 4

Numerical Implementation of the 3D Film-Cooling Jet

This Chapter concerns the numerical implementation of the film-cooling jet model. The choice of the numerical technique is first discussed. Based upon the analysis of the different existing techniques, the Immersed Boundary Method (IBM) is selected. The numerical application of the IBM for the flow problem studied is explained step-by-step. The jet is modeled by a three-dimensional body between the hole exit and the plane of injection. A toroidal shape is chosen to mimic the jet displacement body.

4.1 Implementation Strategy

4.1.1 Immersion of the 3D film-cooling jet in the computational mesh

Injecting the film-cooling jet model

The film-cooling jet model developed and shown in Chapter 3 is solely valid on the plane of injection. It allows the derivation of the jet flow profile just downstream of the hole exit but its use in a CFD code has not been shown yet. The inclusion of the jet model in a CFD code necessitates first to be able to inject the modeled jet flow profile into the computational domain, that is at each node of the computational mesh located nearby the plane of injection. This requires the development of a numerical method allowing the solution algorithm to take account of the modeled jet flow profile on the plane of injection, as shown in Fig. 4.1.
4.1. IMPLEMENTATION STRATEGY

The 3D jet body

Including only the film-cooling jet model in the computational mesh is not sufficient because the jet is essentially three-dimensional. As shown in Chapter 3, its large scale behavior is very similar to that of a solid body between the hole exit and the plane of injection. The presence of this solid body causes a blockage effect against the incoming freestream flow, as shown in Fig. 4.1. The inclusion of this blockage effect is essential for several reasons. Firstly, the freestream fluid cannot enter the plane of injection from upstream. In order to fulfill the law of the conservation of mass on the plane of injection, the incoming freestream flow must therefore be deflected around the jet. Secondly, as the jet blockage effect induces a deviation of the freestream streamlines, a potential field is created around the jet body. In particular, a pressure gradient appears between the upstream and downstream regions of the hole exit, as shown in Fig. 4.2. The pressure field around the jet is of a primary importance. As shown in the detailed overview of the model in Fig. 3.19, the near hole static pressure $P_s$ is an input to the model which actually sets the level of the jet blowing. Consequently, it is necessary to be able to place the jet solid body into the CFD mesh to ensure the conservation of mass and also to create the right potential field in the near hole region.

Figure 4.1: Inclusion of the near-hole 3D jet body.
4.1. IMPLEMENTATION STRATEGY

Local immersion of the 3D jet

This requires to extend the field of application of the numerical scheme shown in Chapter 2\(^1\) to include the jet. It is proposed that the coolant jet near the hole, including its solid displacement body and its modeled flow profile on the plane of injection, should be treated as a local boundary condition within the computational mesh. Furthermore, the boundary condition mimicking the jet surface should represent as well as possible the flow field at this location. In connection to this fact, the modeled three-dimensional shape of the jet surface should be similar to that of the real one. In summary, the implementation of the film-cooling jet model into the computational domain should be carried out as follows.

1. The jet flow profile is computed on the plane of injection

2. The three-dimensional jet displacement body is modeled between the hole exit and the plane of injection.

3. The full three-dimensional jet model is included in the computational mesh

4. The solution algorithm is updated to take into account the presence of the three-dimensional jet, modeled as a local boundary condition.

---

\(^1\)The numerical scheme consists of a Finite Volume Method applied to the Ni-Lax-Wendroff algorithm. This is an explicit, time-marching, second order accuracy scheme.
4.1. IMPLEMENTATION STRATEGY

4.1.2 Choice of the numerical method

The meshing problem

The local inclusion of a body inside the computational mesh is not a trivial task. A certain number of requirements needs to be fulfilled with an acceptable quality in order to perform a relevant numerical simulation. The primary requirements for such a problem are namely the comprehensive handling of a complex geometry, computational efficiency, accuracy, easy implementation and locality\(^2\). These conditions need to be acceptably fulfilled to preserve the stability, convergence rate and the solution quality as well as its potential use within any CFD code. The main problem for the implementation is that the topology of the solid surface to be included is not lying on the given mesh. Furthermore, this surface may move back and forth as the computation advances in time. Indeed, the unsteady behavior of the surface to be included must be considered as film-cooling jets can pulsate. There are several numerical techniques that allow the inclusion of a three-dimensional solid body within a given computational mesh. These can be classified in four main groups as shown in Fig. 4.3.

1. Global Grid Reshaping (GGR)
2. Overlapping Grid (OG) [86]
3. Cartesian Grid Cut Cell (CGCC) [50]
4. Immersed Boundary Method (IBM) [88]

These above methods are discussed in the next sections. A description and comparison of all four methods is done in order to find the best suited for the specific flow problem of this work.

Global Grid Reshaping (GGR)

The GGR method calls for the modification of the given global grid arrangement so as to include the three-dimensional body. It mainly consists of getting a proper body-fitted curvilinear mesh that describes the surface of the three-dimensional body to be included. This must be done either with a multiblock and/or an unstructured approach. It is evident that a lot of very complex issues arise if this method is used. The global grid will be distorted, the inclusion of new blocks

\(^2\)Locality means that the whole computational mesh does not need to be changed
4.1. IMPLEMENTATION STRATEGY

Figure 4.3: Different methods to immerse a body within a computational grid.

might be very tricky, and the creation of a new grid at each time step would certainly dramatically increase the computational time.

**Overlapping Grid (OG)**

The OG method is based on a Chimera interpolation technique, see Steger et al. [86]. The strategy is to patch a surface body-fitted grid inside the global grid. This patched grid exists only locally near the included surface. The solution algorithm is applied in both grids. Flow values are exchanged and interpolated periodically between the two grids so that the surface is taken into account. The main advantage of this technique is that it can theoretically handle any type of geometry. However, its main drawback is the difficult implementation and the related uncertainty of the solution accuracy. Indeed, three-dimensional grid patching needs to be done and the interpolation may lead to spurious numerical waves propagating in the computational domain. Furthermore, the order of accuracy of the solution algorithm...
4.1. IMPLEMENTATION STRATEGY

would certainly decrease near the grid interface, since some interpolations of the flow quantities between the grids must be carried out.

**Cartesian Grid Cell Cut (CGCC)**

In the CGCC method (see Kirkpatrick et al. [50] for a good overview), the included surface is tracked as a sharp interface so that the given computational mesh is cut at the location of the interface. The main advantage of the method is that the geometry of the surface is exactly defined (up to the grid node to node distance), given a proper cutting of the grid. However, this method leads to a large number of different geometries for the cut cells, such as trapezoidal, tetrahedral, pentahedral, hexahedral, etc. cells. In this context, the algorithm needs to be changed to handle very different grid cell geometries, which may reduce the algorithm accuracy near the surface. This induces a very tedious implementation of the method for three-dimensional computation. Furthermore, the allocation of the cell arrays should be fully dynamic as they would often need to be updated. This means that the overall computational efficiency may be reduced.

**Immersed Boundary Method (IBM)**

The strategy of the last method, the so-called Immersed Boundary Method (IBM), is to model indirectly the geometry of the included surface by an external body force acting on the grid nodes near this surface. This method has been introduced by Peskin [73] for low Reynolds biological flows. In his work, the computational boundaries were moving and modeled by a set of elements linked by a spring system. Using this early method, numerically stiff problems often arise. Goldstein et al. [37] and Saiki and Biringen [77] have extended the availability of the method by using a "feedback forcing" to the momentum equation so as to asymptotically set the desired boundary condition to represent a solid surface. However, a very low time step has to be specified ($CFL = 10^{-3}$ to $10^{-2}$) for transient flows [37] and the feedback forcing has been shown to be successful only for low-Reynolds number flows. More recently, Mohd-Yusof [66] and Fadlun et al. [28] have introduced a "direct forcing" approach in which the forcing function representing the action of the immersed three-dimensional body-surface onto the surrounding flow is directly included in the solved equations. This is done using a ghost-cell approach (as explained further in the next section) and low-order interpolation of the flow quantities inside the body. This method has been shown to be very stable and was successfully applied to Large Eddy Simulation (LES) of turbulent shear flows.
Majumdar et al. [59], Balaras [7] and Tseng and Ferziger [88] have very recently upgraded the method in order to use it in any arbitrary complex geometries. The accuracy of the scheme is maintained, the representation of the immersed surface is quasi-exact and it has been implemented in several existing codes, see [88]. The main advantage of this method is that there is absolutely no treatment of the given grid, it stays as it is during the whole computation. Another important advantage lies on its relatively easy implementation into an existing CFD code.

Comparison of the different methods

The four methods above are evaluated in Table 4.1 in terms of the requirements to be fulfilled. This comparison Table has been done using the available literature results and the author’s vision upon the specific problem encountered in this study. The Immersed Boundary Method (IBM) is therefore chosen for the implementation of the three-dimensional film-cooling jet within a CFD code. This is the method that presents the overall best trade-off between all the present requirements.

<table>
<thead>
<tr>
<th>Method</th>
<th>Accuracy</th>
<th>Computational efficiency</th>
<th>Handle complex geometry</th>
<th>Easy implementation</th>
<th>Locality</th>
</tr>
</thead>
<tbody>
<tr>
<td>GGR</td>
<td>++</td>
<td>-</td>
<td>++</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>OG</td>
<td>+/-</td>
<td>-</td>
<td>++</td>
<td>-</td>
<td>+/-</td>
</tr>
<tr>
<td>CGCC</td>
<td>+</td>
<td>+/-</td>
<td>+</td>
<td>-</td>
<td>++</td>
</tr>
<tr>
<td>IBM</td>
<td>+</td>
<td>+</td>
<td>+</td>
<td>+/-</td>
<td>++</td>
</tr>
</tbody>
</table>

Table 4.1: Comparison of numerical methods to include the film-cooling jet model in the computational mesh.

4.2 Immersed Boundary Method

4.2.1 The forcing function

In the first place, it is assumed that the boundaries of three-dimensional body to be included lie exactly upon a collection of grid nodes \( X_g \), as shown in Fig. 4.4. A forcing vector \( f \) mimicking the impact of the included surface to the surrounding
4.2. IMMERSED BOUNDARY METHOD

Fluid is added to the governing momentum and energy equation, see Eq. 2.14 (no body force \( B \) included).

\[
\frac{\partial Q}{\partial t} = - \left[ \frac{\partial F_I}{\partial x} + \frac{\partial G_I}{\partial y} + \frac{\partial H_I}{\partial z} \right] + \left[ \frac{\partial F_V}{\partial x} + \frac{\partial G_V}{\partial y} + \frac{\partial H_V}{\partial z} \right] + f \tag{4.1}
\]

where the forcing vector \( f \) reads

\[
f = \begin{bmatrix}
  f_p \\
  f_x \\
  f_y \\
  f_z \\
  f_E
\end{bmatrix}
\tag{4.2}
\]

The forcing \( f \) is only applied in the vicinity of the surface to be included. The objective of the introduction of this forcing is to compel the state vector \( Q_s \) (of any grid node \( X_s \) lying onto the include surface) into taking a specific value. In fact, the flow field at the surface boundaries can be represented by a boundary condition \( Q_{IBC} \) which is immersed in the computational domain\(^3\). It is assumed that the immersed boundary condition \( Q_{IBC} \) is totally known. In a synthetic way, the immersed boundary method consists of

- *Imposing the forcing \( f \) such as the state vector \( Q_s \) is equal to the immersed boundary condition \( Q_{IBC} \) at any location within the included surface.*

\(^3\)For example, a wall may imply an adiabatic, no-slip boundary condition
The governing equations to be solved (Eq. 4.1) are unsteady by nature, so that their temporal integration, included the forcing $f$, gives (adapted from Eq. 2.41)

$$\frac{Q^{n+1} - Q^n}{\Delta t} = - \left( \frac{\partial F_I}{\partial x} + \frac{\partial G_I}{\partial y} + \frac{\partial H_I}{\partial z} - \frac{\partial F_V}{\partial x} - \frac{\partial G_V}{\partial y} - \frac{\partial H_V}{\partial z} \right)^n -$$

$$- \frac{1}{2} \left[ \frac{\partial (\Delta F_I)^n}{\partial x} + \frac{\partial (\Delta G_I)^n}{\partial y} + \frac{\partial (\Delta H_I)^n}{\partial z} \right] -$$

$$- \frac{1}{2} \left[ - \frac{\partial (\Delta F_V)^n}{\partial x} - \frac{\partial (\Delta G_V)^n}{\partial y} - \frac{\partial (\Delta H_V)^n}{\partial z} \right] + f^n + \frac{1}{2} \frac{\partial f^n}{\partial t}$$

Or in a more compact manner, focusing on the forcing $f$, Eq. 4.3 can be expressed as

$$\frac{Q^{n+1} - Q^n}{\Delta t} = - \text{RHS}^n + f^n + \frac{1}{2} \frac{\partial f^n}{\partial t}$$

where $\text{RHS}^n$ contains all the first and second order fluxes, that is the Right Hand Side of Eq. 2.41. The forcing vector $F^n$ can therefore be computed at any time $n$. 

$$F^n = \text{RHS}^n + \frac{Q^{n+1} - Q^n}{\Delta t}$$

Looking at Eq. 4.5, the computation of the forcing $F^n$ is straightforward and does not require any additional CPU time. Indeed, the term $\text{RHS}^n$ is already known since it represents the fluxes of the time-marching algorithm. The state vector at the current time step $Q^n$ is known and the state vector at the next time step $Q^{n+1}$ is the immersed boundary condition to be fulfilled.

$$Q^{n+1} = Q_{IBC}$$

This is only true for the grid nodes located in the vicinity of the included surface. The introduction of the forcing $f$ does not change the numerical algorithm, it is merely included in the numerical scheme shown in Chapter 2 through the body force term $B$.

### 4.2.2 Immersed Boundary Method procedure

**Non conformal grids: distributing the forcing**

In the above presentation of the Immersed Boundary Method, the included surface is assumed to exactly lie upon a collection of grid nodes. In reality, it is very
unlikely to happen. Actually, the aim of the method is to not take care of the mesh arrangement when immersing the surface; it has to be independent of the grid structure. Thus, it is necessary to use an interpolation/extrapolation method to distribute the forcing $f$ to the grid nodes lying on the vicinity of the included surface. As shown in Fig. 4.5, the surrounding of the included surface can be divided in three zones. There is first the physical domain which contains all grid cells that are completely outside the location of the immersed body. The second zone is the interfacial domain, which denotes all the grid cells that are crossed by the surface of the immersed body. Note here that the grid nodes that pertain to these cells and that are inside the immersed body are so-called "ghost nodes". The last zone is the ghost domain. This is where the grid cells are completely inside the immersed body. The distribution of the forcing $f$ has to be done in the interfacial domain\(^4\). This essential step can be carried out in two different manners. The explicit strategy is to spread (by interpolation and/or extrapolation) the immersed boundary condition $Q_{IBC}$ to all the grid nodes pertaining to the interfacial domain so that the forcing function $f$ is distributed around the included body (inside and outside), see Saiki and Beringen. [77]. The alternative strategy is to impose the state vector $Q_p$ inside the included body, at the ghost-nodes $X_p$ so that the immersed boundary condition $Q_{IBC}$ is implicitly fulfilled at any point $X_s$ of the surface, as shown in Fig. 4.5, (see also Majumdar et al. [59] and Tseng

\(^4\) Some researchers have increased the width of the interfacial domain to 3-4 cells mainly for stability reasons, see Saiki and Beringen [77]
4.2. IMMERSED BOUNDARY METHOD

and Ferziger [88]). The second approach is chosen since it needs less numerical steps and leads a better stability, see Balaras [7]. This implicit immersed boundary method (or extended ghost-cell method [59]) can be summarized as follows

- Set explicitly the state vector $Q_p$ at the ghost nodes $X_p$ so that the Immersed Boundary Condition $Q_{IBC}$ is implicitly fulfilled at the immersed surface so that the forcing function $f$ is implicitly imposed.

Global procedure

The definition of the three zones surrounding the boundaries of the body to be included is a prerequisite for searching where the ghost-nodes $X_p$ are, so that for immersing the boundary conditions. In order to locate these zones in the computational domain, it is first necessary to define the three-dimensional surface of the body. Once it is defined, the searching for the zones can be done. Furthermore, the full immersion of the boundary conditions necessitates to interpolate the flow field in the vicinity of the surface in order to extrapolate the value of the state vector $Q_p$ to be imposed at the ghost-node. Indeed, the state vector $Q_p$ is extrapolated by taking into account the nearby computed flow field $Q_m$ as well as the immersed boundary condition $Q_{IBC}$ at the body surface. The approach chosen for the IBM procedure can therefore be synthesized as follows.

1. Set the shape of the jet displacement body.

2. Detect the different zones nearby the body surface: ghost / interfacial / physical zones.

3. Classify ghost-nodes ($X_p$).

4. Locate and extrapolate location of surface and mirror point ($X_s$ and $X_m$).

5. Interpolate flow field $Q_m$ at the mirror point $X_m$.

6. Extrapolate the flow field $Q_p$ at the ghost node $X_p$ taking into account the immersed boundary condition $Q_{IBC}$ at the surface $X_s$ and also the surrounding flow field $Q_m$ known at the mirror point $X_m$.

7. Update the whole flow field (see Eq. 2.41) using the ghost node state vector $Q_p$ (implicit forcing function).
4.2. IMMERSED BOUNDARY METHOD

It is worth noting that the grid cells that are entirely located in the ghost zone are no more taken into account during the computation. The state variables at their vertices (grid-nodes) are set to coolant flow condition. For the sake of simplicity, the steps 1 to 4 are shown in details in Appendices A and B. Meanwhile, the choice of the shape of the jet displacement body is explained in section 4.3. In fact, the method presented herein to include the three-dimensional film cooling jet in the computational mesh is independent of the selected shape that represents the jet displacement body. Thus, any type of jet shape i.e. hole geometry, could be included in the computational mesh. Steps 5 and 6 are explicated in the next section: they are the core of the Immersed Boundary Method. Step 7 is already known, see Chapter 2.

4.2.3 Immersion of the jet boundary conditions

Extrapolation of the state vector \( Q_p \) to be imposed

It is assumed here that the location of the ghost-nodes \( X_p \) is known. Each ghost-node is linked to the immersed surface through a surface point \( X_s \). In general, the surface point \( X_s \) is as close as possible to the ghost-node \( X_p \). In most situations, the normal line to the surface going through \( X_s \) is also going through the ghost-node \( X_p \). The state vector \( Q_p \) to be imposed at the ghost-node has to be extrapolated from the immersed boundary condition \( Q_{IBC} \) and the nearby surrounding computed flow field. Thus, a mirror point \( X_m \) of the ghost node is defined in the physical domain. This is where the surrounding flow field \( Q_m \) is interpolated. The mirror point \( X_m \) is found as follows.

\[
X_m = \frac{(1 + \lambda) X_s - X_p}{\lambda}
\]  

(4.7)

where \( \lambda = 1 \) by default. The state vector \( Q_m \) is interpolated using the value of the state vector computed at the grid nodes surrounding the mirror point \( X_m \). Indeed, \( X_m \) is not necessarily lying on a grid node. Once \( Q_m \) is found, the extrapolation of the state vector \( Q_p \) to be imposed takes place. If the immersed boundary condition \( Q_{IBC} \) is of a Dirichlet type, that is

\[
Q_{IBC} = Q
\]  

(4.8)

then the extrapolation is given by

\[
Q_p = (1 + \lambda) Q_{IBC} - \lambda Q_m
\]  

(4.9)
Furthermore, a Neumann type of boundary condition may be necessary to be im­
mersed. It is given by

$$Q_{IBC} = \frac{\partial Q}{\partial n}$$  \hspace{1cm} (4.10)\]

where \(n\) is the normal direction to the immersed boundary. In this situation, the
extrapolation scheme is changed to give

$$Q_p = \lambda Q_m - (1 + \lambda)Q_{IBC} |X_s - X_p|$$  \hspace{1cm} (4.11)\]

This is essentially a first order extrapolation that is chosen. Using a higher or­
der extrapolation scheme would necessitate the knowledge of more mirror points.
In fact, the boundary conditions \(Q_{IBC}\) to be imposed to mimic the jet are rela­tively simple, see next section. Furthermore, the knowledge of more mirrors points
would imply more searching, classification and interpolation steps that would slow
down the computational time.

**Jet boundary conditions \(Q_{IBC}\)**

The choice of the boundary conditions \(Q_{IBC}\) to immerse is a strategic decision.
On one hand, as the Immersed Boundary Method introduced has never been ap­
plied for the type of application encountered in this work, relatively simple bound­ary conditions should be imposed first. On the other hand, the immerscd boundary
conditions should reproduce the flow dynamics occurring within the jet boundaries
as accurate as possible to get a relevant numerical solution. There are two different
types of boundary conditions to be imposed. The first one obviously concerns the
plane of injection. The flow field at this location is known from the film-cooling
jet model presented in Chapter 3. In addition to the film-cooling jet model, the
pressure gradient in the normal direction to the plane of injection is assumed to
be very small, relatively to the transverse pressure gradient. Indeed, the plane of
injection is supposed to be located in the region where the jet has almost entirely
rearranged in the freestream direction. Hence, the transverse pressure gradient,
due to the CVP structure and mixing dominates. Therefore, the state vector \(Q_{IBC}\)
on the plane of injection consists of

$$Q_{plane \ of \ injection} = \begin{bmatrix}
u_x = \nu^\text{model} \\
\nu_\xi = \nu^\text{model} \\
u_n = \nu^\text{model} \\
T_p = T^\text{model} \\
\frac{\partial P}{\partial x} = 0 \end{bmatrix}$$  \hspace{1cm} (4.12)\]
4.2. IMMERSED BOUNDARY METHOD

The second type of boundary condition is to be applied at the surface of the jet, between the hole exit and the plane of injection. A three-dimensional piece of the jet boundary in the near hole region is represented in Fig. 4.6. As explained in Chapter 3, the mixing of the coolant jet and the incoming freestream fluid is done on a very small scale between the hole exit and the plane of injection. Moreover there is no creation of vorticity at the jet boundary since there is no solid wall. Therefore, it is assumed that there is no penetration of freestream fluid through the jet surface. This induces as a first approximation no pressure gradient through the surface. The tangential shear stress is neglected so that the jet surface is a slip boundary. Indeed, the jet surface is a singularity as it has no thickness. This means that the difference of velocity between the freestream flow and the coolant flow cannot be defined on the jet surface. The thermal condition at the surface is considered isothermal. Indeed, the jet is not insulated, it merely keeps the same temperature at its boundary. This temperature $T_{mix}$ is actually the averaged temperature in the tiny mixing layer that is represented by the immersed jet surface. It is given by

$$T_{mix} = (1 - \lambda) T^f_T + \lambda T^c_T$$

where $\lambda = 0.5$ by default. In summary, the immersed boundary condition at the jet surface is given by

$$Q_{IBC}^{jet\ surface} = \begin{bmatrix} u_n = 0 \\ \frac{\partial u_x}{\partial n} = 0 \\ \frac{\partial p}{\partial n} = 0 \\ T = T_{mix} \end{bmatrix}$$

Figure 4.6: A three-dimensional piece of the jet boundary in the near hole region.
4.2. IMMERSED BOUNDARY METHOD

Interpolation of the surrounding flow field $Q_m$

The interpolation of the surrounding flow field at the mirror node $X_m$ has to be carried out in the three-dimensional computational mesh. A large number of different interpolation schemes exist and can be used. High-order polynomials interpolation is not considered since it often creates wiggles and spurious numbers when used with a large collection of points. As proposed and extensively tested by Tseng and Ferziger [88], an inverse distance weighting is used. It has the advantage of preserving local maxima and produces in most situations a smooth interpolation. The interpolation of any flow variable $Q_m$ at the mirror node $X_m$ is given by

$$Q_m = \sum_{n=1}^{N} w_n Q_n / q$$

where $n$ denotes a grid node. The total number of grid nodes for doing the interpolation is $N$. The weighting factor $w_n$ is given by

$$w_n = \left( \frac{R - r_n}{R r_n} \right)^p$$

where $p$ is a power parameter that is real and strictly positive. It is equal to 2 by default [88]. The distance between the mirror node $X_m$ and the grid node $X_n$ is given by $r_n = |X_n - X_m|$. Furthermore, the maximum distance $r_n$ is represented by $R$.

$$R = \max_n (r_n)$$

The quantity $q$ is the sum of all the weighting functions.

$$q = \sum_{l=1}^{N} w_l$$

For three-dimensional problem, a first-order interpolation would be given by four points. This is enough to have a three-dimensional linear function. In analogy, to build a three-dimensional quadratic function, which is a second-order interpolation function, ten points would be necessary. As the numerical scheme used in this study is second-order accurate, ten points are taken for the interpolation given in Eq. 4.15.

$$N = 10$$
4.3 Toroidal shape of the jet

4.3.1 Geometrical Characteristics of the 3D near-hole jet

The modeling of the three-dimensional jet surface needs to be accurate from the point of view of the RANS framework. In fact, it is necessary to deflect the incoming freestream flow so as to get the right flow field near the jet, in particular the near-hole static pressure and the freestream streamlines path (important for the downstream entrainment). The main geometrical characteristics of the jet has therefore to be determined between the hole exit and the plane of injection. The cardinal assumption for the modeling of the shape of the jet displacement body is that the cross section of the jet, between the hole exit and the plane of injection, is assumed to be elliptic. Indeed, only jets issuing from cylindrical holes are modeled in this thesis. Thus, the jet starts at the hole exit with an elliptic cross section. It is clear that, in reality, the jet has a tendency to deform from an elliptic to a kidney-type of cross section. This is mainly due to the action of the entrainment of the freestream fluid by the Counter-rotating Vortex Pair (CVP) structure. However, the distance between the hole exit and the plane of injection is of the order of the hole diameter, which is small compared to the total distance covered by the film-cooling jet. As displayed by Abramovich [3], Coehlo and Hunt [21], the cross sectional shape of the jet at one hole diameter downstream of the hole center is still similar to an ellipse even though the CVP structure has already taken action, starting to deform it as a kidney. Between the hole exit and the plane of injection, the jet is behaving almost as a solid body that is bent due to the action of the incoming freestream flow. Then, it becomes obvious that the shape of the jet in this region can be approached by a "bent tube" similar to a curved cylinder, as shown in Fig. 4.7. The axis of this tube is not a line, it has merely a curved axis.

Figure 4.7: A representation of the jet body in the near hole region: a tube with a curved axis.
4.3. TOROIDAL SHAPE OF THE JET

shape. As a function of the flow ratios between the coolant and freestream flow, as well as the angles of injection, the elliptic cross section of the jet may vary toward the streamwise direction, as shown in Fig. 4.8. One of the dominant feature of the jet cross section belongs to its pinching near the trailing edge of the jet. Owing to the cylindrical geometry of the hole, the freestream flow can more easily penetrate downward under the jet near the trailing edge of the hole. Furthermore, the CVP structure starts to take action. This means that the freestream flow starts already to be entrained at the hole exit position. This is therefore a phenomenon to be included in the jet shape model. In summary, the three dimensional shape to represent the near hole jet has to be similar to a tube whose central axis is curved and its cross sectional area is variable. A toroidal shape meets all these geometrical requirements, as shown in the next section. As it will be shown, this toroidal shape is different from a torus because its cross section is an ellipse.

4.3.2 The toroidal shape

A toroidal shape is chosen to represent the jet three-dimensional surface in the near hole region. The mathematical description of the toroidal shape is done in the jet intrinsic frame of reference \((\chi, \xi, \eta)\). It is assumed that the toroidal shape is symmetrical in respect to the \(\eta\)-direction (lateral direction from the point of view of the jet) where the symmetry plane is lying on the \((\chi, \xi)\)-plane, see Fig. 4.9. The big circle, which is the curved axis bearing the solid small ring, has a constant

![Figure 4.8: The penetration of the jet body in the near hole region, from a cross section point of view.](image-url)
4.3. TOROIDAL SHAPE OF THE JET

Center View

Cross View
(small ellipse)

Figure 4.9: Description of the toroidal three-dimensional surface mimicking the jet body near the hole.
4.3. TOROIDAL SHAPE OF THE JET

radius \( R_m \). The center of the big circle is located at point \( X_{tc} (x_0, \zeta_0, 0) \). The big circle goes through the hole exit plane, at the point \( X_{ho} (x_{ho}, \zeta_{ho}, 0) \) and also through the center of the jet \( X_{jc} (x_{jc}, \zeta_{jc}, 0) \) at the plane of injection. The small "circle" of the toroidal shape, which is the circular section that is translated along the big circle axis to create the toroidal surface, is in fact an ellipse. This "small ellipse" represents the cross section of the jet, which may vary in size along the jet path, as explained in the previous section. The two axis of the small ellipse are denoted by \( A (x, \zeta) \) (this is the axis of the small ellipse in the \((x, \zeta)\)-plane) and \( B (x, \zeta) \) (this is the axis of the small ellipse in the \(\eta\)-direction). They actually are an extension in the third dimension \( x \) of the two axis of the ellipse which represents the mixing zone, see Chapter 3. Intuitively, the mathematical description of the toroidal surface is given by

\[
T_{\text{intuitive}} (x, \zeta, \eta) = \left[ \frac{Y(x, \zeta) - R_m}{A(x, \zeta)} \right]^2 + \left[ \frac{\eta - \eta_0}{B(x, \zeta)} \right]^2 - 1 = 0 \quad (4.20)
\]

where

\[
Y(x, \zeta) = \sqrt{(x - x_0)^2 + (\zeta - \zeta_0)^2} \quad (4.21)
\]

\[
R_m = \sqrt{(x_{jc} - x_0)^2 + (\zeta_{jc} - \zeta_0)^2} \quad (4.22)
\]

Obviously, \( \eta_0 \) is equal to zero since the toroidal shape is placed on the \((x, \zeta)\)-plane, where \( \eta \) is equal to zero. To have a more compact expression of the toroidal three-dimensional surface, it is set that

\[
\Omega(x, \zeta) = Y(x, \zeta) - R_m \quad (4.23)
\]

To simplify the notation, the spatial components \((x, \zeta, \eta)\) of the functions are from now on omitted, as they all have been shown. Plugging Eq. 4.23 in Eq. 4.20 and some algebra, the final mathematical expression for the toroidal surface is derived.

\[
T = \eta^2 A^2 + \Omega^2 B^2 - A^2 B^2 = 0 \quad (4.24)
\]

The geometrical place of the toroidal surface is found when the function \( T (x, \zeta, \eta) \) is equal to zero. The complete derivation of the characteristics of the toroidal shape
is given in Appendix A. In particular, it is shown how to compute the numerical values that define the toroidal shape, based on the hole geometry and film cooling jet model. In addition, the numerical immersion of the toroidal shape is explained in Appendix B. In this Appendix, the local search technique to be applied in order to find out where are the ghost-nodes $X_p$ is shown.

4.3.3 3D film-cooling box

The handling of the IBM, used to include the toroidal shape in the computational mesh, is done locally. Indeed, in order not to waist computational resources, a 3D film-cooling box is extracted from the global computational domain, as shown in Fig. 4.10. This three-dimensional box represents the near-hole computational domain, so-called $B_{fcm}$. Typically, the 3D film-cooling box has a streamwise length of $3-4$ hole diameters, a width of $2-3$ hole diameters and a height of $1-2$ hole diameters. The searching and classification of the cells, grid nodes and points that define the toroidal shape are only performed inside this local computational mesh $B_{fcm}$.

4.4 The full CFD-embedded film-cooling jet model

4.4.1 Closure of the model: input data to be fed

For the effective use in a CFD code of the three-dimensional film-cooling jet model using the Immersed Boundary Method, the following steps need to be carried out.

- Set the dimensions of the jet toroidal shape and the location of the plane of injection.
4.4. THE FULL CFD-EMBEDDED FILM-COOLING JET MODEL

- Set the characteristics of the different cells, grid nodes and points inside the 3D film-cooling box.
- Provide the immersed boundary condition $Q_{IBC}$ to be specified at the three-dimensional jet boundaries.

The immersed boundary condition $Q_{IBC}$ at the toroidal surface does not need any external input to be determined. Indeed, $Q_{IBC}$ has been set as a function of the assumptions made concerning the fluid dynamic occurring at the near-hole jet boundary, see section 4.2.3. The immersed boundary conditions $Q_{IBC}$ that need to be specified on the plane of injection are in fact the film-cooling jet model itself, shown in Chapter 3. In order to get the film-cooling jet flow profile on the plane of injection, several input data need to be provided, as it is summarized in Fig. 3.19. Some of these input data concern the geometry of the hole and the others are given by the flow properties of the incoming freestream fluid as well as of the coolant fluid. On one hand, the flow properties of the coolant fluid are supposed to be totally known during a numerical simulation of a film-cooled material. Indeed, there is no simulation of the behavior of the coolant flow in the hole, it is assumed that the user of the film-cooling jet model knows a priori its behavior. On the other hand, the flow properties of the incoming freestream flow can vary and they are primarily determined by the computation. Thus, they need to be probed into the computational mesh during the numerical simulation. The leaning and spatial location of the plane of injection is primarily determined by the hole geometry and the given streamwise position $(X/d)_{jc}$. These features are obviously fixed during the whole computation. The shape and location of the toroid are entirely known if the characteristics and location of the plane of injection are determined.

In summary, the primary input data needed for an effective use of the model in a CFD code are given in Table 4.2.

<table>
<thead>
<tr>
<th>Input by the user</th>
<th>Probe in the computational mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometry $d, \alpha_0, \beta_0, (X/d)_{jc}$</td>
<td>-</td>
</tr>
<tr>
<td>Flow $P_{Te}^c, T_{Te}^c$</td>
<td>$\rho^f, u_f^f, T_f^f, P_s, \delta_{0.99}$</td>
</tr>
</tbody>
</table>

Table 4.2: Input data to the model that need to be probed in the computational mesh.
4.4.2 Probing the computational mesh

There are two types of spatial location where it is needed to probe the flow found by the ongoing numerical computation. First of all, concerning the flow properties of the incoming freestream boundary layer, they are probed in plane perpendicular to the main direction of the incoming freestream flow, lying on the grid, upstream of the location of the hole exit, as shown in Fig. 4.11. The mean value at the upstream plane of any flow properties $\phi$ is found by performing a mass-averaging procedure.

$$\bar{\phi} = \frac{\int_{S_{up}} \rho u \cdot n \phi dS}{\int_{S_{up}} \rho u \cdot n dS}$$  \hfill (4.25)

The boundary layer height $\delta_{0.99}$ might be very tricky to determine if the incoming freestream flow boundary is three-dimensional. This can be the case when the near-hole flow field is affected by secondary flows, shocks, wakes and so on. Michelassi et al. [64] have proposed a very robust method to evaluate the boundary layer height. This method analyzes the absolute vorticity field $\omega(h)$ in the normal direction $h$ to the wall.

$$\left\{ \begin{array}{l}
\delta_{0.99} = h_{lim} \text{ such as } \omega(h_{lim}) = \omega_{lim} \\
\omega_{lim} = \omega_{min} + C_\omega (\omega_{max} - \omega_{min})
\end{array} \right. \hfill (4.26)$$

where $\omega_{min}$ and $\omega_{max}$ are the minimum and maximum vorticity in the profile determined by the direction $h$. In general, the model coefficient $C_\omega$ takes the value

---

5This is explicitly $\rho^f$, $u^f$, $T^f$ and $\delta_{0.99}$
of \( C_\omega = 0.01 \) [64]. The second spatial location to consider is the wall near the hole exit, as shown in Fig. 4.11. This is where the near-hole static pressure \( P_s \) is computed. As there is no flow through the wall, an area-averaging procedure is applied to compute it.

\[
\overline{P_s} = \frac{\int_{S_{pw}} P_s dS}{\int_{S_{pw}} dS}
\]  \hspace{1cm} (4.27)

4.4.3 Full model overview

A detailed overview of the CFD-embedded film-cooling jet model is given in Fig. 4.12. It recapitulates the main steps of the model structure. The inputs to the model that need to be specified by the user are only the hole geometry and the coolant flow conditions. All the other inputs are calibrated so that they are automatically taken into account in the full model.
4.4. THE FULL CFD-EMBEDDED FILM-COOlING JET MODEL

Figure 4.12: Detailed overview of the full CFD-embedded film cooling jet model.
Chapter 5

Experimentally-aided Model Calibration

The experiment that has been used for the calibration of the film-cooling jet model is presented in this Chapter. The experiment has been performed by Bernsdorf [13] in order to provide data for model calibration and analysis of steady and unsteady film-cooling jets. A brief overview of the experimental procedure to acquire the flow data is shown. In this context, Particle Image Velocimetry (PIV) is used to reveal the three-dimensional aerodynamic flow field of a coolant jet near the hole exit. The calibration of the film-cooling jet model as a function of a large range of momentum flux ratios and two different streamwise injection angles is explained. The hole discharge coefficient is modeled with the aid of the measurements provided by Gritsch et al. [38]. The cross section of the jet toroidal body is calibrated so that the conservation of mass is ensured when using the model in a CFD code.

5.1 Experimental setup

5.1.1 Test facility

The test facility, built by Bernsdorf [13], consists of a mainstream closed-loop flow wind tunnel and a secondary air system that delivers cold air to the mainstream through a collection of film-cooling holes, as displayed in Fig. 5.1. Overall, the test facility has four main sections. These are the mainflow loop, the cooling air system, the film-cooled flat plate test section and the PIV system for the measurements.

Mainflow primary loop

The hot air mainflow is driven by a centrifugal compressor connected to a 470 [kW] electrical motor. The impeller is taken from an ABB VTC 254 turbocharger.
5.1. EXPERIMENTAL SETUP

A small blow-out valve is self adjusts the mainflow pressure to ambient. The rotational speed of the impeller sets the pressure ratio i.e. the hot air massflow rate in the system. Theoretically, a maximum hot air massflow of 4 [kg/s] is attainable for a maximum rotational speed of 30'000 [rpm], which corresponds to the supersonic regime. In this thesis, a freestream Mach number of $M_f = 0.3$ is used which gives an impeller rotational speed of about 10'000 [rpm] and an air massflow of 1.5 [kg/s]. Just upstream and downstream of the compressor, the flow temperature is controlled with the use of two heat exchangers. The upstream cooler protects the compressor from overheating while the downstream water-cooled heat exchanger regulates the air temperature between 18 [°C] and 40 [°C]. Upstream of the test section, a steam-air heat exchanger is placed in order to increase the air temperature up to 120 [°C]. This is a 200 [kW] heat exchanger fed by saturated steam at 7 [bar] withdrawn from the building heating system. After the test section, the air is cooled by a water-cooled heat exchanger, which extracts heat from the mainflow. The hot section of the test rig, that is inside of the test section, is wrapped in
5.1. EXPERIMENTAL SETUP

30 [mm] thermal insulation sheets and thermally decoupled from the ambient by ceramic insulators.

**Secondary cold air system**

The air for the coolant flow is delivered from a shop air system. The air is first dried to lower the dew point to -70 [°C]. The dry air goes through a fast self-controlling pressure valve that can very quickly regulate the coolant air massflow. This allows the creation of a pulsating jet. Then, the air flows through a single cycle cooler which can reduce the air temperature down to -63 [°C]. The 12 [m] long secondary piping system is insulated with 40 [mm] sheets. Before reaching the test section, the pipe is attached to a seeding generator system. This seeding generator adds to the coolant air an oil aerosol to be used in connection to the PIV system. Eventually, the coolant air emerges in the mainflow through cylindrical film-cooling holes located in the test section. Seeding particles are also injected in the mainstream section.

**Test section: film-cooled flat plate**

A sketch of the film-cooled flat plate test section is shown in Fig. 5.2. The mainstream air is delivered to the rectilinear channel test section through a bell-mouth. Upstream of the holes, a suction plate is placed to control the boundary layer thickness. The suction plate consists of 327 discrete holes of 1.2 [mm] stacked in 11 arrays over an area of 60 x 180 [mm²]. The air withdrawn from the suction plate is fed back to the mainflow far downstream of the test section. At the rear part of the test section, a two-dimensional diffuser is placed to recover some dynamic pressure. The most interesting part of the test section is the film-cooled flat plate itself. It is placed in a rectilinear channel. It has a length \( L_p \) of 429 [mm], a width \( L_w \) of 181 [mm] and a height \( L_h \) of 40 [mm]. Upstream and downstream of the holes, the flat plate is made out of copper. The top wall, above the flat plat plate, is in fact a 15 [mm] glass window. This is to get an optical access for the PIV laser sheet and cameras. The holes are drilled in a Macor brick that isolate them thermally from the other material parts. There is one row of seven cylindrical holes. Each hole has a diameter of 5 [mm]. They are aligned to the streamwise direction so that no lateral injection is considered. There are two sets of holes available, with a variable streamwise injection angle, namely \( \alpha_0 = 30° \) and \( \alpha_0 = 50° \). The hole dimensions are summarized in Table 5.1.
5.1. EXPERIMENTAL SETUP

Figure 5.2: Schematic of the test section, from Bernsdorf [13].

<table>
<thead>
<tr>
<th>Set</th>
<th>Diameter [mm]</th>
<th>Number of holes</th>
<th>$a_0$ [$^\circ$]</th>
<th>$b_0$ [$^\circ$]</th>
<th>l/d [-]</th>
<th>s/d [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5</td>
<td>7</td>
<td>30</td>
<td>0</td>
<td>2.8</td>
<td>4.0</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>7</td>
<td>50</td>
<td>0</td>
<td>2.8</td>
<td>4.0</td>
</tr>
</tbody>
</table>

Table 5.1: Hole dimensions for the two available sets

5.1.2 Measurement technique: PIV

A brief overview of the PIV technique used to reveal the three-dimensional aerodynamic flow field of the coolant jet near the hole exit is given in this section. The measurements have been carried out by Bernsdorf [13], who gives a very detailed explanation of the PIV set up used.
5.1. EXPERIMENTAL SETUP

PIV system

The PIV set up is based on a commercially available stereoscopic system. Two CCD cameras with a resolution of 1280 x 1024 pixels and a dynamic storage of 12 bits is mounted on a 2-axis high accuracy traversing system, as shown in Fig. 5.3. A double-pulsed Nd:YAG laser, with a power of 120 [mJ] per pulse, is mounted on a additional third traversing axis. The laser is pointing perpendicular to the flat plate surface. A 45° mirror, placed to the left side of the holes, reflects the laser light sheet so as to get the light sheet parallel to the flat plate surface. The typical thickness of the light sheet is 1 [mm]. The two cameras look into the test section through the glass window. They have two different angles to the flat plate surface. The whole system is connected to two PC computers. One is dedicated to synchronize the system and buffering the collected data whereas the second one serves for storing and archiving the collected data.

Measurement technique

The three-dimensional velocity field is taken in a three-dimensional volume whose dimensions are around 9 hole diameters in streamwise direction, 7 hole diameters
on lateral direction and 1.5 hole diameter in vertical direction. The measured flow volume is given by the piling up of all the two-dimensional measurement planes, obtained at different heights. Before the measurement, a warm up of the entire system takes about 3 hours. During that time, the main flow parameters (pressure, temperature, mass flow, Mach number) are checked and set. Then the seeding of particle is released into the system. The field view of a camera is around $45 \times 36 \, [\text{mm}^2]$ on the light sheet. A set of double pictures, separated by 2.5 $[\mu\text{s}]$, is taken throughout all the experiment. In general, a seeding particle travels around 8 pixels between two camera shots\(^1\). The measurements cells have a dimension of $32 \times 32$ pixels. A cross-correlation technique is carried out on each individual cell to get one final three-dimensional velocity vector. The final flow volume consists of 10 to 12 planes with $79 \times 63$ velocity vectors, giving a total number of roughly 50'000 to 60'000 data points.

### 5.1.3 Investigated flow regimes

All experimental results presented are averaged over 20 to 30 double frame shots. The freestream flow conditions have been set constant during the whole measurement campaign. The main freestream flow parameters are given in Table 5.2. The freestream boundary layer characteristics (boundary layer momentum thickness $\theta_f$ and shape factor $H_f$) is measured using a 3-holes Cobra-head probe GEM 3H 2, see Bernsdorf [13]. The first measurement campaign focuses on a steady film-cooling jet. A large number of different density ratios $DR$ and blowing ratios $BR$ have been investigated for two streamwise injection angle $\alpha_0$. The aim is to provide a database of the three-dimensional flow field near the hole exit for the model coefficient calibration. Table 5.3 summarizes all the different coolant to freestream flow regimes investigated. These different flow regimes are represent-

\[^1\text{based on a freestream flow velocity of } 110 \, [\text{m/s}]\]

<table>
<thead>
<tr>
<th>Momentum thickness $\theta_f / d$ [-]</th>
<th>Shape factor $H_f$ [-]</th>
<th>Mach number $M_f$ [-]</th>
<th>Reynolds number $Re_{L_f}$ [-]</th>
<th>Total pressure $P_T^f$ [bar]</th>
<th>Total temperature $T_T^f$ [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.051</td>
<td>2.3</td>
<td>0.3</td>
<td>$2.8 \cdot 10^6$</td>
<td>1</td>
<td>300</td>
</tr>
</tbody>
</table>

Table 5.2: Freestream flow parameters.
5.2. Calibration of the film-cooling jet model

The film-cooling jet model coefficients that need to be tuned are recapitulated in Table 5.4. It is evident that the number of model coefficients to be tuned is relat-
5.2. CALIBRATION OF THE FILM-COOLING JET MODEL

ivamente large. Furthermore, these model coefficients may be, *a priori*, dependent on a large number of different quantities, from hole geometry\(^2\) to flow ratios\(^3\). Thus, some assumptions are made so as to simplify the calibration process.

5.2.1 Assumptions made for the calibration

**Location of the plane of injection**

The streamwise position \(X/d\) where the *plane of injection* should stand is first set. At this location, the model is calibrated. The plane of injection is arbitrarily placed just downstream of the hole exit so that

\[
\begin{align*}
(X/d)_{\alpha_0=30^\circ} &= 1.25 \\
(X/d)_{\alpha_0=50^\circ} &= 1.00
\end{align*}
\]  

(5.1)

**Streamwise injection: jet symmetry**

The film-cooling jet is considered symmetrical with respect to the \(\eta\)-direction. This assumption has already been used for the definition of the jet toroidal shape. Indeed, this assumption is close to the reality when observing film-cooling jets which are injected strictly in the streamwise direction (\(\beta_0 = 0\)). This means that the jet flow field is mirrored through the \((\xi, \eta)\)-plane, that is

\[
\begin{align*}
B_{m1} &= B_{m2} \\
B_{w1} &= B_{w2} \\
\Gamma^*_1 &= \Gamma^*_2 \\
a_{1}\gamma &= a_{2}\gamma \\
(\xi_{1}\gamma, \eta_{1}\gamma) &= (\xi_{2}\gamma, -\eta_{2}\gamma)
\end{align*}
\]  

(5.2)

**Lateral injection: redistribution of vorticity**

For film-cooling jets that have a lateral injection component (\(\beta_0 \neq 0\)), it is observed that they become asymmetric, see for instance Jung and Lee [48]. In a first attempt to deal with lateral jets also, without complicating the calibration process, it is assumed that the circulation of the two vortices of the CVP structure is

---

\(^2\)such as \(l/d, s/d, \alpha_0, \beta_0\)

\(^3\)such as \(DR, BR, IR\)
redistributed as follows

\[
\begin{align*}
\Gamma_{1r}^* &= \Gamma_{sym}^* \left[ 1 + (\sin \beta_0)^2 \right] \\
\Gamma_{2r}^* &= \Gamma_{sym}^* \left[ 1 - (\sin \beta_0)^2 \right]
\end{align*}
\]  

(5.3)

where \( \Gamma_{sym}^* \) is the circulation computed for the same jet without lateral injection. It has been systematically observed [48] that, for lateral injection, the "front side" vortex becomes much weaker than the "back side" vortex of the CVP structure\(^4\), see for instance Chapter 8. Besides the redistribution of vorticity suddenly occurs which explains the \( \sin^2 \) square function. Note here that there is no general rule yet concerning the redistribution of the vorticity at the two sides of the CVP structure. The selected \( \sin^2 \) square function is a tentative to model this effect but it should be confirmed by further experimental studies. All the other model coefficients are identical to the ones used for streamwise injection. The consequences of these assumptions for lateral injection are discussed in Chapter 8.

**Functional variables: IR and \( \alpha_0 \)**

In the available experimental database, the flow behavior of a film-cooling jet near the hole is provided for two different streamwise injection angles \( \alpha_0 \) and for different density ratios \( DR \) and blowing ratios \( BR \). Thus, it is assumed that all the model coefficients are functionally dependent on these quantities only. No assumption can be made concerning the impact of other geometrical or flow quantities because they have not experimentally been investigated. The impact of the density and blowing ratios upon the jet behavior is taken into account in a condensed form, that is the momentum flux ratio \( IR \).

\[
IR = \frac{BR^2}{DR}
\]  

(5.4)

The choice of the momentum flux ratio \( IR \) as the main variable setting the numerical value of the model coefficients is based on several reasons. Firstly, it is easier to tune the model coefficients as a function of two variables (\( \alpha_0 \) and \( IR \)) than as a function of three variables (\( \alpha_0, DR \) and \( BR \)). Secondly, the momentum flux ratio scales the level of the dynamical force of coolant relatively to the freestream flow. The relative level of the dynamical force (flow momentum) between the coolant

\(^4\)The "front side" side vortex is in front of the incoming freestream whereas the "back side" vortex is located at the rear part of the hole
and the freestream is a first order flow effect. Thus, the momentum flux ratio should be the most appropriate variable to describe the type of flow interaction which occurs between the coolant jet and the freestream flow. In fact, Bernsdorf et al. [12] provide a visualization of the velocity field of a coolant jet in the near hole region, for different density ratios and blowing ratios. It is clearly seen that the coolant jet behavior strongly correlates with the momentum flux ratio, see also experimental data and modeling results in Appendix D. As a consequence, it is assumed that all the model coefficients are only function of the streamwise injection angle $\alpha_0$ and the momentum flux ratio $IR$. As only two streamwise injection angles are available, the calibration of the model coefficients is done twice, for each angle. This actually simplify the calibration process. Indeed, for each fixed streamwise injection angle $\alpha_0$ case (that is for $\alpha_0 = [30^\circ, 50^\circ]$), the model coefficients are assumed to be only function of the momentum flux ratio $IR$. When the film cooling jet model is used for a streamwise injection angle $\alpha_0$ where it has not been expressly calibrated (i.e. $\alpha_0 \neq [30^\circ, 50^\circ]$), the model coefficients are linearly interpolated from their known value at $\alpha_0 = [30^\circ, 50^\circ]$, see for instance Chapter 8 and Chapter 10.

Types of functional dependencies

There are three different types of functional dependencies that are considered. In the first place, a model coefficient can be insensitive to the variation of the momentum flux ratio, so that it is equal to a constant value $c$. In the second place, a model coefficient can be a linear function of the momentum flux coefficient: its rate of change as a function of the momentum flux ratio is constant. In the third place, the effect of a model coefficient (i.e. of a sub-model) upon the global jet flow field can change as a function of the momentum flux ratio. This represents a typical transition process from one type of flow behavior to another one. Indeed, to the author’s view, a typical film-cooling jet is ranging from an almost plane jet (very low $IR$) to a free jet in cross flow (very high $IR$). This means that the flow physics of the coolant jet may vary as a function of the momentum flux ratio. Note that the rate of change of the jet flow physics may also vary as a function of the momentum flux ratio. This issue is not directly addressed in the modeling, in order to not add too much complexity, but it is attempted to be taken into ac-

---

5Only the behavior of the wake region does not strongly correlates.
6For example, the impact of the wake flow damping upon the coolant jet flow structure seems to be weak for a low momentum flux ratio case whereas it seems to be strong for a high momentum flux ratio case, see for instance Appendix D.
5.2. CALIBRATION OF THE FILM-COOlING JET MODEL

Count in the calibration. For this purpose, a power law is chosen because it brings both flexibility (the power index can be tuned) and simplicity (only the power index and the argument of the function need to be tuned). In summary, the three different types of functional dependencies of any model coefficient $\phi$ are

\[
\begin{align*}
\phi &= c_0 \\
\phi &= c_0 + c \cdot IR \\
\phi &= c_0 + c \cdot IR^n
\end{align*}
\] (5.5)

It is obvious that there is a part of arbitrariness in the choice of these functional dependencies. However, there is one argument that strongly support the choice of these functional dependencies. Globally, the flow structure of a coolant jet is monotonically changing as a function of the momentum flux ratio, at least in the range of $IR$ studied, see for instance Appendix D. This means that the model coefficient should vary monotonically as a function of the momentum flux ratio. In this context, the simplest case is a constant functional dependency, then a linear functional dependency and finally a non-linear functional dependency.

5.2.2 Calibration process and results

A priori, it is difficult to determine the behavior of the model coefficients as a function of the momentum flux ratio. In addition, there are $16^7$ coefficients that need to be tuned. This is certainly a very large number of different coefficients to take care which furthermore interact between each other. This means that it is not possible to find a unique and mathematically well-defined solution for the numerical value of the model coefficients. Thus, it becomes obvious that it is somehow needed to observe and to find out what is the bulk behavior of the main flow features of the film-cooling jet as a function of the momentum flux ratio. This can help to determine how the model coefficients interact with each others.

Qualitative observation of the film-cooling jet behavior

The first step that is carried out to perform the model calibration is to systematically inspect the measured velocity field at the position of the plane of injection for the different momentum flux ratios and streamwise injection angles, see Appendix D. Fig. 5.4 illustrates a model made out of this flow investigation. It is important

\[\text{that is } C_{n,\alpha,\beta} A_{m1,2}, B_m, A_{w1,2}, B_w, \Gamma_w^{n,T} (\xi_w, \eta_w), \Gamma_{1r}, \alpha_{1r}, \xi_{1r}, \eta_{1r}\]
5.2. CALIBRATION OF THE FILM-COOLING JET MODEL

Figure 5.4: Evolution of the main flow features of a film-cooling jet as a function of the momentum flux ratio.

to underline that the proposed model describing the behavior of the model coefficients is not unique. Furthermore, the model illustrated in Fig. 5.4 is not based on quantitative but on qualitative observations. Indeed, owing to the fact that, on one hand, the principle of superposition is used for the modeling of the jet and, on the other hand, the interaction between the coolant and the freestream flow is non-linear, it is not possible to decouple explicitly the measured flow field for each sub-model. Based on this fact, the center of the jet is ill-defined. Hence, it has been set that the center of the jet is located at the same height as the two lateral high velocity cores (see Appendix D for a detailed view). This allows to calibrate the pressure flow resistance coefficient \( C_{n_{\alpha}} \). Then, it is proposed that the vertical distance between the center of the two counter-rotating vortices and the center of the jet is increasing with the momentum flux ratio. The vertical distance between the center of the wake and the center of the jet (that is \( \xi_w \)) is approximated to stay constant\(^9\). The mixing limit \( (A_{m1,2}, B_m) \) of the jet is also set constant. Indeed, in observing the different measured coolant flow profiles, it seems that the wake zone is much more changing in size and intensity than the overall jet mixing region.

\(^8\)The lateral flow resistance coefficient \( C_{n_{\beta}} \) is calibrated with the use of the measurement performed by Lee and Jung [48], see also Chapter 8. Based on measurements of film-cooling effectiveness obtained for different lateral injection angles \( \beta_0 \), the lateral path of the coolant jet are inferred. These experimental jet trajectories are used to tune the jet trajectory model, so that the flow resistance coefficient \( C_{n_{\beta}} \).

\(^9\)Here the center of the wake is ill-defined and cannot be quantified. So that, it has been qualitatively observed that the vertical distance between the local velocity deficit (the "wake center") and the center of the jet stays constant.
Thus, the axes \((A_w, B_w)\) of the ellipse representing the wake mixing region are set to vary as a function of the momentum flux ratio. The lateral position \((\eta_{1r,2r})\) of the CVP is observed to stay pretty much constant relatively to the jet center position. Meanwhile, the CVP strength (circulation coefficient \(\Gamma_{1r,2r}^T\)) is strongly decreasing with the momentum flux ratio, which corresponds to the circulation scaling analysis done in Chapter 3, see Eq. 3.35.

The wake intensity \(I_w^T\) for the temperature field cannot be calibrated since no information is provided in the experiment of Bernsdorf [13] concerning the thermal field. In order to solve this issue, the calibrated velocity profile is applied to the experimental measurement of Rydholm [75], see Appendix E. In this experiment, the near-hole aerothermal flow field of the film-cooling jet is measured at similar flow regimes and for almost the same hole geometry. Using these experimental results and the calibrated velocity field, the tuning of the thermal wake intensity coefficient \(I_w^T\) can be done. It is found that the wake intensity is very small; at the location of the \(plane of injection\), the CVP structure has not taken full action yet, which is in agreement with the assumptions made in Chapter 3, section 3.1.4.

**Calibrated functional dependencies of the model coefficients**

An iterative calibration procedure is carried out by taking into account the flow observation made on the \(plane of injection\) using the PIV experimental data. The iterative process consists of guessing the value of the model coefficients and visually comparing the obtained jet flow profile with the experimental data. In fact, it has first been considered to use an automatic least square method to find out all the model coefficients. However, it has soon been recognized that the combination of the high number of model coefficients to be tuned (16), the non-linear interactions between the sub-model and the three-dimensional flow pattern recognition that should be made certainly lead to have at hand a very complex algorithm in order to find a relevant result. Since the development of such an algorithm was not the main objective of this thesis, it has been decided that the most efficient (but still time consuming) technique to calibrate the model coefficients is a simple visual inspection. By iterating between some numerical values of the model coefficients and the resulting flow field on the \(plane of injection\), a final result is found for all momentum flux ratios. In particular, it is ensured that the circulation and location of the CVP structure obtained in the calibrated model correspond to the experimental data. The final result of the calibration process is shown in Appendix F for \(\alpha_0 = 30^\circ\) and in Appendix G for \(\alpha_0 = 50^\circ\). The three-dimensional flow fields obtained with the calibrated film-cooling jet model on the \(plane of injection\) for
5.3 Calibration for the Conservation of Mass

5.3.1 Discharge Coefficient

The procedure for the determination of the actual coolant massflow $\dot{m}_{\text{actual}}$ to be injected has been described in section 3.4.1 of Chapter 3. In order to effectively compute the actual coolant mass flow, the near-hole static pressure $P_s$ as well as the hole discharge coefficient $C_d$ need to be known. The near-hole static pressure is probed in the computational domain, see section 4.4.2 of Chapter 4. It is therefore necessary to have a model for the hole discharge coefficient as a function of the coolant to freestream flow regime. For this purpose, the measurements of Gritsch et al. [38] are utilized. A broad range of different flow regimes are investigated to characterize the discharge coefficient of a cylindrical hole having a streamwise injection angle of $\alpha_0 = 30^\circ$. In particular, discharge coefficients as a function of the total to static pressure ratio $\left(\frac{P_T}{P_s}\right)$ through cylindrical holes are provided for different freestream Mach numbers. These experimental correlations are used to derive a calibrated model for the discharge coefficient, for a freestream Mach number of $M^f = 0.3$.

$$C_d = c_0 + c \cdot \left(\frac{P_T}{P_s} - 1\right)^n$$

where the coefficient $c_0$ is equal to 0.00, $c$ is equal to 0.83 and $n$ is equal to 0.07. Eq. 5.6 is commonly used to determine the discharge coefficient $C_d$.

5.3.2 Cross section of the toroidal jet body

In order to verify the law of the conservation of mass, the overall coolant flow profile specified on the plane of injection should correspond to the actual coolant mass flow to be injected. This can be formally written as

$$\dot{m}_{\text{actual}}^c = \int_{S_{pi}} \rho u \cdot ndS$$

where $S_{pi}$ is the total surface of the plane of injection. Now, the total surface of the plane of injection is delimited by the cross section (small ellipse) of the toroidal jet body, see section A.2 of Appendix A. The small ellipse of the jet
toroidal body on the plane of injection is characterized by three coefficients, $A_1$, $A^{jet}_2$ and $B^{jet}_2$, which are its semi-axes. Looking at Eq. 5.7 and remembering that the film-cooling jet model is given by a superposition of different non-linear models, it is very unlikely that the actual coolant mass flow can be integrated analytically. As a consequence, in order to calibrate the axes of the small ellipse of toroidal body, an a posteriori numerical integration of the mass flow on the plane of injection is performed iteratively. In this iteration process, the axes of the small ellipse of the toroidal shape are guessed so that the total area $S_{pi}$ of the plane of injection is known. This allows to perform the numerical calculation of Eq. 5.7 on a given grid. This is done for all the momentum flux ratio studied. As a result, a calibrated functional dependency of the axes of the small ellipse of the toroidal body is derived, see Appendix H. The mass flow error, that is the difference between the actual coolant mass flow given by the model and the one found by integration of Eq. 5.7, has been ensured to be less than 1% for any momentum flux ratio, see also Chapter 6.
Chapter 6
Performance Issues

This Chapter is dedicated to the performance issues related to the use of the film-cooling jet model in a CFD code for different computational mesh sizes. This performance analysis is carried out strictly in the framework of the numerical method employed in this study, coupled with an isotropic, algebraic turbulence model, namely Baldwin-Lomax. A grid independency study is presented in order to determine the best-suited mesh density for an optimal use of the film-cooling jet model in a CFD code. To this effect, a systematic survey of grid-independent solution versus computational cost is performed for a range of different mesh densities. An engineering solution in using the model for design purpose is proposed. The geometry and flow used in this performance study are based on the film-cooled flat plate test rig of Bernsdorf [13].

6.1 Computational domain

6.1.1 Geometry

The geometry of the computational domain is basically represented by a rectilinear three-dimensional channel as shown in Fig. 6.1. There is absolutely no mesh inside the hole due to the fact that the CFD-embedded film-cooling jet model is utilized for the coolant injection. This means that the presence of the hole is only virtual. For obvious computational resources saving, the seven holes in the row\(^1\) are not computed together. In fact, it is assumed that the row is infinite so that only one hole passage is taken into account. Thus, the width of the computational domain is given by the hole to hole pitch distance \(s/d\). For the same reason, the top side of the computational domain is not placed at the glass window surface but below it, that is inside the channel where it is assumed that the flow is quasi one-dimensional. Hence, there is no need to solve the boundary layer flow close to the

\(^1\)see section 5.1.1 for more description of the film-cooled flat plate geometry
glass window. The inlet and outlet cross planes are located far from the hole which ensures a very low disturbance of the inlet and outlet boundary conditions upon the solution accuracy near the hole. The geometrical size of the computational domain is listed in Table 6.1. The steady flow set S4 (see Table 5.3 in Chapter 5) is selected. Unfortunately, the density ratio \( DR = 1.20 \) is a bit smaller (4.7\%) than in the experiment \( DR = 1.26 \), because of a slight error in the specification of the coolant total temperature. However, it does not affect the quality of the results since the error is small. In addition, the study of the computational performance of the model is not dependent on the experimental data. The blowing ratio is \( BR = 2.0 \).

<table>
<thead>
<tr>
<th>Dimension [-]</th>
<th>Width ( L_w/d )</th>
<th>Height ( L_h/d )</th>
<th>Upstream length ( L_u/d )</th>
<th>Downstream length ( L_d/d )</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>5</td>
<td>14</td>
<td>27</td>
<td></td>
</tr>
</tbody>
</table>

*Table 6.1: Geometrical size of the computational domain.*
6.1.2 Boundary conditions

The film-cooled flat plate surface is represented by a no-slip, adiabatic wall boundary condition. As an infinite row of holes is considered, a periodic boundary condition is applied to the two lateral sides of the computational domain. A slip boundary condition is assumed at the top side of the computational domain. Indeed, this boundary is standing inside the real channel, in the freestream flow region. Furthermore, it is assumed that it is located far from the hole so that the flow is quasi-rectilinear and does not affect the near hole region. Meanwhile, the slip boundary condition on the top side mirrors the coolant jet out of the hole, which doubles the blockage due to the jet. However, the mirrored jet is located at 10 hole diameters above the computed jet. This is higher than the height of the experimental channel (8 hole diameters). Thus, the blockage effect of the mirrored jet is negligible. At the inlet and outlet planes, the boundary conditions are given by the experiment, see Chapter 5, section 5.1.3 for more information.

6.1.3 Computational mesh

The grid density in the direction $Z$, normal to the flat plate, remains unchanged for all mesh densities tested. Indeed, the characteristics of the boundary layer flow trigger the mesh density to be set in this direction. As a two-layer turbulence model is utilized, its proper use leads to impose that the non-dimensional wall coordinate $Z^+$ of the first grid node in the normal direction has to be approximately equal to 1 ($Z^+ \approx 1$), see Appendix I for more information. Furthermore, it is ensured that about 15 grid nodes in the normal direction are located within the boundary layer region. In this region, a constant distance between two grid nodes in the normal direction is ensured to perfectly preserve the second order accuracy of the Ni-Lax-Wendroff algorithm. In total, there are 51 grid nodes in the normal direction. The mesh density in the lateral and axial (streamwise) direction is varied in order to analyze its impact upon grid-independent solution versus computational cost. The mesh density is characterized by $N_S$, which represents the number of grid nodes in the $S$-direction ($S = X, Y$) per hole diameter. In the near-hole and downstream region, that is between $X/d = -4$ to $X/d = +27$, the grid node-to-node distance in the axial and lateral directions is determined by $N_S$. In the upstream region, that is between $X/d = -14$ to $X/d = -4$, the node to node distance remains constant ($N_{X,Y} = 4$) whatever the mesh density used downstream is. The minimum axial density tested is $N_X = 1$. The minimum lateral density tested is $N_Y = 5$. Indeed, less grid nodes per hole diameter would mean that the CVP structure would not be
captured at all. In total, 20 different mesh densities are tested. The different mesh densities tested are listed in Table 6.2 with the corresponding total number of grid nodes and the identity code of the mesh (e.g. A1 means $N_X = 1$ and $N_Y = 5$).

<table>
<thead>
<tr>
<th>$N_Y$</th>
<th>$N_X = 1$</th>
<th>$N_X = 3$</th>
<th>$N_X = 5$</th>
<th>$N_X = 9$</th>
<th>$N_X = 13$</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>A1 53'754</td>
<td>B1 80'631</td>
<td>C1 134'385</td>
<td>D1 241'893</td>
<td>E1 349'401</td>
</tr>
<tr>
<td>9</td>
<td>A2 104'346</td>
<td>B2 156'519</td>
<td>C2 260'865</td>
<td>D2 469'557</td>
<td>E2 678'249</td>
</tr>
<tr>
<td>13</td>
<td>A3 154'938</td>
<td>B3 232'407</td>
<td>C3 387'345</td>
<td>D3 697'221</td>
<td>E3 1'007'097</td>
</tr>
<tr>
<td>17</td>
<td>A4 205'530</td>
<td>B4 308'295</td>
<td>C4 513'825</td>
<td>D4 924'885</td>
<td>E4 1'335'945</td>
</tr>
</tbody>
</table>

Table 6.2: Identity code, axial $N_X$ and lateral $N_Y$ mesh densities and total number of grid nodes.

6.2 Computational history

6.2.1 Computational convergence

It is verified in this section that stable and converged solutions are found when using the three-dimensional film-cooling jet model embedded in a CFD code. This is shown for a broad range of computational mesh densities.

Computational hardware

All the computations presented in this study have been carried out on the LSM NEC PC cluster [17], using MULTI3 [15] as the CFD code (see Chapter 2). Each computation was run on one CPU as no parallel version of MULTI3 was available at this time. Each CPU consists of an Intel Pentium 4 processor with a 2.4 GHz speed and having a 800 MHz bus speed. The maximum RAM space available is 2Gb (no swapping).
6.2. COMPUTATIONAL HISTORY

Computational stability

The numerical immersion of the three-dimensional jet model could be successfully carried out within all the mesh densities tested. A stable and converged solution is obtained, similar to what can be obtained without any injection. Before going any further, it is first intended to give to the reader a qualitative view of the successful immersion of the three-dimensional jet model in the computational mesh, for different representative mesh densities. A three-dimensional view of the surface points $X_s$ describing the toroidal jet surface and the plane of injection is shown in Fig. 6.2, for the mesh densities B1, C2, D3 and E4. The center plane $(Y/d = 0)$ is superimposed upstream and upon the jet surface, where freestream streamline can be seen. Furthermore, at the axial position $(X/d = 1.25)$ of the plane of injection, a cross section is displayed where the cross velocity vectors can be visualized. It can be observed that for very fine meshes such as D3 and E4, the number of surface point $X_s$ to be treated is quite large (1300 and 1990 surface points for D3 and E4 respectively) whereas it is less for C2 (696 surface points) and much less for B1 (285 surface points). Overall, there is already a clear feeling that the rate of change in the description of the toroidal jet surface and the plane of injection seems higher when going from mesh B1 to C2 than from mesh D3 to E4.

Residuals history

The Root Mean Square (RMS) residual $DQ_{rms}$ histories obtained during the calculation process, for different mesh density cases, are plotted in Fig. 6.3. In detail, the convergence histories are compared for different axial mesh densities in Fig. 6.3-I (left) and for different lateral mesh densities in Fig. 6.3-II (right). The RMS residual $DQ_{rms}$ is defined as being the total root mean square sum of the residuals $DQ^2_i$ of each equation $i$ ($i = 1..5$) to be solved at each grid node $k$ ($k = 1..knodes$).

$$DQ_{rms} = \sqrt{\frac{\sum_{i=1}^{5} \sum_{k=1}^{knodes} DQ^2_i (k)}{5 \cdot \sum_{k=1}^{knodes} k}}$$  \hspace{1cm} (6.1)

Looking first at Fig. 6.3-I, one can see that the RMS residual reaches its lowest value for the coarsest axial mesh density case (A2). When the axial mesh density increases, the RMS residual monotonically increases, for a constant iteration

\hspace{1cm}2Here, $DQ_i$ is the same as the $\delta Q$ in the right hand side of Eq. 2.43 in Chapter 2
number. Furthermore, it seems that the convergence starts to be very similar for an axial mesh spacing of $N_X = 5$ (C2 case) or higher. The same trend is found for other lateral mesh density cases. The effect of the lateral mesh spacing on the convergence rate is given in Fig. 6.3-II for a fixed axial mesh density (C cases, that is $N_Y = 9$). Same type of results are found for other axial mesh density cases. It is observed that the convergence is very similar whatever the lateral mesh spacing is, except for the coarsest lateral case (C1) where the residual reaches a smaller value, for a constant iteration number. In general, it is well known that calcula-
6.2. COMPUTATIONAL HISTORY

Figure 6.3: Root Mean Square residual histories for different mesh densities: as a function of axial mesh density, based on a fixed lateral mesh density ($N_Y = 9$) (left -I) and as a function of lateral mesh density, based on a fixed axial mesh density ($N_X = 5$) (right -II).

...tions on coarse meshes lead to a very high artificial numerical dissipation of the small flow structures. A coarse mesh acts similarly to an extra numerical (artificial) viscosity because flow gradients are too coarsely represented. Small flow perturbations are not necessarily dissipated at the same rate in fine meshes, they are actually less dissipated. Indeed, they keep traveling back and forth through the computational domain because they are captured by the mesh resolution. They are especially concentrated inside low Mach number regions such as in the boundary layer. Indeed, in these regions the local time step (see Chapter 2) is much smaller, compared to freestream regions. Thus, the rate of convergence is reduced. As a conclusion, having a global view of the RMS residual histories displayed in Fig. 6.3, it is verified that the proposed Immersed Boundary Method used to include the three-dimensional film-cooling jet model in a computational mesh leads to stable and converged solutions for a wide range of mesh densities.

6.2.2 Computational time

Total computational time

The total computational time to get a fully converged solution is compared in Fig. 6.4 for all the different mesh densities tested. A fully converged solution is declared when the RMS residual is lower than $10^{-5}$ and the global mass flow error
evolution attains a stable behavior, see section 6.3.1. The total number of iterations ranges from 9'000 for A1 through 29'000 for E4\(^3\). In observing Fig. 6.4, it is found that the total computational time is exponentially increasing with the linear increase of the mesh refinement in axial and/or lateral direction. In addition, except for cases C4, D3, D4, E3 and E4, a fully converged solution is found in less than 10 hours, \textit{i.e.} overnight. This total computational time does indicate how much time in total it is needed to get a solution but does not provide any information whether or not the use of the three-dimensional film-cooling jet model is computationally efficient.

\textbf{Computational overhead when using the model}

In order to quantify the loss of computational time due to the use of the model, the computational overhead \(\sigma\) is introduced

\[
\sigma = 100 \cdot \left[ \frac{\Delta t_{fcm} - \Delta t_{noc}}{\Delta t_{noc}} \right] \quad (6.2)
\]

\(^3\)At a first glance, it could be argued that the total number of iterations is high. However, the numerical method used herein is explicit time marching, so that each iteration takes little time to be executed. Note also that at the time of the work, no multigrid method has been implemented in the numerical algorithm.
where $\Delta t_{fem}$ and $\Delta t_{noc}$ are the times to iterate once the whole numerical algorithm to compute the flow field in the channel with and without a coolant jet respectively. The time has been averaged over 200 iterations chosen randomly in the convergence history. The overhead $\sigma$ gives the percentage of computational time lost due to the use of the model, for one infinite row of holes, in a steady flow environment. Although one hole is effectively computed the addition of $n$ holes at the sides of the computed hole adds $n$ time the computational mesh used. So that the computational overhead analyzed here concerns effectively one row of holes. The computational overhead is plotted in Fig. 6.5 as a function of all the different mesh densities tested. First of all, it is noticed that the overhead for

![Figure 6.5: Overhead in using the model as a function of the mesh density.](image)

the mesh density case $A$ ($N_X = 1$) is behaving differently from the other mesh density cases. Actually, the overhead for the case $A$ is monotonically increasing with the increase of the lateral mesh density. Furthermore, the overhead in case $A$ is shown to be relatively high, having its lowest value equal to 2.1% for $A1$. Concerning the overhead for mesh density cases $B$ through $E$, it is observed the same functional pattern. The overhead is the highest ($3.3 < \sigma < 5.0$) for the lowest lateral mesh spacing and then decreases down to small values ($0.3 < \sigma < 1.2$) for the fine lateral mesh density case ($N_Y = 13$). Finally, it increases again for the highest ($N_Y = 17$) lateral mesh density case ($1.2 < \sigma < 1.8$). Overall, it seems that the mesh density cases $C$ and $D$ give the best overhead whatever the mesh density used. Indeed, the overhead is proposed to be qualitatively classified in
three categories and tabulated in Table 6.3.

<table>
<thead>
<tr>
<th>Good</th>
<th>Medium</th>
<th>Poor</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma &lt; 1.3$</td>
<td>$1.3 &lt; \sigma &lt; 3.3$</td>
<td>$3.3 &lt; \sigma$</td>
</tr>
<tr>
<td>A1 A2 A3</td>
<td>B1 B2 B3</td>
<td>A4</td>
</tr>
<tr>
<td>C1 C2 C3 C4</td>
<td>D1 D2 D3 D4</td>
<td>B4</td>
</tr>
<tr>
<td>E1 E2 E3</td>
<td></td>
<td>E4</td>
</tr>
</tbody>
</table>

Table 6.3: Qualitative classification of the overhead level as a function of the mesh density.

### 6.3 Grid-independent solution

#### 6.3.1 Mass Flow Conservation

The conservation of mass flow is first checked for all the mesh densities computed. At this point, two different types of mass flow error are distinguished, namely a *global* and a *local* mass flow error.

**Global mass flow error**

In the first place, a global mass flow error $\varepsilon^G_m$ is defined

$$
\varepsilon^G_m = 100 \cdot \left[ \frac{\dot{m}_{out} - \left( \dot{m}_{in} + \dot{m}_{eff}^C \right)}{\dot{m}_{in}^f + \dot{m}_{eff}^C} \right]
$$

(6.3)

where $\dot{m}_{in}^f$ and $\dot{m}_{out}$ are the mass flow at the channel freestream inlet and outlet planes respectively. The coolant mass flow effectively injected on the *plane of injection* is denoted by $\dot{m}_{eff}^C$. The global mass flow error determines the error between the total mass flow injected versus the total mass flow going out of the computational domain. The evolution of the global mass flow error $\varepsilon^G_m$ during the iterative history is plotted in Fig. 6.6 for a sample of representative mesh densities (left - Fig. 6.6-I). The final global mass flow error obtained for all mesh density cases computed is given in Fig. 6.6-II (right). In Fig. 6.6-I,
6.3. GRID-INDEPENDENT SOLUTION

Figure 6.6: Global mass flow error history for a different mesh densities (left - I) and Global mass flow error found in all mesh density cases results (right - II).

the global mass flow error is observed to stabilize during the iterative history for all types of mesh density calculations. A small periodic oscillating behavior last for computations with high mesh densities. This is to be put in connection with the comments made previously about the residuals history, that is in fine meshes, small flow perturbations somehow keep traveling back and forth even when the computation is said to be converged. This is especially true for central scheme, such as the one used in this study, even if an artificial viscosity is added. In Fig 6.6-II, it is shown that for the mesh densities C to E, the global mass flow error is about 1%, which is typical when using the Ni-Lax-Wendroff scheme with the Baldwin-Lomax turbulence model. However, the global mass flow error for the mesh densities A to B appears to be to high ($\epsilon_m^G \approx 2\%$).

Local mass flow error

The global mass flow error does not give any indication on whether or not the effective coolant mass flow $\tilde{m}_{eff}^C$ on the plane of injection corresponds exactly to the one that should be injected, that is $\tilde{m}_{actual}^C$. Indeed, the coolant mass flow may be one or more order(s) of magnitude less than the total mass flow in the

---

4The small ellipse axes of the toroidal jet body have been calibrated such as it is ensured to inject $\tilde{m}_{actual}^C$, see Chapter 5. However, due to the discrete representation of the flow field on the plane of injection in the computational mesh, the effective mass flow $\tilde{m}_{eff}^C$ injected may differ from the one that should be injected
computational domain. In order to quantify the error in the coolant mass flow injection, a local mass flow error \( \epsilon_m^L \) is introduced.

\[
\epsilon_m^L = 100 \cdot \left[ \frac{\dot{m}_{\text{eff}}^c - \dot{m}_{\text{actual}}^c}{\dot{m}_{\text{actual}}^c} \right]
\] (6.4)

The local mass flow error found in the converged solution is plotted in Fig. 6.7 of all the mesh densities. Surprisingly, a clear distinction is observed between mesh densities A-B and C-E. For the coarser axial density grids (A-B), it is noticed that the local mass flow error is quite large \( (1.5\% < \epsilon_m^{\text{LOC}} < 4.1\%) \), whereas for the finer axial density grids, the local mass flow error is kept small \( (|\epsilon_m^{\text{LOC}}| \leq 1\%) \) whatever the lateral mesh density is. This indicates that the axial mesh density near the plane of injection has to be sufficiently fine, namely \( N_X > 3 \), to ensure an accurate coolant mass flow injection. Furthermore, it is observed that the local mass flow error is very similar for a lateral mesh density of \( N_Y = 9 \) or higher, for a fixed axial mesh spacing. Thus, in terms of local mass flow error, it is advised to have a mesh with a density of \( N_X > 3, N_Y > 5 \) (C2 case or higher) to ensure a grid independent solution. In summary, it has been shown here that globally and even locally the mass flow conservation is ensured when using the film-cooling jet model in a CFD code.
6.3. GRID-INDEPENDENT SOLUTION

6.3.2 Vortex circulation

In order to quantify when a grid-independent solution is obtained, it is first proposed to observe the axial evolution, downstream of the plane of injection, of the non-dimensionalized circulation $\Gamma'_{XZ}$ of one of the two vortices pertaining to the CVP structure. The circulation of the two vortices, quantifying their strength, is actually a major feature that sets the entrainment rate of the freestream flow underneath the jet. It is very important to ensure that it is well captured as it has direct consequences on the prediction of wall heat transfer and/or film-cooling effectiveness. The non-dimensionalized circulation $\Gamma'_{XZ}$ is defined as

$$\Gamma'_{XZ} = \Gamma'_{XZ}(X) = \frac{\int_{-2d}^{0} \int_{0}^{5d} \sqrt{\omega_X^2 + \omega_Z^2} \, dZ \, dY}{u_f / d}$$ (6.5)

Note here that the boundary layer vorticity is preemptively set to zero. The lateral component of vorticity $\omega_Y$ is not taken into account since it is marginal: the coolant fluid is strictly injected in the streamwise direction so that the CVP structure is mainly bending in the X-Z plane. Thus, the "circulation" $\Gamma'_{XZ}$ merely scales the total amount of vorticity flux (or strength) of one of the vortices of the CVP structure. The axial evolution of $\Gamma'_{XZ}$ is plotted in Fig. 6.8 as a function of different axial mesh densities (left, Fig. 6.8-I) and as a function of different lateral mesh densities (right, Fig. 6.8-II). At this point, it is worth mentioning that similar results are found for the other mesh density cases not shown here. Before proceeding to the analysis of the impact of the mesh density upon the solution, it is interesting to note that the circulation is monotonically decreasing following the axial distance. Furthermore, the slope $S_T = \Delta \Gamma'_{XZ} / \Delta (X/d)$ of the $\Gamma'_{XZ}$ curve is higher in the nearby downstream region of the hole, so-called region-I ($S_T \approx -0.05$ for $X/d \leq 10$) than in the far downstream region, so-called region-II ($S_T \approx -0.005$ for $X/d \geq 10$).

Impact of the axial mesh density

Now, in observing the impact of the axial mesh density in Fig. 6.8-I (left), it can be noticed that, with meshes A2 and B2, the circulation $\Gamma'_{XZ}$ is predicted 25% and 13% higher on average compared to the solutions obtained with meshes C2 to E2. Furthermore, it is clearly seen that the discrepancy in predicting $\Gamma'_{XZ}$ for A2

---

5It is actually only non-negligible in the boundary layer, which has not to be taken into account here. Indeed, the goal is to study the evolution of the vorticity coming from the coolant flow only.
6.3. GRID-INDEPENDENT SOLUTION

and B2 is already present just downstream of the plane of injection, see levels of \( \Gamma'_{XZ} \) at \( X/d = 2 \). This means that the quality of the solution which is ultimately obtained downstream of the plane of injection is very sensible to the axial mesh density near the hole. Thus, the axial mesh density near the hole should be controlled to ensure an accurate computation. In other words, the correct imposition of the immersed boundary condition on the plane of injection necessitates (but it is not sufficient) an axial mesh density of about \( N_X = 5 \). This observation can help to shed light on the relatively high local mass flow error found for meshes A and B. In addition, the circulation \( \Gamma'_{XZ} \) found in the region-II is quasi-identical for meshes C2 to E2 whereas in region-I, it exits a relatively small discrepancy of 3% for the solution found with the mesh case C2 compared to the identical solution obtained with the meshes D2 and E2. Overall, ensuring a grid-independent solution in region-II necessitates to have an axial mesh density of \( N_X = 5 \) or higher. In region-I an axial mesh density of \( N_X = 9 \) or higher is advised, although with an axial mesh density of \( N_X = 5 \) the solution is fairly grid-independent.

Impact of the lateral mesh density

Focusing on Fig. 6.8-II (right), one can observe the impact of the lateral mesh density upon the solution, for a constant axial mesh density (\( N_X = 5 \)). It is first noticed that a mesh having a low lateral spacing of \( N_Y = 5 \) (C1 case) gives an axial distribution of the circulation \( \Gamma'_{XZ} \) that is too low by 30% on average com-
pared to the other distributions found, for higher lateral mesh density. Furthermore, the result found in the C2 case ($N_Y = 9$) is found the same as for the finest lateral mesh resolution (C3 and C4) in region-II (far downstream). On the contrary, in the region-I, near the injection site, the circulation $\Gamma'_{XZ}$ found in the medium lateral mesh density case solution C2 is about 10% lower on average compared the finer mesh density cases C3 and C4. The two most axially refined mesh, C3 and C4 leads virtually to the same solution. As a conclusion, it is advised that the lateral mesh density should be of an order $N_Y = 9$ or higher to get a grid-independent solution in the far downstream (region-II). However, close to the injection site (region-I), a lateral mesh density of $N_Y = 13$ or higher is advised even though a lateral mesh density of $N_Y = 9$ can lead to a fairly grid-independent solution.

6.3.3 Adiabatic film-cooling effectiveness

Ultimately, the three-dimensional film-cooling jet model is designed to obtain accurate prediction of the wall adiabatic film-cooling effectiveness and/or heat transfer coefficient. In this section, the adiabatic film-cooling effectiveness obtained with the different mesh densities tested is analyzed. This is to check to which mesh density level it is ensured to get a grid-independent solution for this value. For this purpose, the laterally averaged adiabatic film-cooling effectiveness $\bar{\eta}$ is defined.

$$\bar{\eta} = \bar{\eta}(X) = \frac{\int_{-2d}^{2d} \eta(X,Y) \, dY}{\int_{-2d}^{2d} \, dY} \quad (6.6)$$

where $\eta(X,Y)$ is the local film-cooling effectiveness on the flat plate, at the location $(X,Y)$. The axial evolution of $\bar{\eta}$ is plotted in Fig 6.9 for a fixed lateral mesh density ($N_Y = 9$) and different axial mesh spacing (left - Fig. 6.9-I) and for a fixed axial mesh density ($N_X = 5$) and different lateral mesh spacing (right - Fig. 6.9-II). As previously mentioned for the results concerning the circulation $\Gamma'_{XZ}$, similar trends are found here for mesh densities that are not displayed in Fig. 6.9. Two regions where the axial evolution of $\bar{\eta}$ is different are distinguished. There is a region near the hole (region-I, where $X/d \leq 10$) and a region far downstream of the hole (region-II, where $X/d \geq 10$). This distinction has already been introduced in the previous section, but it is now clearly seen here. Indeed, the laterally averaged film-cooling effectiveness $\bar{\eta}$ very quickly drops in the region-I and exhibits a hollow shape. It is recalled that the momentum flux ratio studied here is
6.3. GRID-INDEPENDENT SOLUTION

Figure 6.9: Axial evolution of laterally averaged film-cooling effectiveness $\bar{\eta}$ for different axial mesh densities, based on a lateral mesh density of $N_Y = 9$ (left - I) and for different lateral mesh densities, based on a axial mesh density of $N_X = 5$ (right - II).

relatively high ($IR = 3.2$) so that a detachment of the coolant jet may be expected. In the region-II, the axial evolution of $\bar{\eta}$ stabilizes around $\bar{\eta} = 0.016$.

Impact of the axial mesh density

In the first place, the impact of the axial mesh density is observed in Fig. 6.9-I (left). The solution $\bar{\eta}$ obtained with the coarsest axial mesh density (A2 case) is remote from the other solutions, by more than 100%, in the region-I. Furthermore, in this region, the solutions found with the finest mesh density cases D2 and E2 are quasi identical, confirming that a totally grid-independent solution can be found in the near hole downstream region with a axial mesh spacing of $N_X = 9$. However, the solution for $\bar{\eta}$ obtained with a medium axial mesh density (C2) is not too far from solutions found in the finest axial mesh resolution ($\bar{\eta}$ is 30% too high on average) whereas the discrepancy on $\bar{\eta}$ starts to be quite high for the B2 case ($\bar{\eta}$ is 55% too high on average). On the contrary, in the region-II, the grid-independent solution in predicting $\bar{\eta}$ appears to be less dependent of the axial mesh spacing. Indeed, in comparison to the solutions obtained with the D2 and E2 mesh density cases, $\bar{\eta}$ is predicted 18% too low for the A2 case and only 6% too low for the C2 case.
6.4. Toward an Engineering Solution

Impact of the lateral mesh density

The impact of the lateral mesh density upon the prediction of \( \eta \) is observed in Fig. 6.9-II (right). First of all, it is noticed that the solution obtained with the coarsest lateral mesh density, that is \( N_Y = 5 \) (C1 case) differs fundamentally from the other solutions. Indeed, there is no differentiation between region-I and region-II since the axial evolution of \( \eta \) is monotonically decreasing: the hollow shape of the \( \eta \) curve in the region-I is not found. The finest lateral mesh density cases C3 and C4 give a very similar solution for \( \eta \): the difference in the prediction of \( \eta \) is less than 10% on average. The medium lateral mesh density case C2 allows to get a prediction of \( \eta \) that is not too far (less than 13% on average) from the finest lateral mesh refinement cases C3 and C4. Furthermore, it is underlined that the impact of the lateral mesh density is identical in region-I and region-II since the differences found on the prediction of \( \eta \) for the different lateral mesh density tested are the same in both regions. As a conclusion, a complete grid-independent solution is attained with a lateral mesh resolution higher than \( N_Y = 13 \) but is near to be with a lateral mesh resolution of \( N_Y = 9 \) (less than 13%\(^6\) different all along the flat plate).

6.4 Toward an engineering solution

An engineering solution for the utilization of the model for design purpose, as well as for investigation of different engineering flow configurations, is proposed. First, the possible gain in computational accuracy, compared to a totally grid-independent solution, is systematically analyzed as a function of the mesh density. This is done for one row of holes. Then, an evaluation of the mesh needed to simulate the flow through a film-cooled turbine blade is given.

6.4.1 Accuracy versus computational time

The need for quantifying accuracy versus computational time

It is intended in this section to quantify the extra computational time needed to gain a given percentage of accuracy. It is assumed here that the accuracy is the best for the finest mesh case, namely E4. Actually, even if the solution found using the computational mesh E4 is not totally accurate compared to available experimental data, it is the best solution, intrinsically to CFD, that can be obtained. First of

\(^6\)This difference is based on a very low value of \( \eta \).
all, as it has been demonstrated in the previous sections, it is needed to have a mesh density of at least $N_X = 9$, $N_Y = 13$ (D3 case) to obtain a quasi grid-independent numerical prediction. In addition, as it has been plotted in Fig. 6.4, the total computational time to find a converged solution in such a mesh is about 16 hours. Now, if a mesh of type C2 is used ($N_X = 5$, $N_Y = 9$), a converged solution is obtained in approximately 4 hours, which is 4 times less of computational time compared to D3. Furthermore, the results in terms of vortex circulation and adiabatic film-cooling effectiveness are found quite similar for both computations\(^7\). From a purely scientific point of view, it is evident that a computation using a mesh density similar to D3, or even E4 is desired, since the total computational time is not the main issue. However, from an engineering point of view, if the purpose of the three-dimensional film-cooling jet model is to be used in a design iterative process or for the investigation of different flow configurations, it is then obvious that the total computational time becomes a very important issue. In fact, the principal (but not unique) goal of the development of the film-cooling model is to deliver the designer a numerical tool to accurately predict the aerodynamics and heat transfer through film-cooled turbine blade passages in a reasonable time scale. Therefore, coming back to the single row of film-cooling holes problem on a flat plate, it would be relevant to quantify the gain in computational accuracy obtained if it is spent one more computational time unit. Note here that spending more computational time means increasing the grid size. This functional relation has been shown in Fig. 6.4.

**Quantification: the marginal gain coefficient**

In order to quantify the gain of computational accuracy versus extra computational time, the marginal gain coefficient $C_Q$ is introduced.

$$C_Q = \frac{\Delta Q^*}{\Delta t^*}$$  \hspace{1cm} (6.7)

where $\Delta Q^*$ is given by

$$\Delta Q^* = 2 \left[ \frac{Q_2 - Q_1}{Q_2 + Q_1} \right]$$ \hspace{1cm} (6.8)

$Q$ represent a physical value, such as the circulation or the adiabatic film-cooling effectiveness. $\Delta Q^*$ is the rate of the change of this value, obtained with two

\(^7\)Discrepancy in predicting $\Gamma'_{XZ}$ is less than 3% and for $\bar{\eta}$ is less than 10%, averaged over the whole flat plate length, for a low value of $\bar{\eta}$
different mesh densities 1 and 2. The two different mesh density cases 1 and 2 can be expressed in terms of total computational time $t_1$ and $t_2$, see Fig. 6.4. The added computational time $\Delta t^*$ can be defined as being as follows

$$\Delta t^* = \frac{t_2 - t_1}{t_{\text{unit}}}$$

(6.9)

where $t_{\text{unit}}$ is the reference time unit. This is the time that the user of the model refers as being the critical time when willing to spend more computational time for better accuracy. It is assumed here that $t_{\text{unit}}$ is equal to 1 hour. The marginal gain coefficient $C_Q$ can practically be described as follows

- Given a solution found in a time $t_1$ (or mesh 1), the marginal gain coefficient $C_Q$ quantifies the gain in accuracy in predicting $Q$ if one is willing to spend one more computational time unit.

In the following, the marginal gain coefficient, based on vortex circulation and adiabatic film-cooling effectiveness, is analyzed.

**Marginal gain coefficient for circulation**

The circulation $\bar{\Gamma}_{XZ}$ taken for the marginal gain coefficient $C_T$ is the axially averaged value of $\Gamma'_{XZ}$

$$\bar{\Gamma}_{XZ} = \frac{\int_{X/d=2}^{22} \Gamma'_{XZ} (X) \, dX}{\int_{X/d=2}^{22} dX}$$

(6.10)

The marginal gain coefficient $C_T$, based on circulation, is plotted in Fig. 6.10 as a function of the axial mesh density (so that computational time) for a constant lateral mesh density (left - Fig. 6.10-I) and as a function of the lateral mesh density (so that computational time) for a constant axial mesh density (right - Fig. 6.10-II). First of all, as it can be noticed in the two plots of Fig. 6.10, the marginal gain $C_T$ is explicitly computed at axial and lateral mesh density levels located in between the ones that have been used for the CFD computations. Indeed, the marginal gain coefficient $C_Q$ is a function of the first order derivatives of the quantity observed, that is the circulation in this case. Globally, in observing the two plots of Fig. 6.10, it is found that the marginal gain is very high (more than 50% at least) for meshes having the coarsest density, that is $0 < N_X < 2$ for any fixed lateral mesh density (Fig. 6.10-I) and $0 < N_Y < 7$ for any fixed axial mesh density (Fig.
Figure 6.10: Marginal gain coefficient $C_T$ as a function of the axial mesh density for a constant lateral mesh density (left - I) and as a function of the lateral mesh density for a constant axial mesh density (right - II).

6.10-II). Now, focusing on Fig. 6.10-I, one can see that the marginal gain $C_T$ very quickly drops when the axial mesh density is increased from $N_X = 2$ to $N_X = 7$. For example, for a constant lateral mesh density $N_Y = 9$ (C case) and a current solution with an axial mesh density of $N_X = 2$, if one is willing to spend one more hour for the computation, the gain in accuracy for predicting the circulation is about 6.1%. Furthermore, for a current solution with an axial mesh density of $N_X = 7$, if it is willing to spend one hour more for the computation, the gain in accuracy for predicting the circulation is only of about 0.13%. In fact, for any fixed lateral mesh density equal to $N_Y = 9$ or higher, the marginal gain of accuracy in predicting the circulation is ensured to be lower than 0.13%, when having an axial mesh density of $N_X = 7$. This shows that from an engineering point of view, it is best to use a mesh having an axial mesh density around $(4 < N_X < 7)$ (for a lateral mesh density of $N_Y = 9$ or higher) for predicting the aerodynamic of a coolant jet using the model. Now, looking at Fig. 6.10-II, the marginal gain coefficient for any fixed axial mesh density is analyzed. It is remarkable to observe that the marginal gain coefficient is still high $(2.7% < C_T < 4.9%)$ for a lateral mesh density of $N_Y = 7$ and a fixed axial mesh spacing of $N_X = 5$ or higher. However, the marginal gain coefficient rapidly decreases down to small values $(0.31% < C_T < 0.93%)$ for a lateral mesh density of $N_Y = 11$ and a fixed axial mesh spacing of $N_X = 5$ or higher. Therefore, it is quantified that in the zone of a mesh density of $(4 < N_X < 7)$ and $(7 < N_Y < 11)$, the marginal gain of accuracy in predicting aerodynamic flow features drops down to values
(0.31% $< C_T < 5.5\%$) that are acceptable from an engineering point of view.

**Marginal gain coefficient for adiabatic film-cooling effectiveness**

The film-cooling effectiveness $\langle \eta \rangle$ taken for the marginal gain coefficient $C_\eta$ is the axially averaged value of $\bar{\eta}$.

$$\langle \eta \rangle = \frac{\int_{X/d=2}^{22} \bar{\eta}(X) \, dX}{\int_{X/d=2}^{22} dX}$$  \hspace{1cm} (6.11)

The marginal gain coefficient $C_\eta$ based on the spatially averaged adiabatic film-cooling effectiveness is plotted in Fig. 6.11 as a function of the axial mesh density (so that computational time) for a constant lateral mesh density (left - Fig. 6.11-I) and as a function of the lateral mesh density (so that computational time) for a constant axial mesh density (right - Fig. 6.11-II). The trend of the marginal gain of accuracy in the two plots of Fig. 6.11 is relatively similar to what has been found in the analysis of the marginal gain based on circulation, in Fig. 6.10. For example, it is quantified that in the zone of a mesh density of $(4 < N_X < 7)$ and $(7 < N_Y < 11)$, the marginal gain of accuracy in predicting heat transfer flow features drops down to values $(0.01\% < C_T < 3.2\%)$. As a conclusion, if the adiabatic film-cooling effectiveness does not represent the heat transfer at the flat plate surface, but it is related to it.
three-dimensional film-cooling jet model has to be used in a design iterative pro­
cess, where total computational time and accuracy are both equally important, it is
advised to use a mesh density of \((4 < N_X < 7)\) and \((7 < N_Y < 11)\).

6.4.2 Making use of the model for film-cooled turbine flow prob­
lems

The type of mesh density that is the best suited to have an optimal use of the film­
cooling model in term of design process has been quantified. This quantification is
strictly given for a row of film-cooling holes. Owing to the fact that the goal of the
film-cooling model is to be used for predicting flows through film-cooled turbine
passages in a reasonable time scale, it is carried out here an evaluation of what
should be the mesh size for such a problem. The evaluation is carried out upon the
grid-independency study made in the previous sections.

Postulate

It is first considered that a film-cooled turbine blade can only contains rows of
holes, \(i.e\). the cooling holes are radially aligned, and each radial arrangement of
holes is a row. In addition, it is assumed that the grid topology available to mesh
such a blade passage is organized such as it is a spanwise (radial) pile-up of two­
dimensional meshes lying on radial planes, see Chapter 10 for more information.
No local grid refinement is considered.

Mesh size

In general, the number \(N_{2D}\) of grid nodes pertaining to one plane is fixed. Actu­
ally, as it will be shown in Chapter 10, the number of grid nodes for a plane, in the
axial direction (inlet to outlet), does almost not vary as a function of the number
row of holes\(^9\). In the tangential direction (pressure to suction side), the number
of grid nodes is driven by the mesh resolution needed in the boundary layer. The
order of magnitude of \(N_{2D}\) is about \(N_{2D} \approx 5000\). Thus, it becomes obvious that
the main parameter that will affect the total mesh size is the maximum number of
film-cooling holes \(N_h\) in a row. Indeed, the number of piled-up planes is determ­
ined by the number of holes in the spanwise direction. Furthermore, for each hole,
it is desired to have enough mesh resolution, which means that the lateral mesh
density \(N_Y\) has to be specified. Near the hub platform and casing, the boundary

\(^9\)Given the fact that an axial mesh density of \(N_X \approx 4 - 7\) is respected
layer has to be accurately captured, so that it is needed to add $N_{bl}$ mesh planes at these two locations. It can be estimated that about 15 mesh planes should be put in the vicinity of both endwall. Gluing all these parameters together allows the derivation of an expression evaluating the total number $N_T$ of grid nodes needed.

$$N_T \sim [2N_{bl} + (2N_h - 1) N_Y] N_{2D} \quad (6.12)$$

As an example, the total number of grid nodes $N_T$ needed to simulate a film-cooled turbine blade with the aid of the model, having no more than $N_h = 30$ holes per rows (and several rows) and a lateral mesh density $N_Y = 9$ is about $N_T = 2.8 \cdot 10^6$.

### 6.5 Conclusion

The numerical strategy to immerse the film-cooling jet model appears to be very robust and is computationally efficient: the computational overhead, when using the model, is only $\sigma \simeq 1\%$ when simulating one infinite row of cooling holes. The local mass flow error on the plane of injection is of the order of $\varepsilon_{mf} \simeq 1\%$. A marginal gain coefficient $C_Q$ is introduced. It quantifies the gain in computational accuracy if it is intended to spend one more time unit for the computation (or increasing the mesh density). Using this marginal gain coefficient, the optimal use of the CFD-embedded film cooling model is determined: it is advised to have about $(4 < N_X < 7)$ and $(7 < N_Y < 11)$ grid nodes per hole diameter. The proposed marginal gain coefficient is a general concept. Thus, further use of this coefficient to evaluate accuracy versus computational time when using CFD is proposed. Based on the grid independency study, the use of the model in a film-cooled turbine, with about 30 holes per rows (and with several rows per blade side), should lead to a computational mesh with about $2.8 \cdot 10^6$ grid nodes.
Chapter 7

Steady Aerodynamic Validation

7.1 Introduction

7.1.1 Strategy for validation

A steady-state validation of numerical predictions obtained with the aid of the numerically-immersed film cooling jet model, versus experimental data, is performed in this Chapter. To this effect, the film-cooled flat plate of Bernsdorf [13] is used. All the steady-state injection sets Sl to SlI (see Chapter 5) are investigated. Thus, this investigation covers two streamwise injection angles ($\alpha_0 = [30^\circ, 50^\circ]$) combined with a large range of momentum flux ratios $IR$. For each injection case, the mesh C2 is utilized since it has been shown to be the best suited for engineering applications. Stability, convergence rate and mass flow conservation for all computed cases are found in agreement with the results presented in Chapter 6.

Comparison to experimental data

In a first step, it is intended to show the effect of the streamwise injection angle $\alpha_0$ and momentum flux ratio $IR$ upon the aerodynamics of a film-cooling jet in the vicinity of the hole ($X/d \leq 6.0$). In particular, the jet penetration, spreading and vortical system, in terms of velocity and vorticity field, are analyzed. This analysis is based upon the available experimental data in order to get a proper view of the physical flow field. Based upon this analysis, each numerical prediction is systematically compared to the experimental data of Bernsdorf [13] to validate the results. Agreements and deficiencies of the prediction with CFD using the film-cooling model are highlighted. For the sake of simplicity, only flow solutions with two different momentum flux ratios, which are $IR = [1.0, 4.0]$, combined with the two streamwise injection angles $\alpha_0$ aforementioned are presented. Flow results with other momentum flux ratios exhibit the same trend of flow evolution as the ones covered here.
7.1. INTRODUCTION

Comparison to other numerical method

In a second step, the numerical results found with the proposed film-cooling model (FCM) are compared to CFD predictions obtained using two other injection strategies. This is done for the injection set S4. In the first place, a wall injection (WI) computation is shown, where the coolant jet is simply injected by setting the velocity vectors and temperature at the wall, where the hole should stand. In the second place, a full injection (FI) computation is performed, where the plenum chamber and hole are gridded. Overall, this comparison shows the merit and benefit of using the proposed film-cooling model as a numerical tool for predicting film-cooling jet aerodynamics.

7.1.2 Overview of jet flow features

To give an insight into the main flow features studied in the following, the predicted three-dimensional flow field near the injection site, using the film cooling model, is plotted in Fig. 7.1. This is shown for a streamwise injection angle $\alpha_0 = 50^\circ$ and a momentum flux ratio of $IR = 4.0$. The freestream streamlines show the lateral and vertical deflection of the freestream fluid particles near the hole exit, demonstrating the right numerical treatment to immerse the jet surface. Just after the plane of injection, located at $X/d = 1.0$, it is clearly observed that the streamlines are joining back in the jet center revealing the three-dimensional entrainment of the freestream fluid due to the Counter-rotating Vortex Pair (CVP). Further downstream, the freestream fluid particles are convected inside the jet and slowly start to diverge. As it can be seen in three different cross planes ($X/d = 2.0, 7.0, 12.0$), the streamwise vorticity $\omega_x^*$, normalized by inlet freestream velocity $U_f$ and hole diameter $d$, represents the downstream evolution and decay of the vortices. On the flat plate, the normalized static pressure $\Delta P_s$ change relative to freestream static pressure $P_{s,f}$ is depicted. The increase of static pressure just upstream of the hole exit is found by the numerical prediction, which confirms the blockage effect created by the locally embedded jet toroidal surface. Also, in a close inspection near the injection plane, one can see the low level of relative static pressure due to the flow deviation and the low momentum wake zone.
7.2. COMPARISON TO EXPERIMENTAL DATA: VELOCITY FIELD

7.2.1 Flow regions of the coolant jet

Jet penetration and windward side

Contours of predicted streamwise velocity $U/U^*$, normalized by the freestream velocity $U^*$, in the center plane $Y/d = 0.0$, superimposed with freestream streamlines ribbons are represented in yellow. 

Figure 7.1: Numerical prediction, using the film-cooling model, of the flow field near the hole exit. On the flat plate ($Z/d = 0.0$), contours of normalized static pressure $\Delta P_s$ are represented. Normalized streamwise vorticity $\omega_x^*$ is shown at three different cross sections ($X/d = 2.0, 7.0, 12.0$). The freestream streamlines ribbons are represented in yellow.
7.2. COMPARISON TO EXPERIMENTAL DATA: VELOCITY FIELD

lines, are first shown and compared to experimental measurements in order to verify the right implementation of the immersed three-dimensional film-cooling model. This is shown in Fig. 7.2 for a streamwise injection angle of $\alpha_0 = 30^\circ$ and in Fig. 7.3 for a streamwise injection angle of $\alpha_0 = 50^\circ$. The streamlines exhibit the freestream flow deviation and also the rapid turning of the jet. As already noticed by Bernsdorf et al. [12], the streamwise injection angle $\alpha_0$ seems to play a minor role in the jet trajectory. In contrast, the momentum flux ratio $IR$ is much more important in affecting the coolant jet path. It typically penetrates 50% more vertically in the $IR = 4.0$ case, compared to the $IR = 1.0$ case. This feature is well captured by the CFD prediction using the CFD-embedded film cooling jet model. The windward side (velocity overshoot) is clearly seen in the high momentum ratio case ($IR = 4.0$). The computation predicts this phenomenon with high accuracy, but it is to be expected as the model has been carefully tuned to the experiment. Interestingly, the maximum streamwise velocity just after the plane of injection is higher in the $\alpha_0 = 30^\circ$ case ($U_{max}/U_f = 1.47$) than in the $\alpha_0 = 50^\circ$ case ($U_{max}/U_f = 1.36$); this might be because the jet has originally more streamwise momentum when emerging from the hole exit.

Wake region

As the coolant flows downstream, it is moving away from the wall, forming a wake region of low streamwise velocity magnitude. In a close inspection of the streamwise velocity contours in this region, the velocity deficit is overpredicted, in the $\alpha_0 = 30^\circ$ case, by 17% on average for $IR = 1.0$ and by 15% on average for $IR = 4.0$. In the $\alpha_0 = 50^\circ$ case, this zone is better captured in the computation, although there is a small overprediction of 8% on average for the velocity deficit. Furthermore, it is worth mentioning that the shape of the wake region differs as a function of the streamwise injection angle $\alpha_0$. In fact, for a high streamwise injection angle ($\alpha_0 = 50^\circ$), see Fig 7.3, the streamwise velocity contours in the wake region exhibit a lobe-type of shape, where the minimum of velocity is not unique, so that not only located at the lowest measured vertical position but also just under the windward side. On the contrary, it is observed that in the low streamwise injection case ($\alpha_0 = 30^\circ$), see Fig. 7.2, the velocity minima is located at the lowest measured vertical position only. This feature is found in the CFD prediction using the film cooling model.
7.2. COMPARISON TO EXPERIMENTAL DATA: VELOCITY FIELD

Figure 7.2: Contours of normalized streamwise velocity $U/U_f$, superimposed with freestream streamlines, in the center plane $Y/d = 0.0$, for a streamwise injection angle $\alpha_0 = 30^\circ$. 
Figure 7.3: Contours of normalized streamwise velocity $U/U_f$, superimposed with freestream streamlines, in the center plane $Y/d = 0.0$, for a streamwise injection angle of $\alpha_0 = 50^\circ$. 


7.2. COMPARISON TO EXPERIMENTAL DATA: VELOCITY FIELD

7.2.2 Influence of the streamwise injection angle $\alpha_0$ and momentum flux ratio $IR$

In order to quantify the influence of the momentum flux ratio $IR$ and streamwise injection angle $\alpha_0$ upon the jet penetration and wake, in Fig. 7.4 profiles of streamwise velocity $U/U_f$ and in Fig. 7.5 profiles of vertical velocity $W/U_f$ are presented. In each figure, flow profiles obtained in the measurements as well as those predicted using CFD with the immersed film-cooling model are given. The velocity profiles, given in the center plane $Y/d = 0.0$, are always displayed in four different axial locations, namely at $X/d = 2.0, 4.0, 6.0$ (near hole downstream - region-I) and at $X/d = 14.0$ (far hole downstream - region-II). All the following velocity field analysis is carried out only for region-I, as no experimental data is available for region-II.

**First order influence - $IR$**

First of all, focusing on the streamwise velocity profiles $U/U_f$ plotted in Fig. 7.4, it is clearly confirmed that the level of the momentum flux ratio $IR$ has the greatest influence upon the shape of the velocity profile as well as upon the jet aerodynamic. The presence of a velocity overshoot in the windward side is found in the high momentum flux ratio case ($IR = 4.0$) whereas there is no evidence of it in the low momentum flux ratio case ($IR = 1.0$). Moreover, the velocity deficit in the wake region is more pronounced, by 40% on average, for $IR = 1.0$ than for $IR = 4.0$. From a global perspective, the streamwise injection angle $\alpha_0$ seems to influence only moderately the penetration of the coolant jet. For both streamwise injection angles, same flow patterns are found for the different momentum flux ratios $IR$ investigated. However, in details, for a high momentum flux ratio ($IR = 4.0$), the velocity peak $U_{max}/U_f$ is located about 15% higher vertically for the $\alpha_0 = 50^\circ$ case compared to the $\alpha_0 = 30^\circ$ case. Furthermore, the magnitude of this velocity peak is about 12% higher in the lowest streamwise injection angle case ($\alpha_0 = 30^\circ$) compared the highest streamwise injection angle case ($\alpha_0 = 50^\circ$).

**Second order influence - $\alpha_0$**

In a close inspection of Fig. 7.4, it is confirmed that the wake region does not get the same flow pattern between a low and high streamwise injection angle $\alpha_0$, for a fixed momentum flux ratio $IR$. Whatever the momentum flux ratio, the stream-
Figure 7.4: Profile of streamwise velocity \( \left( U/U_f \right) \) at different axial locations \((X/d = 2.0, 4.0, 6.0, 14.0)\), in the center plane \(Y/d = 0.0\), for a streamwise injection angle of \( \alpha_0 = 30^\circ \) (top) and \( \alpha_0 = 50^\circ \) (bottom).

The streamwise velocity profile obtained with a high streamwise injection angle \((\alpha_0 = 50^\circ)\) exhibits a local minima\(^1\) that is hardly found in the low streamwise injection angle case \((\alpha_0 = 30^\circ)\). Furthermore, this low momentum pocket vertically travels as the jet advances toward the axial direction\(^2\). In order to shed light on this phenomenon, one can observe the profile of the vertical component \(W/U_f\) of the velocity vector in Fig. 7.5. It is first observed, just after hole exit \((X/d = 2.0)\), that the vertical velocity magnitude is about 45% smaller for a low momentum flux ratio \((IR = 1.0)\)

\(^1\) e.g. at \(X/d = 2.0\), it is located at \(Z/d = 0.4\) in the \(IR = 1.0\) case and at \(Z/d = 0.6\) in the \(IR = 4.0\) case

\(^2\) e.g. at \(X/d = 6.0\), it is located at \(Z/d = 0.6\) in the \(IR = 1.0\) case and at \(Z/d = 1.4\) in the \(IR = 4.0\) case
Figure 7.5: Profile of vertical velocity \((W/U_J)\) at different axial locations \((X/d = 2.0, 4.0, 6.0, 14.0)\), in the center plane \(Y/d = 0.0\), for a streamwise injection angle of \(\alpha_0 = 30^\circ\) (top) and \(\alpha_0 = 50^\circ\) (bottom).

compared to a high momentum flux ratio \((IR = 4.0)\). This can explain the much lower jet vertical penetration observed previously for a low momentum flux ratio flow case. However, a very stringent fact is that the vertical velocity component \(W/U_J\) dies very quickly in the low streamwise injection angle case \((\alpha_0 = 30^\circ)\) whereas this phenomenon is not observed with the high streamwise injection case \((\alpha_0 = 50^\circ)\). Indeed, at the axial position \(X/d = 6.0\), the maximum vertical velocity value is \(W/U_J = 0.17\) for \(\alpha_0 = 30^\circ\) whereas it is still \(W/U_J = 0.31\) for \(\alpha_0 = 50^\circ\). Thus, the rapid dying of the vertical component \(W/U_J\) of the velocity vector, in the low streamwise injection angle case \((\alpha_0 = 30^\circ)\) might explain the lack of transport of low momentum fluid upward, in contrast to the high streamwise injection case \(\alpha_0 = 50^\circ\).
7.2. COMPARISON TO EXPERIMENTAL DATA: VELOCITY FIELD

Conclusion

The velocity field has been observed to be primarily affected by the momentum flux ratio $IR$. The coolant jet typically penetrates 50% more vertically in the high momentum flux ratio case ($IR = 4.0$) than in the low momentum flux case ($IR = 1.0$). The streamwise injection angle $\alpha_0$ has less influence but leads locally to different flow velocity profiles, especially in the wake region. For a low streamwise injection angle $\alpha_0 = 30^\circ$, the upward fluid motion decays more quickly than for the high streamwise injection case $\alpha_0 = 30^\circ$. The velocity field in region-I exhibits large gradients (windward side and wake region) and changes rapidly, in contrast to region-II.

7.2.3 Agreements and discrepancies of the CFD prediction

Agreements - windward side and low $IR$

From a global point of view, looking at Figs. 7.4 and 7.5, it is found that the main flow regions, that are the windward side and wake region, are predicted in agreement with the experiment. It is observed in Fig. 7.4 that the windward side of the jet, for a high momentum flux ratio ($IR = 4.0$) is well captured by the CFD prediction, all along region-I. The accuracy in the prediction of the vertical position of the streamwise velocity maxima $U_{max}/U^j$ is almost perfect, at least relatively to the vertical density of measurement points (each $\Delta Z/d = 0.2$ increments). The discrepancy in predicting the maximum streamwise velocity magnitude $U_{max}/U^j$ is no more than 7% within region-I for both streamwise injection angles, which demonstrates a close agreement to experimental data. In general, numerical results for a low momentum flux ratio ($IR = 1.0$) are closer to experimental data than for a high momentum flux ratio ($IR = 4.0$). For instance, looking at the numerical results obtained in the $IR = 1.0$ case, the streamwise velocity is moderately underpredicted, in the wake region only, by 6% on average for the $\alpha_0 = 30^\circ$ case. In the $\alpha_0 = 50^\circ$ case, the predicted streamwise velocity is remarkably matching the experimental data. For high momentum flux ratio cases ($IR = 4.0$), although the CFD prediction of the windward side of the jet is in close agreement with the measurements, as explained previously, the prediction of the streamwise velocity profile in the wake region suffer systematically from an underestimation.
7.2. COMPARISON TO EXPERIMENTAL DATA: VELOCITY FIELD

Discrepancies - wake and high $IR$

Indeed, in the low streamwise injection angle case ($\alpha_0 = 30^\circ$), the streamwise velocity $U/U_f$ is moderately (9%) overpredicted close to the injection site ($X/d = 2.0$), but further downstream, it is underpredicted by about 14%. To better understand this sign change of the misprediction, in Fig. 7.5 the predicted vertical velocity profile $W/U_f$ is observed. It is noticed that, although the vertical velocity magnitude is predicted about 20% too low near the injection site ($X/d = 2.0$), it is highly overpredicted further downstream, by more than 100%. In fact, it seems that the CFD does not manage to predict the very abrupt decreases of the magnitude of the vertical velocity $W/U_f$, for a streamwise injection angle of $\alpha_0 = 30^\circ$. In return, this leads the CFD overpredicting the upward lift-off of the low momentum fluid located near the wall surface, which consequently leads to an underprediction of the streamwise velocity in the wake region. Concerning the $\alpha_0 = 50^\circ$, $IR = 4.0$ case, the streamwise velocity is also underpredicted in the wake region but the arising process of this discrepancy appears to be different. Indeed, already at $X/d = 2.0$, an 12% underprediction of the streamwise velocity in the wake region is found. Moreover, the local streamwise minima is not found. Further downstream, this discrepancy stays at the same level but the local minima can now be observed, at the same vertical position than in the experiment. To explain this discrepancy, the vertical velocity profile $W/U_f$ is observed in Fig. 7.5. One can noticed the remarkably close agreement between the predicted and measured vertical velocity profile $W/U_f$. Thus, in the $\alpha_0 = 50^\circ$ case, the strength of this upward lift-off motion is predicted in agreement with the experiment. This means that the discrepancy in predicting the streamwise velocity, found near the injection site, is transported toward the axial direction.

Far downstream - region-II

The predicted profile of streamwise and vertical velocity in the region-II is plotted in Fig. 7.4. At this location, the upward motion of the flow is negligible in the low momentum flux ratio case ($IR = 1.0$), whatever the streamwise injection angle is. On the contrary, a remaining upward motion of the flow is observed for a high momentum flux ratio case. The boundary layer is established and stable. Furthermore, it is observed that the boundary layer velocity deficit is about 10% higher in the low momentum flux ratio case ($IR = 1.0$), compared to the higher one ($IR = 4.0$).

---

\[a\] very similar solution is found further downstream
Figure 7.6: Contours of normalized streamwise velocity ($U/U_f$), superimposed with cross velocity vectors ($V/U_f, W/U_f$), in the cross section $X/d = 4.0$, for a streamwise injection angle of $\alpha_0 = 30^\circ$. 

\[ \alpha_0 = 30^\circ \quad DR = 1.0 \quad BR = 1.0 \quad IR = 1.0 \quad X/d = 4.0 \]

\[ \alpha_0 = 30^\circ \quad DR = 1.0 \quad BR = 2.0 \quad IR = 4.0 \quad X/d = 4.0 \]
7.2. COMPARISON TO EXPERIMENTAL DATA: VELOCITY FIELD

Figure 7.7: Contours of normalized streamwise velocity \( \frac{U}{U_f} \), superimposed with cross velocity vectors \( \frac{V}{U_f}, \frac{W}{U_f} \), in the center section \( X/d = 4.0 \), for a streamwise injection angle of \( \alpha_0 = 50^\circ \).
7.2. COMPARISON TO EXPERIMENTAL DATA: VELOCITY FIELD

Jet spreading - cross velocity field

To better understand the formation of the wake zone and the corresponding upward fluid motion, contours of normalized streamwise velocity are plotted in Figs. 7.6 and 7.7 in the cross section $X/d = 4.0$. The secondary flow vectors are superimposed to the contour plots. In Fig. 7.6, experimental measurements are compared to CFD prediction for a streamwise injection angle of $\alpha_0 = 30^\circ$. Same plots are given in Fig. 7.7 for a streamwise injection angle of $\alpha_0 = 50^\circ$. For all cases, although the spreading of the wake zone and windward side are well predicted, it is confirmed that the velocity deficit in the wake zone is systematically too pronounced. In observing the secondary flow vectors, exhibiting the counter-rotating vortex pair structure, the position of the center of vortices are found insensitive to the streamwise injection angle but sensitive to the momentum flux ratio, in which the CFD gives a good answer. However, numerical results end up constantly overpredicting the lateral spreading of the center of vortices by 15% to 30%. Furthermore, it is confirmed that with a high streamwise injection angle ($\alpha_0 = 50^\circ$), the wake has the tendency to get a local minima of streamwise velocity. These plots illustrate the action of the CVP structure to lift-off low-momentum fluid upward.

Conclusion

The use of the numerically-immersed film cooling jet model to predict the velocity field of the coolant jet has been validated for a wide range of momentum flux ratios ($0.6 < IR < 7.2$) and two streamwise injection angles $\alpha_0 = [30^\circ, 50^\circ]$. It is found that jet flows with a low momentum flux ratio are better predicted than with a high momentum flux ratio. Furthermore, the wake region of the jet is where the highest discrepancy arises, even more pronounced for the low streamwise injection angle case ($\alpha_0 = 30^\circ$). This discrepancy in the wake zone is shown to be linked to the overprediction of the upward motion of the low momentum fluid, induced by the CVP structure.

$^4$It is centered around $Z/d = 0.55$ for $IR = 1.0$ and around $Z/d = 0.90$ for $IR = 4.0$
7.3 Comparison to experimental data: vorticity field

7.3.1 Main issues of the comparison

Motivation

It is intended to validate the use of the CFD-embedded film cooling jet model for the prediction of the vortical structures that pertain to a film cooling jet. The right prediction of the vorticity field shall give more confidence in using the film cooling model to predict the aerodynamic of a coolant jet. In a way, it is intended to perform an *a posteriori* validation of the image vortex model describing the CVP structure as well as the immersion of the jet toroidal surface. In addition, although the position of the vortices has been found to be about the same in both streamwise injection angle cases for a fixed momentum flux ratio, it has been observed that the CVP structure seems to play a much stronger role in the $\alpha_0 = 50^\circ$ case, since the upward motion of fluid remains longer. Thus the evolution of the strength of the vortices should influence the streamwise evolution of the wake zone, which has been shown to be the most difficult flow region to predict. In order to focus on these issues, the normalized streamwise $\omega_X$ and vertical $\omega_Z$ vorticity components are plotted in Figs. 7.8 and 7.9, in an horizontal plane $Z/d = 0.4$ above the flat plate surface (inside the wake region). As the the vertical penetration of the jet has been shown to be very similar for both streamwise injection angle, for a fixed momentum flux ratio, the same horizontal plane can be observed whatever the streamwise injection angle is. For clarity reason, only results with a high momentum flux ratio ($IR = 4.0$) are displayed; the results with a low momentum flux ratio ($IR = 1.0$) are similar but with a lower vorticity magnitude.

Vortical structures

Overall, a good agreement between measured and predicted vorticity values is observed, which verifies the model. The two branches (of opposite sign) of the counter-rotating vortex pair are seen in the streamwise vorticity plot (structure A). The four central legs in the vertical vorticity plot exhibit the downwash and upwash shearing sides of the two vortices (structure B and C). Furthermore, on the lateral sides of the counter-rotating vortex pair, a trace of vorticity of opposite sign can be seen in the experiment and also reproduced in the computation (structure D and E). This might indicate a presence of a horseshoe vortex but no formal conclusion can be drawn here, although it can be noticed that the axial and vertical strength of this structure is more than 100% smaller than the CVP structure.
7.3. COMPARISON TO EXPERIMENTAL DATA: VORTICITY FIELD

\[ \alpha_0 = 30^\circ \quad DR = 1.0 \quad BR = 2.0 \quad IR = 4.0 \quad Z/d = 0.4 \]

**Experiment**

**CFD Prediction**

\[ \omega_x^* \]
\[ \omega_z^* \]

![Figure 7.8: Contours of normalized streamwise vorticity \((\omega_x^*)\) (top) and normalized vertical vorticity \((\omega_z^*)\) (bottom) for a streamwise injection angle of \(\alpha_0 = 30^\circ\) and momentum flux ratio of \(IR = 4.0\).](image)

7.3.2 Analysis of the measured and predicted vorticity field

**Vorticity generation**

It would be of primary importance to understand the generation process of the vorticity that is observed in Figs. 7.8 and 7.8. Indeed, in searching the flow regions where the vorticity is generated, it is intended to show that the numerically-immersed film cooling jet model is able to reproduce the physical process of vorticity generation. For this purpose, it is first proposed to carefully analyze the vorticity field displayed in Fig. 7.8. As it can be seen, the vorticity of the CVP structure is already existing before the plane of injection. This is found both in the experiment and in the computation. In the experimental results, it can be easily understood that the vorticity is coming from the coolant flow boundary layer in the hole. Even if an explicit clue of this vorticity generation process cannot be given here, it has been already shown by many authors (see for instance Morton and Ibbetson [67]) and used as one of the main assumption for the model development.
7.3. COMPARISON TO EXPERIMENTAL DATA: VORTICITY FIELD

![Figure 7.9: Contours of normalized streamwise vorticity \( \omega_X \) (top) and normalized vertical vorticity \( \omega_Z \) (bottom) for a streamwise injection angle of \( \alpha_0 = 50^\circ \) and momentum flux ratio of \( IR = 4.0 \).](image)

Now, in the computation, as the vorticity of the CVP structure is already present before the plane of injection, it could be concluded that it should come from the upstream freestream boundary layer. This is in contradiction with all the previous observations and assumptions made. In order to focus on this issue, two cross views of the flow, at an axial position \( X/d = -1.0 \) (leading edge of the hole) and at \( X/d = 0.0 \) (center of the hole), are plotted in Fig. 7.10. The predicted axial vorticity \( \omega_X \) is only shown here but it has been verified that the same vorticity field pattern is found in the experiment. At the leading edge of the hole (\( X/d = -1.0 \)), the vorticity lines pertaining to the incoming boundary layer flow starts to be deflected around the jet, so that an axial component of vorticity is seen near the flat plate surface. The sign of the axial vorticity indicates a typical horseshoe type of vortical system, even if, looking at the cross velocity vectors (not shown here), it is merely a lateral skewing of the freestream boundary layer. However, there is no trace of vorticity typical of a CVP structure. In observing the axial vorticity at the cross section \( X/d = 0.0 \) (center of the hole), it can be clearly seen that the bound-
7.3. COMPARISON TO EXPERIMENTAL DATA: VORTICITY FIELD

Figure 7.10: Contours of predicted normalized streamwise vorticity ($\omega^*_x$), in the cross sections $X/d = -1.0$ and $X/d = 0.0$, for a streamwise injection angle of $\alpha_0 = 30^\circ$.

Boundary layer vorticity is flowing at the sides of the jet, as it is also observed in the experiment. Furthermore, the appearance of the vorticity pertaining to the CVP structure is noticed at the sides of the toroidal jet surface. Indeed, this vorticity is not generated by the incoming freestream boundary layer but by the presence of the jet toroidal surface. This is a very important observation. As it has been assumed in Chapter 4, a slip boundary condition is applied at the jet surface. In itself, this slip boundary condition does not allow any generation of vorticity at the jet surface. However, the presence of the immersed toroidal jet enforces a turning of the viscous freestream boundary layer flow around it. Since the velocity vector inside the jet, at any of the ghost-node $X_p$, is set so that the freestream flow is deflected around the jet, the vorticity observed at the sides of the toroidal surface at $X/d = 0.0$ is effectively coming from the jet; the coolant jet vorticity is embedded within the imposed immersed boundary condition. The consequence of this analysis can be decoupled in two statements. On one hand, the immersed boundary condition implicitly induces the diffusion of the coolant jet vorticity through

---

$^5$This fact is actually observed in the computational results: CVP vorticity is observed inside the jet toroidal body
7.3. COMPARISON TO EXPERIMENTAL DATA: VORTICITY FIELD

the viscous freestream fluid. This conforms experimental observations. But on the other hand, the diffusion of this vorticity is not controlled, in contrast to the one injected on the plane of injection. More precisely, the coolant vorticity that is diffused from the immersed jet surface is directly linked to the shape of this surface. Indeed, the rate of flow turning, function of the jet surface shape, is a measure of the vorticity. This means that the total amount of vorticity fed by the model might be inaccurate. This issue is analyzed later in this section.

Impact of the streamwise injection angle upon the vortical structure evolution

In looking more closely at the strength evolution of the vortices downstream of the hole for the \( \alpha_0 = 30^\circ \) case (see Fig. 7.8), the streamwise vorticity \( \omega_{x}^* \) more rapidly decays in the experimental data that what it is found in the CFD prediction. On the contrary, in the \( \alpha_0 = 50^\circ \) case (see Fig. 7.9), the decay is shown to be of the same order and also persists much longer, in terms of the axial distance. This feature is constantly observed in the other horizontal planes \( Z/d \) near to the flat plate surface. To quantify this effect, lateral profiles of the normalized vorticity \( \omega_{x,Z}^* \) are plotted in Fig. 7.11. Two axial locations are represented, one just after the plane of injection, and the second one at \( X/d = 4.0 \). The lateral profile of the vorticity \( \omega_{x,Z}^* \)

![Normalized streamwise vorticity \( \omega_{x,Z}^* \) lateral profile in the normal plane \( Z/d = 0.4 \), for an injection angles \( \alpha_0 = 30^\circ \) (left) and \( \alpha_0 = 50^\circ \) (right).](image)

of the CVP structure exhibits a notable difference between the experiment and the CFD prediction just after the plane of injection. Indeed, the vorticity \( \omega_{x,Z}^* \) is much more flattened in the experiment that what is found in the computation (\( \Delta \omega_{x,Z}^* \)
from peak maxima to local central minima is about 140% overpredicted). This can be explained by the fact that the CVP model does not account for any downward freestream momentum push and/or any particular lateral spreading. In observing the downstream evolution of the vorticity \( \omega_{xz}^* \), at \( X/d = 4.0 \), its lateral profile is well captured in the \( \alpha_0 = 50^\circ \) case. In the \( \alpha_0 = 30^\circ \) case, it is experimentally observed to have almost spread out completely. In this case, the discrepancy in predicted maximum \( \omega_{xz}^* \) is about 60% too high.

**Isotropic versus anisotropic turbulence in the wake region**

It has been experimentally shown (see Kaszeta and Simon [49]) that isotropic turbulence is not true anymore in the wake region. Although no definitive statement can be drawn from the present data, there is a presumption that the strength of the vortices is even more reduced by the enhanced turbulent stress in lateral direction. This feature seems to be more pronounced in the low streamwise injection angle case \( (\alpha_0 = 30^\circ) \). Furthermore, it has been shown (Lakehal [52]) that isotropic turbulence models are not adequate to treat the near wall region in this type of turbomachinery flows; the turbulent fluxes, which counteract against the entrainment effect of the vortices, are not promoted enough. This might explain the fact that the CFD predictions in this study underpredict the decay of the near wall absolute vorticity and also the spreading of the vortices. In return, the velocity deficit in the wake zone is overpredicted, especially for the low streamwise injection angle case \( (\alpha_0 = 30^\circ) \).

![Figure 7.12: Circulation \( \Gamma_{xz}^* \).](image)

---

### Figure 7.12: Circulation \( \Gamma_{xz}^* \).
7.3. COMPARISON TO EXPERIMENTAL DATA: VORTICITY FIELD

Circulation

A comparison of the measured and predicted axial evolution of the circulation $\Gamma_{xz}^*$ is displayed in Fig. 7.12 for all momentum flux ratios and streamwise injection angles studied in this Chapter. One can see that the level of circulation is much more dependent of the momentum flux ratio than of the streamwise injection angle. Three main zones can be differentiated in these plots. The first zone is before the hole, that is when $X/d < -1.0$. The circulation is low ($\Gamma_{xz}^* < 0.20$) since the only vorticity region is embedded in the freestream boundary layer, principally within the lateral vorticity component $\omega_y^*$. The second zone is located between $-1.0 < X/d < 1.0$, where a high increase of the circulation, up to about $\Gamma_{xz}^* \approx 0.35$ for a low momentum flux ratio ($IR = 1.0$) and up to $\Gamma_{xz}^* \approx 0.85$ for a high momentum flux ratio ($IR = 4.0$) is observed. The increase of the circulation is due to the vorticity coming from the coolant jet. As it has been analyzed earlier, the immersed boundary condition are able to reproduce this phenomenon. However, it is shown here that the total amount of circulation fed by the coolant jet cannot be set with high accuracy when using the model. For a high streamwise injection angle ($\alpha_0 = 50^\circ$), the maximum circulation is predicted more than 20% too high, whereas for a low streamwise injection angle ($\alpha_0 = 30^\circ$), the maximum circulation is accurately predicted for a low momentum flux ratio ($IR = 1.0$) and 16% too high for a high momentum flux ratio ($IR = 4.0$). The third zone, located downstream of the hole ($X/d > 1.0$) is exhibiting the decay of the circulation, i.e. the decay of the action of the CVP structure. The slope $(\Delta \Gamma_{xz}^*/\Delta X \approx -0.005$ for $IR = 1.0$ and $\Delta \Gamma_{xz}^*/\Delta X \approx -0.067$ for $IR = 4.0$) of the decay is accurately predicted by the CFD, even though the total amount of circulation is too high. The decay is about 10 times higher in the $IR = 4.0$ case compared to the $IR = 1.0$ case.

Conclusion

The prediction of the CVP structure agrees with experimental observations. It is shown that the immersed jet body allows the deflection of the freestream boundary layer so as to reproduce a phantom trace of a horseshoe vortex, seen in the experiment. The shape of the immersed jet body induces the turning of the viscous flow around the jet. Although a slip boundary condition is applied at the jet boundary, the imposition of the immersed boundary conditions inside of the jet toroidal shape implicitly feed vorticity. As a consequence, even if the CVP circulation is carefully calibrated on the plane of injection, there is an overprediction of about
10% to 20% of the circulation downstream of the injection site due to the uncontrolled feeding of vorticity at the toroidal jet surface. The axial evolution of the strength of the CVP structure is better captured with a high streamwise injection angle \( (\alpha_0 = 50^\circ) \). This leads to higher discrepancies when predicting the velocity and vorticity field in the wake region for a low streamwise injection angle case \( (\alpha_0 = 30^\circ) \).

7.4 Comparison with other numerical strategies

7.4.1 Brief description of the other numerical techniques

Motivation

As part of the validation process, it is intended to show the prediction performance of the CFD-embedded film cooling model (FCM) in comparison to other numerical strategies to inject the coolant flow. The goal is not to give an exhaustive overview of the other strategies, it is merely wanted to pinpoint few but important issues for predicting accurately the aerodynamics of film-cooling jets, in a reasonable time scale. In particular, the predicted static pressure field near the hole is analyzed. Indeed, it could not be compared to experimental data. The prediction of the near hole static pressure is essential to ensure the right level of jet blowing. Then the velocity field is compared to highlight the prediction capability of the FCM strategy. Eventually, some important drawbacks of the two alternative injection strategies are discussed. For this purpose, only one injection case is selected, that is the S4 case, in which the streamwise injection angle is \( \alpha_0 = 30^\circ \), the density ratio is \( DR = 1.26 \) and the blowing ratio is \( BR = 2.0 \).

Wall injection (WI) and Full Injection (FI)

The *wall injection* (WI) strategy consists of injecting the coolant flow by setting the velocity vectors and temperature at the wall, where the hole should stand. This is the simplest approach for treating film-cooling jets in CFD. This approach has been the first to be utilized for flow problems involving film-cooling in turbines, see for instance Vogel [89]. More information about the definition of the strategy can be found in Appendix J. The *full injection* (FI) computation consists of meshing the plenum chamber and hole region. Up to date, this is the most complex method to deal with film-cooling jets. This approach is still far from being used in film-cooled turbines problems involving hundreds of holes. As a reference, one of the
first application of this method for predicting film-cooled turbine flow field has been shown by Walters and leylek [91]. More information about the computation set up can be found in Appendix K. A comparison of the three numerical coolant injection strategies, in terms of mesh size, numerical method, turbulence model, convergence level and total resolution time, is given in Table 7.1.

<table>
<thead>
<tr>
<th></th>
<th>FCM</th>
<th>WI</th>
<th>FI</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of grid nodes</td>
<td>2.6·10^5</td>
<td>2.6·10^5</td>
<td>7.7·10^5</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>Baldwin-Lomax</td>
<td>Baldwin-Lomax</td>
<td>SST k – ω</td>
</tr>
<tr>
<td>Residual (DQ_{rms})</td>
<td>10^{-5}</td>
<td>5·10^{-5}</td>
<td>7·10^{-5}</td>
</tr>
<tr>
<td>Total resolution time [h.min]</td>
<td>4.27</td>
<td>4.24</td>
<td>24.40</td>
</tr>
</tbody>
</table>

Table 7.1: Comparison of the different numerical strategies to inject the coolant jet.

7.4.2 Static pressure field

The blockage effect in the incoming freestream boundary layer due to the presence of the coolant jet is of primary importance as the potential field created adjusts the static pressure in the near hole region. Furthermore, pressure waves (wakes, jet, etc.) incoming to the hole exit modulate the near hole static pressure \(P_s\). As shown by Abhari and Epstein [1], this is a very important feature to determine the blowing ratio level. In connection to this fact, as it has been described in Chapter 3, The near hole static pressure \(P_s\) is a critical part of the film cooling model. Looking back at Eq. 3.19, it is seen that the near-hole static pressure \(P_s\) is a primary feature that sets the expansion of the coolant flow through the hole, so that the determination the blowing ratio, jet trajectory and so on. Thus, it is
important to validate the near hole static pressure field obtained with the use of the CFD-embedded film-cooling jet model.

\[
\alpha_0 = 30^\circ \quad DR = 1.3 \quad BR = 2.0 \quad IR = 3.2 \quad Z/d = 0.0
\]

Figure 7.13: Contours of normalized static pressure \( \Delta P_s \) for different injection strategies, with a streamwise injection angle \( \alpha_0 = 30^\circ \). On the bottom right, axial profile of \( \Delta P_s \) on the center plane \( Y/d = 0.0 \) is represented.
7.4. COMPARISON WITH OTHER NUMERICAL STRATEGIES

Contours of normalized near hole static pressure $\Delta P_s$ predicted with the three different injection strategies are plotted in Fig. 7.13. It is observed that the prediction using the Wall Injection (WI) strategy leads to a highly oscillating pressure field near the hole exit. In comparison with the two other numerical results, it is unlikely that it accurately represents the near hole static pressure field. In fact, the flow field near the injection is strongly disturbed by the forcing of the velocity and temperature field. Furthermore, a central scheme is used which is very sensitive to regions where flow with high gradients occurs (see for instance Hirsch [45]). Now, as illustrated in Fig. 7.13, the near hole static pressure field predicted in the Full Injection (FI) computation is found to be similar with the one obtained with the film-cooling model (FCM). The upstream high pressure zone ($\Delta P_{s,max} = 2.2\%$ for FI and $\Delta P_{s,max} = 1.9\%$ for FCM) is similar in both results. In the downstream low-pressure zone, moderate discrepancies\(^6\) are also observed but the shape of the contours of pressure matches. There are two branches of low static pressure zone which correspond to the action of the CVP structure, which enables an acceleration zone toward the center of the jet, near the wall. This numerical back-to-back comparison implies that the toroidal shape chosen to simulate the near hole jet body, and the associated immersed boundary condition treatment, are able to create a neat and relevant static pressure field near the hole.

7.4.3 Jet penetration and spreading

Contours of streamwise velocity

A comparison of measured and predicted contours of the streamwise velocity $U/U_f$ in the center plane $Y/d = 0.0$ is given in Fig. 7.14. The main flow regions analyzed in the previous section are found, that is the high velocity zone (windward side) and the low momentum region (wake). From a global perspective, it seems that the quality of the prediction is the worst with the WI strategy. Indeed, the wake zone shows to be much larger in size than with the two other injection strategies. In addition, the velocity peak in the windward side is less pronounced in comparison to the experiment and the two other injection strategies. The numerical results obtained with the FCM and FI strategies look pretty close to each other, even if the wake structure appears to differ.

\(^6\)For FCM, $\Delta P_{s,min} = -1.7\%$ and for FI, $\Delta P_{s,min} = -2.5\%$) after $X/d = 1.25$
Figure 7.14: Comparison of measured and predicted contours of normalized streamwise velocity $U/U_f$ at the center plane $Y/d = 0.0$. 

$\alpha_0 = 30^\circ \quad DR = 1.3 \quad BR = 2.0 \quad IR = 3.2 \quad Y/d = 0.0$

7.4. COMPARISON WITH OTHER NUMERICAL STRATEGIES
7.4. COMPARISON WITH OTHER NUMERICAL STRATEGIES

Velocity profiles

In order to quantify the differences in the prediction aforementioned, profiles of the streamwise velocity $U/U^f$ in the center plane $Y/d = 0.0$ are displayed in Fig. 7.15. In observing Fig. 7.15, one can see that numerical predictions highly differ just after the hole exit, at $X/d = 2.0$. The maximum velocity magnitude in the windward side of the jet, in the WI strategy, is 10% lower than the measured one, whereas in the FCM and FI strategies, it is 13% and 16% higher than the measured one. The streamwise velocity profile in the wake region is not matched accurately in any computation. The WI computational result exhibits an almost linear decrease of the streamwise velocity. Inspecting carefully the measurement data shows that the streamwise velocity in the wake region is pretty constant below $Z/d = 0.4$, whereas it is strongly decelerating from $Z/d = 0.6$ down to $Z/d = 0.4$. This behavior is better captured with the FCM computation, even if the streamwise velocity is about 15% overpredicted. The FI computational result shows also a better prediction of the wake zone, but in this case the streamwise velocity in the wake is about 8% underpredicted; the velocity deficit is too pronounced. Further downstream, the solution obtained with the WI strategy ends up constantly underpredicting the streamwise velocity. The solution accuracy using this strategy is left behind the two others numerical solutions by more than 18%. The windward side of the jet is almost perfectly predicted in the FCM and FI computations, whereas on the wake, some discrepancies persist. In the FCM computation, the usual small underprediction (6% on average) of the streamwise velocity in the wake is observed. In the FI computation, it is interesting to note that the streamwise velocity in the wake is rapidly increasing from $X/d = 2.0$ to $X/d = 4.0$, resulting in an overprediction of 4% on average.

Cross velocity

A cross view of the coolant jet is presented in Fig. 7.16, where the streamwise velocity $U/U^f$ contours are displayed, superimposed with the cross velocity vectors $(V/U^f, W/U^f)$. The high velocity zone $(U/U^f > 1.0)$, forming a kidney type of shape, is shown for the FCM and FI strategies, whereas it is not the case in the WI computation. In the FI computation, the kidney shape is the most pronounced, in the sense that the high velocity zone is bending even more that what is found in the experiment. Actually, in the FI computation, it is observed that the high velocity zone is so much pinching that it almost re-attaches in its lower side, near the wall. The process of this pinching is due to the CVP structure, so that its
Figure 7.15: Comparison of measured and predicted profiles of streamwise velocity ($U/U_j$) at different axial locations $X/d = 2.0, 4.0, 6.0$ and 14.0, in the center plane $Y/d = 0.0$, for different numerical injection strategies.

strength should be too highly predicted in the FI case. Unfortunately, the position of the two counter rotating vortices is hardly found in the experiment. In the CFD predictions, the center of the two vortices is found almost at the same location for the FCM and WI computations, that is at $Y/d = 0.49$ for both FCM and WI, and $Z/d = 0.57$ and $Z/d = 0.53$ for FCM and WI respectively. On the contrary, the prediction of the center of vortices in the FI case is quite different. Indeed, in this numerical result, the center is found at $Y/d = 0.31$, $Z/d = 0.50$, which corresponds to less spreading of the vortices. The shorter distance between the vortices leads to increase the counter-rotating motion of the fluid as well as the bending of the high velocity zone. As observed in section 7.2.1, the numerical solution of the coolant jet flow in the FCM computation ends up constantly overpredicting the center of the vortices. Thus, the FI computation should lead to a better result in predicting the CVP centers but no formal conclusion can be drawn here.

7.4.4 Drawbacks of the different injection strategies

Wall injection (WI)

There are two major drawbacks that have been observed when utilizing the WI strategy. The first one resides in the inaccurate imposition of the coolant boundary condition at the wall. Indeed, it has been shown that the flow field near the injection site gets a spurious oscillating behavior since the WI strategy induces very
sharp flow gradient at the virtual sides of the hole exit. This leads to a numerical wiggling problem at this location, see Fig. 7.14. A good numerical strategy for
coolant injection should be automatic. As a function of the computed near-hole static pressure, the blowing level of the jet is inferred. In the WI strategy, it is really not ensured that the blowing level of the jet can be accurately known from the computed near hole static pressure field. The second drawback that has been felt during the analysis of the velocity field is that the penetration seems more diffused since the windward side of the jet is much less pronounced than in the experiment and predictions using the other injection strategies. To shed light on this observation, a comparison of the predicted and imposed absolute velocity contours \( \frac{U_a}{\bar{U}_a} \) at the hole exit plane \((Z/d = 0.0)\) between the FI and WI computations is plotted in Fig. 7.17. The imposed coolant velocity profile in the WI strategy is constant by default \((\frac{U_a}{\bar{U}_a} = 1.0)\). Nevertheless, it is clearly observed that the FI prediction leads to have a non constant profile of the coolant velocity at the hole exit. Near the leading edge of the hole, a high absolute velocity zone is seen \((\frac{U_a}{\bar{U}_a} > 1.0)\), which will eventually shape the windward zone, whereas near the trailing edge of the hole, the absolute velocity reduces \((\frac{U_a}{\bar{U}_a} < 1.0)\). This fact has already been numerically observed by many authors (see for instance Walters and Leylek [90], Martelli and Adami [60]). As a consequence, the constant velocity profile imposed in the WI strategy is erroneous, which leads to higher discrepancies in the prediction compared to the other injection strategies.

Figure 7.17: Contours of normalized absolute velocity \( \frac{U_a}{\bar{U}_a} \), at the horizontal plane \( Z/d = 0.0 \) for the full injection (FI - left) and wall injection (WI - right) strategies, with a streamwise injection angle \( \alpha_0 = 30^\circ \).
Full Injection (FI)

The numerical results found with the FI strategy have been shown to be as accurate as the ones obtained with the FCM strategy. However, it is observed that the streamwise velocity in the wake is overpredicted. It results from the fact that the kidney shape of the jet is too pronounced. Thus, the CVP structure seems to play a stronger role in this strategy compared to the other strategies and in comparison to the experimental results. To better understand the action of the CVP structure, the axial evolution of the CVP circulation $\Gamma'_{XZ}$ extracted from the numerical results obtained with the different injection strategies and also from the measurements is plotted in Fig. 7.18. The circulation is too highly predicted for all the numerical strategies, that is by about 17% for the FCM strategy, 24% for the WI strategy and about 100% for the FI strategy. Meanwhile, the slope of the decrease of circulation as a function of the axial distance is accurately found in the FCM and WI strategy ($\Delta \Gamma'_{XZ}/\Delta (X/d) \simeq -0.026$) whereas it is much higher in the FI computation ($\Delta \Gamma'_{XZ}/\Delta (X/d) \simeq -0.053$). Thus, the CVP structure has too much strength in the FI prediction when emerging from the hole. In return, the shaping of the kidney shape is too pronounced. Martelli and Adami [60] have deeply studied the flow dynamic occurring inside the hole and plenum chamber. They have shown that the flow is already complex inside the hole, and has already counter-rotating vortical structures. Furthermore, the prediction of the flow through the hole is very sensitive to accurate geometrical definition of the hole, especially at its corner. A
good illustration of these facts is given in Fig. 7.19. Note here that the secondary flows, explicated by \( \left( \frac{U_b}{U_f}, \frac{U_t}{U_f} \right) \), are derived from the velocity vector projected onto the plane perpendicular to the main flow direction. As a consequence, the major drawback of the FI strategy lies on the complex definition of the geometry and numerical discrepancy arising in the hole, due to the complex flow structure generated at this location.

Figure 7.19: Illustration of the complexity of the FI simulation. Mach number in the center plane \( Y/d = 0.0 \) (left) and secondary flow vector \( \left( \frac{U_b}{U_f}, \frac{U_t}{U_f} \right) \) at the horizontal plane \( Z/d = -0.35 \) (right).

**Merit and benefit of the film-cooling model (FCM)**

The CFD-embedded film-cooling model combines the positive aspects of the two alternative numerical strategies tested (WI and FI). On one hand, it has been shown that the computational overhead when using the film-cooling model is very small, less than a percent. As a consequence, a numerical solution can be found in approximatively the same amount of computational time as with the simplest coolant injection approach, that is the Wall Injection strategy. On the other hand, the accuracy of the numerical prediction obtained is, at least, of the same order as the one obtained with the most complex injection strategy (FI). This is done in much less computational time (about 6 times faster). Indeed, the knowledge of coolant jet flow physics embedded in the film-cooling jet model, coupled with an efficient numerical immersion of the coolant jet model leads to a stable, neat and computationally efficient solution for prediction of the aerodynamic of film-cooling flows.
Chapter 8

Steady Heat Transfer Validation

8.1 Introduction

8.1.1 Thermal field versus heat transfer level prediction

One of the ultimate goals of the CFD-embedded film-cooling jet model is to provide a numerical tool for the designer and the researcher that allows predicting wall adiabatic film-cooling effectiveness $\eta$ in a reasonable time scale. The wall adiabatic film-cooling effectiveness is not a direct measure of the heat transfer level at a wall. It is merely a measure of the surface temperature obtained at a perfectly insulated wall. Hence, it quantifies the impact of the heat transfer process occurring nearby the wall, upon the thermal field. Film cooling performances are typically evaluated in laboratory studies by means of the wall adiabatic film-cooling effectiveness, but also with the heat transfer coefficient, Nusselt number or Stanton number. It is well known that, for turbomachinery flows, accurate heat transfer prediction are in general more difficult to obtain than aerothermal prediction, see for instance Ameri et al. [4], Schiele and Wittig [79]. It is recalled that heat transfer is function of the first order derivative of the temperature, so that discrepancies in predicting it arise more quickly than for temperature. The effect of the turbulent thermal mixing, as well as transition process, are still under way to be well understood and modeled (for a RANS framework) i.e. accurately predicted. Furthermore, heat transfer levels can be evaluated by many different means. This can be done by only considering the external forced convective heat process (and isothermal and/or diabatic wall boundary conditions) or also by taking into account the internal (wall material) heat conduction process\textsuperscript{1}. Thus, in a first step, although the accurate prediction of heat transfer is a major issue in turbomachinery, the present Chapter limits the validation of the use of the CFD-embedded film-

\textsuperscript{1}A heat transfer problem involving heat conduction and forced convection is so-called conjugate heat transfer problem.
cooling jet model for prediction of wall adiabatic film-cooling effectiveness and surface metal temperature. In connection to this, it is underlined here that "heat transfer validation" is an abuse of the language, it is merely a validation of the use of the film-cooling model to accurately predict the result of heat transfer process.

8.1.2 Validation procedure

Two experimental film-cooled flat plate test cases are used to demonstrate the capability of the numerically-immersed jet model to predict film-cooling effectiveness. They are both taken from the literature, so that they are pure validation test cases. The first experimental test case (Saumweber et al. [78]) is very similar to the test case of Bernsdorf [13]. Thus, it allows extending the aerodynamic validation done in the previous Chapter to film-cooling effectiveness prediction. The second experimental test case (Jung and Lee [48]) concerns streamwise but also lateral coolant injection. In this test case, the streamwise injection angle is $\alpha_0 = 35^\circ$, so that in between the two streamwise injection angles ($\alpha_0 = 30^\circ$, $50^\circ$) used for the calibration of the model. Furthermore, lateral coolant injection is investigated with lateral angles from $\beta_0 = 0^\circ$ (no lateral injection) up to $\beta_0 = 90^\circ$ (extreme lateral injection, just before counter-flow injection). The goal of this second validation step is to check to which extend the current film cooling model is able to perform accurate prediction of film-cooling effectiveness. It is recalled that there are only two extensions of the model for treating lateral injection: the lateral pressure resistance coefficient $C_{nj3}$ for predicting the lateral trajectory of the jet and the exchange of circulation (vortex strength) between the two vortices of the CVP structure using a $\sin$ square function, see Chapter 5. No upgrade of the mixing model and boundary conditions for the jet toroidal surface for lateral injection is proposed in this study. Overall, this heat transfer validation shall give an increased confidence in the use of the film cooling model for predicting turbomachinery flows.

8.2 Streamwise injection

8.2.1 Test case definition

Experimental apparatus

The University of Karlsruhe film-cooled flat plate experiment of Saumweber et al. [78] is briefly described. The test rig consists of a continuous flow wind tunnel. From a secondary flow loop, cool air is injected into the hot mainstream through a
row of three holes, each of them having a length of 6.0 hole diameters. A pitch distance of 4.0 hole diameters separates them. Each hole has a diameter of 5 [mm], and a streamwise injection angle $\alpha_0 = 30^\circ$. A turbulence grid is placed far upstream from the holes to control freestream turbulence intensity and length scale. As the turbulence model used in this study is a zero-equation model (Baldwin-Lomax), results with the lowest freestream turbulence, namely 3.5%, are utilized. An infrared camera allows to map the temperature at the flat plate surface, carefully insulated to respect quasi-adiabatic condition. Film-cooling effectiveness is inferred from the temperature mapping, with an uncertainty of $\pm 4\%$. Five measurements series are available, for a fixed density ratio of $DR = 1.7$ and five different blowing ratios, namely $BR = [0.5, 0.75, 1.0, 1.25, 1.5]$. Uncertainty in fixing the blowing ratio is $\pm 2\%$. Freestream and coolant inlet flow characteristics, such as total pressure and temperature, are provided by Saumweber [78]. The freestream Mach number is equal to $M = 0.3$. The Reynolds number, based on the freestream velocity $U_f$ and hole diameter $d$ is equal to $Re_d = 2.7 \cdot 10^4$. Only results with the lowest ($BR = 0.5$) and highest ($BR = 1.5$) blowing ratios are presented since they are the only available experimental data.

Computational domain

The same computational domain as the one utilized in Chapter 7 is used here. Indeed, the test case of Saumweber et al. [78] is almost identical to the one of Bernsdorf [13]. Numerical calculations are therefore carried out on the grid C2 (see Chapter 6 for a complete definition of the grid), having an axial density of $N_X = 5$ and lateral density of $N_Y = 9$. Total computational time and overhead due to the use of the film-cooling model, as well as residuals and mass flow error histories, are similar to the ones analyzed in Chapter 6.

8.2.2 Predictive capability of the model - standard cases

Impact of the momentum flux ratio upon the jet thermal field

The jet penetration and thermal mixing is first observed in Fig. 8.1. The thermal field is represented by the normalized total temperature $\theta$, defined by

$$\theta = \frac{T_f^J - T_T}{T_T - T_f}$$

(8.1)
Only the predicted thermal field is represented as no experimental data is available. This is shown for the lowest \( BR = 0.5 \) and the highest blowing ratio\(^2 \) \( BR = 1.5 \) investigated. A fundamental difference is noticed. In the low blowing ratio case \( BR = 0.5 \), the thermal field of the jet is attached to the wall surface, in which the jet core \( \theta_{\text{max}} = 0.73 \) almost flows on the flat plate surface. On the contrary, in the high blowing ratio case \( BR = 1.5 \), the numerical solution exhibits a jet thermal field almost completely detached from the flat plate surface. Furthermore, the CFD prediction in the \( BR = 1.5 \) case shows the pinching of the jet due to the enhanced CVP strength and high vertical penetration (shown in Chapter 7). Thus, the blowing ratio (momentum flux ratio) is strongly influencing the evolution of the jet thermal field.

\(^2\)The density ratio \( DR = 1.7 \) is held constant in this experiment, so that the impact of the momentum flux ratio level is proportional to the impact of the blowing ratio level.
Figure 8.2: Measured and predicted contours adiabatic film-cooling effectiveness $\eta$ on the flat plate surface ($Z/d = 0.0$) for a low blowing ratio ($BR = 0.5$, top) and high blowing ratio ($BR = 1.5$, bottom).
8.2. STREAMWISE INJECTION

Prediction of surface adiabatic film-cooling effectiveness

The predicted contours of adiabatic film-cooling effectiveness $\eta$ is compared to the experimental measurements in Fig. 8.2 for the lowest ($BR = 0.5$) and highest ($BR = 1.5$) blowing ratios. Overall, the contours pattern exhibits the same trend both in the measurement and prediction. In particular, the film-cooling effectiveness contour levels appear to be higher in the low blowing ratio case ($BR = 0.5$). It is interesting to note that the decrease of the film-cooling effectiveness $\eta$ contour levels in the axial (streamwise) direction seem more pronounced in the low blowing ratio case ($BR = 0.5$). Indeed, for the highest blowing ratio case ($BR = 1.5$), downstream of the axial position $X/d = 9.1$, the contour level of $\eta$ in the centerline $Y/d = 0.0$ does not change anymore ($\eta = 0.2$) whereas in the lowest blowing ratio case ($BR = 0.5$), for the same axial locations, contours of $\eta$ drop by two levels. In addition, it is felt that the spreading of the coolant thermal field is underpredicted. As an example of this observation, taking the $BR = 0.5$ case and $X/d > 14.0$, it is noticed that the contour $\eta = 0.05$ covers the whole hole to hole pitch distance ($-2.0 \leq Y/d \leq 2.0$) in the experiment whereas it is not the case in the CFD prediction.

Prediction of laterally-averaged adiabatic film-cooling effectiveness

In order to quantify these observations, it is very useful for the designer to compact the information presented in Fig. 8.2 in terms of the laterally-averaged film-cooling effectiveness $\bar{\eta}$ (see Eq. 6.6 in Chapter 6 for the definition of this term). It gives a sense of the "hole to hole" film-cooling effectiveness along the axial distance $X/d$. The measured and predicted laterally-averaged film cooling effectiveness $\bar{\eta}$ are plotted in Fig. 8.3. The laterally-averaged film-cooling effectiveness $\bar{\eta}$ is accurately predicted for the lowest ($BR = 0.5$) and highest ($BR = 1.5$) blowing ratios investigated. For the lowest blowing ratio case ($BR = 0.5$), the CFD prediction is almost identical to the experimental value, whereas in the highest blowing ratio case ($BR = 0.5$), the numerical computation leads to a underprediction of 15% on average, of $\bar{\eta}$. If this discrepancy seems to be high, it should to be recall that, at this very low value of $\bar{\eta}$ ($\bar{\eta} \leq 0.07$), a small discrepancy in the prediction of this highly three dimensional flow field leads to high relative error much more quickly than of high value of $\bar{\eta}$ are analyzed. In addition, the experimental uncertainties in fixing the blowing ratio ($\pm 2\%$) may lead to a slightly higher blowing ratio in the measurement. In Fig. 8.3, a different behavior of the thermal mixing between a low and a high blowing ratio is clearly identified. In the $BR = 0.5$ case,
the laterally-averaged film-cooling effectiveness is monotonically decreasing as a function of the axial direction $X/d$, where $\bar{\eta} = 0.23$ at $X/d = 2.0$ and $\bar{\eta} = 0.11$ at $X/d = 22.0$. On the contrary, $\bar{\eta}$ is slightly increasing over the whole axial distance investigated ($\bar{\eta} = 0.06$ at $X/d = 2.0$ and $\bar{\eta} = 0.07$ at $X/d = 22.0$). This behavior is reproduced by the CFD prediction. As noted by Sinha et al. [85], in comparison to a coolant jet with a low blowing ratio, a jet with a higher blowing ratio is synonymous of higher mass and energy fluxes which eventually result in a better film cooling protection far downstream. Indeed, the coolant jet with a high blowing ratio almost de-attached close to the hole, see Fig. 8.1, which dramatically lowers the film-cooling effectiveness. However, the molecular diffusion, fed by more coolant material, eventually counter-balanced the high vertical penetration of the jet.

**Surface metal temperature**

As it has already been discussed in Chapter 1, the surface metal temperature is the ultimate value for the designer. Indeed, the arrangement of film cooling holes he has designed should lead to a surface metal temperature that is below a given temperature threshold that the material can sustain in a long term without significant modification of its properties. Thus, the prediction of the surface metal temperat-
8.2. STREAMWISE INJECTION

ure with the model should be useful for the designer, i.e. the error in predicting the surface metal temperature should be minimal. In this context, the predictive error \( \epsilon_{\Delta T} \) of surface metal temperature, versus experimentally measured surface temperature is given by

\[
\epsilon_{\Delta T} = 100 \cdot \frac{T_{aw}^{CFD} - T_{aw}^{EXP}}{T_{aw}^{EXP}} = 100 \cdot \left( \frac{\eta^{CFD} - \eta^{EXP}}{\eta^{EXP} + \eta_{nec}^{EXP}} \right) \tag{8.2}
\]

where \( \eta_{nec}^{EXP} = T_{rec}/(T_T - T_{rec}) \) and is given by the experimental conditions. Based on the measured and predicted values of the laterally-averaged adiabatic film cooling effectiveness \( \bar{\eta} \) plotted in Fig. 8.3 and Eq. 8.2, the laterally-averaged predictive error \( \bar{\epsilon}_{\Delta T} \) of surface metal temperature is plotted in Fig. 8.4. As it can be observed, the predictive error \( \epsilon_{\Delta T} \) is lower than 1% for both blowing ratios investigated. In terms of absolute temperature, it corresponds to an error of prediction of less than 2.5 [K] for the low blowing ratio case \( (BR = 0.5) \) and to an error of less than 5.3 [K] for the medium blowing ratio case \( (BR = 1.5) \). This error might be larger in real engine because of higher heat load and more complicated flow field but this shows that the use of the model can deliver useful prediction of surface metal temperature for the designer.

![Figure 8.4: Laterally-averaged predictive error \( \epsilon_{\Delta T} \) of the surface metal temperature for for a low and medium blowing ratio \( (BR = 0.5, 1.5) \).](image)
Discrepancies in the prediction - near hole region

As it can be observed in Fig. 8.3, the prediction of the laterally-averaged film cooling effectiveness systematically suffers from a small and local discrepancy in the near downstream region \( (X/d \leq 4) \). This is attributed to the fact that the coolant to freestream mixing is suddenly specified on the 2D plane of injection. In this context, the numerical algorithm needs some computational cells to relax this immersed boundary condition.

Discrepancies in the prediction - coolant lateral spreading

It has been verified that the numerically-immersed film-cooling jet model is capable to accurately predict laterally-averaged values of adiabatic film-cooling effectiveness. However, as it is observed in Fig. 8.2, it seems that the accurate prediction of \( \bar{\eta} \) comes from different local values of \( \eta \) between the experiment and computation, in particular toward the lateral direction. To quantify this issue, the distribution of the centerline film-cooling effectiveness (strictly along \( Y/d = 0.0 \)) and also the lateral distribution of \( \eta \) in the cross line located at \( X/d = 8.0 \) are plotted in Fig. 8.5. In Fig. 8.5-I (left), the centerline film-cooling effectiveness \( \eta \) is observed to be systematically overpredicted in the computational result for the \( BR = 0.5 \) case, typical value 30%. However, the prediction of the centerline effectiveness is accurate within the experimental contours resolution \( (\Delta \eta = 0.1) \)
8.3. LATERAL INJECTION

in the BR = 1.5 case. In addition, in Fig.8.5-II (right), the lateral film-cooling effectiveness \( \eta \) is not enough spread, particularly in the low blowing ratio case (BR = 0.5). On one hand, the film-cooling effectiveness \( \eta \) at the center location \( Y/d = 0.0 \) is about 33% overestimated and, on the other hand, at the lateral position \( Y/d = \pm 1.5 \) \( \eta \) is found to be null\(^3\) in the computation whereas there is still a trace of cooling in the experiment (\( \eta = 0.05 \)). Interestingly, this discrepancy in the lateral diffusion of the coolant is a bit less pronounced in the high blowing ratio case (BR = 1.5). Although there is a small discrepancy in the prediction of the local film-cooling effectiveness at \( X/d = \pm 1.0 \) (where \( \eta = 0.0 \) in the computation and \( \eta = 0.05 \) in the measurements), in the center position \( X/d = 0.0 \), it is predicted accurately to within the experimental resolution. This trend of over-prediction of centerline film-cooling effectiveness and underprediction of lateral spreading of the coolant is in line with previous numerical studies on film-cooling jets, using an isotropic turbulence model, see for instance Lakehal [52], Walters and Leylek [90]. The comments made in Chapter 7 on the inaccurate prediction of the evolution of the counter-rotating vortex pair and the anisotropic behavior of the turbulent flow field near to the wall might help to shed light on this result. The near wall pinching is too pronounced and the lateral turbulent diffusion is not adequately promoted.

8.3 Lateral injection

8.3.1 Test case definition

Experimental apparatus

The Seoul National University film-cooled flat plate experiment of Jung and Lee [48] is briefly described. The test rig consists of an open-circuit, subsonic wind tunnel. From a secondary duct, hot air is injected into the cold mainstream through a row of five holes, each of them having a length of 4.0 hole diameters. A pitch distance of 3.0 hole diameters separates them. Each hole has a diameter of 20 [mm], and a streamwise injection angle \( \alpha_0 = 35^\circ \). Experiments are conducted at a fixed freestream mean velocity of \( U_f = 10 \) [m/s] (very low Mach number, \( M \approx 0.05 \)) and at a fixed freestream and coolant temperature of \( T_f = 293 \) [K] and \( T_c = 313 \) [K] respectively. The resultant density ratio for all measurements is \( DR = 0.93 \). A boundary layer trip wire of 1.8 [mm] diameter is located 30 hole diameters upstream of the holes. The measurement plate, starting at \( X/d = 1.0 \)

\(^3\)It is actually a bit lower than 0.0: \( \eta = -0.02 \) minimum
and insulated with formed polystyrene, is covered with a Thermochromic Liquid Crystal (TLC) sheet (100 [μm]-thick polyester), as well as black paint, adhesive layer and beneath it a polycarbonate plate. The temperature is measured at the plate surface by mapping color changes of the TLC sheet, the uncertainty being about 0.17 [K]. The resulting uncertainty in film-cooling effectiveness $\eta$ is 2.5% for $\eta = 0.5$ and 29.5% for $\eta = 0.05$. T-type thermocouple are used to measure the coolant and freestream total temperature $\theta$ field downstream of the injection site, at $X/d = 2.5, 50, 10.0$. Uncertainty in the measurement is about 6.4% for $\theta = 0.25$. Twelve measurements series are available, at four different lateral injection angles, namely $\beta_0 = 0^\circ, 30^\circ, 60^\circ, 90^\circ$ and three different blowing ratios, namely $BR = 0.4, 0.9, 1.9$.

**Computational domain**

The same computational domain as the one utilized in Chapter 7 is used here since it has been shown to be well suited for this type of injection problem. Numerical calculations are therefore carried out on the grid C2 (see Chapter 6 for a complete definition of the grid), having an axial density of $N_X = 5$ and lateral density of $N_Y = 9$. Meanwhile, the width of the channel is reduced to 3 hole diameters as in the experiment. Since the freestream Mach number is very low in the experiment ($M \approx 0.05$), it is artificially increased up to $M \approx 0.3$ to stabilize and speed up the computation. In the meantime, the hole diameter is reduced down to $d = 5$ [mm] as well as the total pressure level so as to ensure the same Reynolds number $Re_d$ (based on hole diameter $d$ and freestream Mach number $U_f$) as in the experiment. The scaling of the flow conditions made for the computation is given in Table 8.1. Total resolution time and overhead due to the use of the film-cooling model, as well as residuals and mass flow error histories, are similar to the ones analyzed in Chapter 6.

**8.3.2 Predictive capability of the model - extreme cases**

**The three-dimensional flow field**

The flow structure existing near the hole exit and downstream of it changes passably compared to a strictly streamwise injection case. The predicted freestream streamlines in the near hole region, for a lateral injection angle of $\beta_0 = 60^\circ$ and a blowing ratio of $BR = 1.9$, are displayed in Fig. 8.6. It is noticed that the deflection of the freestream flow around the immersed jet toroidal surface produces an
asymmetric flow. Indeed, a large scale vortex, so-called "Back Side" vortex $V_{BS}$, starting in the downstream side of immersed jet body is clearly identified. In the upstream (front) side of the immersed jet body, there is not any objective visualization of a vortex structure, but merely a deviation of the freestream streamlines. The back side vortex is induced from the coolant jet boundary layer in the hole. Its counter-part is no more existing. Lee et al. [54] propose a model of the near hole three-dimensional flow pattern for different lateral injection angles (from $\beta_0 = 0^\circ$ to $\beta_0 = 90^\circ$). They claim that between a lateral injection angle of $\beta_0 = 15^\circ$ and $\beta_0 = 30^\circ$, the CVP structure transforms to a single vortex rotating in the sense of the back side vortex in Fig. 8.6. For strictly streamwise injection, the two counter-rotating vortices have the same strength, or circulation. When there is a lateral component to the coolant injection, there is a rapid concentration of vorticity to the vortex located in the "back side" part of the jet at hole exit. This exchange of circulation, as well as the lateral trajectory of the jet, is shown to be executed in the model thanks to the circulation redistribution using a $\sin$ square function (see Eq. 5.3 in Chapter 5) and the jet trajectory model.

### Impact of the lateral injection angle $\beta_0$ on the aero-thermal field

A significant change in the mixing between the hot freestream flow and the coolant due to the structural change of the vortical flow and orientation of the coolant injection is expected. In this context, measured and predicted contours of total temperature $\theta$ are plotted in Figs. 8.7, 8.8 and 8.9 at a cross section located at

<table>
<thead>
<tr>
<th></th>
<th>Experiment</th>
<th>Computation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hole diameter [mm]</td>
<td>20</td>
<td>5</td>
</tr>
<tr>
<td>Freestream Mach number [-]</td>
<td>0.05</td>
<td>0.29</td>
</tr>
<tr>
<td>Freestream total pressure [bar]</td>
<td>1.00</td>
<td>0.41</td>
</tr>
<tr>
<td>Reynolds number $Re_d$ $BR = 0.4$</td>
<td>6'400</td>
<td>6'400</td>
</tr>
<tr>
<td>Reynolds number $Re_d$ $BR = 0.9$</td>
<td>12'700</td>
<td>12'700</td>
</tr>
<tr>
<td>Reynolds number $Re_d$ $BR = 1.9$</td>
<td>25'400</td>
<td>25'400</td>
</tr>
</tbody>
</table>

Table 8.1: Scaling of the flow conditions made for the computation.
8.3. LATERAL INJECTION

\[ \alpha_0 = 35^\circ \quad \beta_0 = 60^\circ \quad DR = 0.9 \quad BR = 1.9 \quad IR = 3.2 \]

Figure 8.6: Predicted freestream streamlines near hole exit, for a blowing ratio of \( BR = 1.0 \) and lateral injection angle of \( \beta_0 = 60^\circ \).

\( X/d = 5.0 \). To show the impact of increasing lateral injection angle and blowing ratio, results for \( \beta_0 = 0^\circ \) and \( BR = 0.4 \) are shown in Fig. 8.7, results for \( \beta_0 = 30^\circ \) and \( BR = 0.9 \) are shown in Fig. 8.8 and results for \( \beta_0 = 60^\circ \) and \( BR = 1.9 \) are shown in Fig. 8.9. Cross velocity vectors \( (V/U_f, W/U_f) \) are superimposed in the CFD prediction. Indeed, the effect of the lateral angle \( \beta_0 \) variation is found very similar for any fixed blowing ratio and vice-versa. Due to the small blowing ratio in the flow case with \( BR = 0.4 \) and \( \beta_0 = 0^\circ \), see Fig. 8.7, the coolant fluid is attached to the wall surface. This is seen both in the experimental and CFD results. Furthermore, the strength of the CVP structure is relatively small in regards
8.3. LATERAL INJECTION

\[ \alpha_0 = 35^\circ \quad \beta_0 = 00^\circ \quad DR = 0.9 \quad BR = 0.5 \quad IR = 0.2 \quad X/d = 5.0 \]

**Figure 8.7:** Measured and predicted contours of non-dimensionalized temperature \( \theta \) at the cross section \( X/d = 5.0 \) for a lateral injection angle of \( \beta_0 = 0^\circ \) and a blowing ratio of \( BR = 0.4 \).

\[ \alpha_0 = 35^\circ \quad \beta_0 = 30^\circ \quad DR = 0.9 \quad BR = 0.9 \quad IR = 0.9 \quad X/d = 5.0 \]

**Figure 8.8:** Measured and predicted contours of non-dimensionalized temperature \( \theta \) at the cross section \( X/d = 5.0 \) for a lateral injection angle of \( \beta_0 = 30^\circ \) and a blowing ratio of \( BR = 0.9 \).

...to the higher blowing ratio solutions. This induces a weak pinching of the coolant jet. The coolant jet is simply convected toward the axial direction and diffused
8.3. LATERAL INJECTION

\[ \alpha_0 = 35^\circ \quad \beta_0 = 60^\circ \quad DR = 0.9 \quad BR = 1.9 \quad IR = 3.7 \quad X/d = 5.0 \]

Figure 8.9: Measured and predicted contours of non-dimensionalized temperature \( \theta \) at the cross section \( X/d = 5.0 \) for a lateral injection angle of \( \beta_0 = 60^\circ \) and a blowing ratio of \( BR = 1.9 \).

Laterally. In the \( BR = [0.9, 1.9] \) and \( \beta_0 = [30^\circ, 60^\circ] \) cases, see Figs. 8.8 and 8.9, the effect of the lateral injection angle \( \beta_0 \) is clearly observed. The shape of the isothermal contours becomes asymmetric, and this asymmetry is amplified as the lateral injection angle increases. The secondary flow vectors, displayed only in the CFD prediction, show that already with a lateral injection angle of \( \beta_0 = 30^\circ \), the presence of a "front side” vortex \( V_{FS} \) (counter-part of the trailing edge vortex) is hardly seen. The position of this front side vortex \( V_{FS} \) is found very close to the wall, at \( Y/d = 1.47, Z/d = 0.1 \). In the experiment of Lee et al. [48], the first measurement point is at \( Z/d = 0.15 \). Thus, although it is confirmed by the CFD that there is only one vortex for a lateral injection angle of \( \beta_0 = 60^\circ \), it is still unclear at which exact lateral injection angle orientation the front side vortex disappears. From the CFD data, it is however certain that if the front side vortex is still existing at \( \beta_0 = 30^\circ \), its strength is more than 57% smaller\(^4\) than the back side vortex. The assumption of a rapid dying of the front side vortex, made in Chapter 5 with the sin square modulation of the circulation between the two vortices, is therefore qualitatively confirmed. For a lateral injection angle higher than \( \beta_0 = 30^\circ \), there is only one strong counter-clockwise rotating vortex (seeing

---

\(^4\)This is measured from CFD data. Furthermore, in this situation it is objectively not possible to separate the boundary layer vorticity due to its lateral squeezing and the vortical motion. Thus the strength of the front side vortex should be even much more smaller
8.3. LATERAL INJECTION

from downstream) that leads squeezing the coolant to freestream mixing zone. In the near wall region\( (Z/d \leq 0.35 \text{ for } \beta_0 = 30^\circ, BR = 0.9 \text{ and } Z/d \leq 0.85 \text{ for } \beta_0 = 60^\circ, BR = 1.9) \), the back side vortex is shifting the coolant jet toward the positive lateral direction so that the boundary layer is skewed. On the contrary, above the back side vortex center, the coolant flow mixing is enhanced toward negative lateral direction, which tends to bring backward (toward the central position \( X/d = 0.0 \)) coolant fluid materials. As a consequence, the effect of the vortex is mainly to increase the mean mixing area between the coolant and freestream fluids, so that it can be expected that coolant diffusion is enhanced with the increase of the lateral injection angle \( \beta_0 \).

**Discrepancies in the prediction - coolant lateral mixing**

In all Figs. 8.7 to 8.9, the lack of lateral diffusion and transport of coolant fluid is systematically observed. There are two main indications of this statement. The first one lies on the fact that the maximum temperature \( \theta_{\text{max}} \) in the core of the coolant jet is predicted 30% too high on average. As mentioned by Jung and Lee\(^{[48]}\), the maximum temperature in the jet core should slightly decrease with the increase of \( \beta_0 \), at a fixed axial position \( X/d \). This fact is hardly found in the CFD. Thus, as the lateral injection angle \( \beta_0 \) increases, the accuracy of the CFD in predicting the thermal mixing field decreases. The second indicator of a numerical discrepancy comes from the lack of lateral spreading of the predicted isothermal contours. This observation becomes more and more obvious as the lateral injection angle increases. For example, in the \( \beta_0 = 60^\circ, BR = 1.9 \) case, the isothermal contours higher than \( \theta = 0.0 \) covers the whole measured cross section below the coolant jet core, whereas it is not the case in the CFD prediction. Therefore, as it has been shown for streamwise injection, the lateral spreading of coolant is not promoted enough. This discrepancy increases with the increase of the lateral injection angle. In addition, although there is a lack of experimental data in the area of the hole \( (-1.0 < X/d < 1.0) \), it is presumed that the boundary conditions imposed at the jet toroidal surface become not anymore valid when the lateral injection angle increases. The collide of the coolant and freestream flow at this location may certainly lead to significant differences relatively to the assumptions made in the modeling. This is an open question for further model development, but it can be proposed that a penetration boundary condition could be investigated first.
Prediction of adiabatic film-cooling effectiveness

The measured and predicted contours of adiabatic film-cooling effectiveness $\eta$ on the flat plate surface are shown in Figs. 8.10 to 8.12 for the same flow regimes studied previously. In addition, comparisons of the laterally-averaged film-cooling effectiveness $\bar{\eta}$ between the experiment and computation are plotted in Fig. 8.13 for all the lateral injection angles $\beta_0$ and blowing ratios $BR$ investigated. For $U_o = 35^\circ$, $\beta_0 = 0^\circ$, $DR = 0.9$, $BR = 0.5$, $IR = 0.2$, $Z/d = 0.0$.

![Figure 8.10: Measured and predicted contours of adiabatic film-cooling effectiveness $\eta$ on the flat plate surface ($Z/d = 0.0$) for a lateral injection angle of $\beta_0 = 0^\circ$ and a blowing ratio of $BR = 0.4$.]

For no lateral injection angle cases ($\beta_0 = 0^\circ$), taking into account the experimental uncertainties, the prediction gets the same solution as what is found in the experiment. The lack of lateral diffusion of the coolant fluid is seen in Fig. 8.10 but this was to be expected as shown previously. In inspecting Figs. 8.11 and 8.13, it is found that the film-cooling model is capable of giving an fairly accurate prediction of film cooling effectiveness for a lateral injection angle of $\beta_0 = 30^\circ$ and $BR = 0.9$. Indeed, it is observed in Fig. 8.11 that the temperature trace of the coolant jet trajectory is predicted similar to the measured one, taking into account the lack of lateral diffusion. However, the prediction of $\bar{\eta}$ (see Fig. 8.13) suffers from an underprediction for the highest blowing ratio case ($BR = 1.9$), as seen in Fig. 8.13. For this high blowing ratio, the CFD prediction does not manage to capture the augmentation of the film cooling protection in the far downstream region ($X/d > 10$), where, in the experiment, $\Delta \bar{\eta}/\Delta (X/d) \simeq 0.012$ and in the CFD prediction, $\Delta \bar{\eta}/\Delta (X/d) \simeq 0.006$. For higher lateral injection angle cases,
8.3. LATERAL INJECTION

\[ \alpha_0 = 35^\circ \quad \beta_0 = 30^\circ \quad DR = 0.9 \quad BR = 0.9 \quad IR = 0.9 \quad Z/d = 0.0 \]

**Figure 8.11:** Measured and predicted contours of adiabatic film-cooling effectiveness \( \eta \) on the flat plate surface \( (Z/d = 0.0) \) for a lateral injection angle of \( \beta_0 = 30^\circ \) and a blowing ratio of \( BR = 0.9 \).

\[ \alpha_0 = 35^\circ \quad \beta_0 = 60^\circ \quad DR = 0.9 \quad BR = 1.9 \quad IR = 3.7 \quad Z/d = 0.0 \]

**Figure 8.12:** Measured and predicted contours of adiabatic film-cooling effectiveness \( \eta \) on the flat plate surface \( (Z/d = 0.0) \) for a lateral injection angle of \( \beta_0 = 60^\circ \) and a blowing ratio of \( BR = 1.9 \).

that is for \( \beta_0 = 60^\circ, 90^\circ \), the discrepancies in the prediction largely increases. Looking first at Fig. 8.12, it is confirmed that the film-cooling protection spans better toward the lateral direction, since the variation of the film-cooling effective-
8.3. LATERAL INJECTION

Figure 8.13: Measured and predicted levels of laterally-averaged adiabatic film-cooling effectiveness $\bar{\eta}$ for a lateral injection angle of $\beta_0 = 0^\circ$ (top left), $\beta_0 = 30^\circ$ (top right), $\beta_0 = 60^\circ$ (bottom left), $\beta_0 = 0^\circ$ (bottom right).

...ness contour levels are smaller$^5$ than for lower lateral injection angle cases$^6$. This trend is seen in the numerical solution but by inspection of the contour shape, this is not accurately predicted. This results in an inaccurate prediction of the laterally-averaged film-cooling effectiveness $\bar{\eta}$, as observed in Fig. 8.13. This is especially true for high blowing ratio where the trend of the axial evolution of $\bar{\eta}$ is not even captured.

$^5$typical value at $X/d = 5.0$ is $\Delta \bar{\eta} = 0.1$ for $\beta_0 = 60^\circ$

$^6$typical value at $X/d = 5.0$ is $\Delta \bar{\eta} = 0.25$ for $\beta_0 = 30^\circ$
8.4 Conclusion

It has been shown that the CFD-embedded film-cooling jet model is a numerical tool capable of giving accurate prediction of laterally-averaged adiabatic film-cooling effectiveness for different streamwise injection angles and momentum flux ratios. In fact, the error in predicting the surface metal temperature is less than 1% (or 5.3 [K] in the investigated case) for all blowing ratio investigated. The main discrepancy in the prediction of film-cooling effectiveness is the lack of lateral transport of coolant material. This is not mainly due to inappropriate modeling of the jet but it is mostly due to the modeling of the fluid dynamic occurring downstream of it, in particular the turbulent transport and diffusion. In addition, adiabatic film-cooling effectiveness prediction can even be done fairly accurately for coolant jets having a lateral angle component of less than $\beta_0 = 30^\circ$ and moderate blowing ratio, less than $BR = 1.9$. For higher lateral injection angle $\beta_0$, upgrades of the numerically-immersed jet model need to be carried out. In particular, the mixing process at the immersed jet surface has to be revised. This can be done by, for example, setting a penetration boundary condition at the immersed jet body. An upgraded modeling of the mixing taking place on the plane of injection should also be considered.
Chapter 9

Toward the Use of the Model in an Unsteady Flow Environment

The pulsating jet problem is introduced in this Chapter. As it is further demonstrated, film-cooling jets often pulsate in film-cooled turbines, due to the large scale fluctuation of pressure over the blade surface. A necessary introduction to these non-trivial flows is first presented. The range of flow regimes encountered are delimited. For a quasi-steady jet behavior, a one-dimensional model of the evolution of the jet blowing ratio is derived. This "blowing ratio model" serves to validate the use of the CFD-embedded film-cooling jet model in such flows. In this context, the overhead when using the model is evaluated, for different pulsating frequencies. A modification of the current film-cooling jet model is proposed to take account of the adjustment of the coolant blowing ratio as a function of near-hole static pressure. Several quasi-steady jet pulsation regimes are analyzed in terms of streamwise velocity and adiabatic film-cooling effectiveness.

9.1 Introduction to pulsating jet in turbines

9.1.1 Impact of flow unsteadiness in high-pressure turbines

Modes of flow unsteadiness

The high-pressure section of a turbine gets one of the most complicated flow environment of all energy conversion machines. In particular, the flow is significantly unsteady, often transonic, and overall at high temperature and pressure levels. The flow interaction between the stator and the downstream moving rotor is the main cause of unsteadiness. Several modes of unsteadiness due to rotor-stator interaction can be delimited. These are periodic potential flow interaction between the two blade rows, wake passing from the upstream blade row, oscillation of the shock system between the two blade rows and the unsteady feeding of vorticity gener-
ated at upstream blade walls (passage, tip and leakage vortices) to the downstream blade row. The effect of these modes of unsteadiness upon blade lives require careful investigations. Blade life of a high pressure turbine is strongly linked to the performance of its cooling system. Thus, it is important to study the effect of flow unsteadiness upon the behavior of film-cooling jets. The different modes of unsteadiness encountered in turbines are summarized in Table 9.1. In the follow-

<table>
<thead>
<tr>
<th>Unsteady mode</th>
<th>Physical measure</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>Potential flow interaction</td>
<td>Static pressure</td>
<td>midspan, endwalls, mainly on upstream blade</td>
</tr>
<tr>
<td>Passing shock waves</td>
<td>Static pressure</td>
<td>midspan, endwalls</td>
</tr>
<tr>
<td>Passing wakes</td>
<td>Total pressure</td>
<td>midspan, endwalls</td>
</tr>
<tr>
<td>Shedding vortices</td>
<td>Total pressure (Vorticity)</td>
<td>endwalls</td>
</tr>
</tbody>
</table>

Table 9.1: Different modes of turbine unsteadiness.

ing, only the midspan region of a rotor blade is considered. The effect of flow unsteadiness near the endwalls is not investigated in the present work.

**Impact of large scale pressure fluctuation**

In their seminal work, Abhari and Epstein [1] have experimentally shown that large scale static pressure fluctuations on the blade surface, resulting from passing shock waves and potential interaction, lead to a 12% increase and 5% decrease of the time-averaged heat transfer rate in the suction side and the pressure side respectively, compared with values with no static pressure fluctuations¹. Large scale fluctuations are defined as being flow fluctuations \( \tilde{\phi} \) whose time of appearance (inverse of their frequency) are longer than the time taken by a coolant particle to flow through the hole and downstream. More formally, any flow quantity \( \phi \) can be decomposed on a mean value \( \bar{\phi} \), a large scale, phase-averaged, fluctuation value \( \tilde{\phi} \) and a small scale fluctuation value \( \phi' \).

\[
\phi = \bar{\phi} + \tilde{\phi} + \phi' = \bar{\phi} + \phi' \quad (9.1)
\]

¹This means no presence of upstream shocks waves, wakes and potential interaction, so that no upstream blade row
In this context, Abhari and Epstein [1] propose a graphical model of the typical behavior the large scale fluctuation of static pressure $P_s$ at the blade surface, derived from an unsteady CFD prediction, as shown in Fig. 9.1. The authors explain that

![Diagram showing graphical model of midspan rotor surface pressure](image)

**Figure 9.1:** Graphical model, derived from unsteady CFD, of the midspan rotor surface pressure. Adapted from Abhari and Epstein [1].

the large scale fluctuation of the static pressure $\tilde{P}_s$ at the blade surface modulates the blowing rate of film-cooling jets as well as the downstream heat transfer rate. Indeed, as it is hypothesized in the film-cooling jet model in Chapter 3 (see Fig. 3.19 for a clear overview), the coolant jet expansion through the hole is mainly driven by the static pressure level $P_s$ at hole exit. Furthermore, the change of total pressure level in the plenum $P_f$ is generally negligible, due to its relatively large volume and also because of a constant feeding rate of coolant. Focusing back on Fig. 9.1, it is noticed that the pressure fluctuation $\tilde{P}_s$ is large around the mean pressure $P_s$ level all along the pressure side, and the highest in the first half of the suction side. In the second half of the suction side, that is after the throat, the large scale pressure fluctuation is shown to be insignificant. In summary, since the fluctuation of the static pressure is shown to be large, one could argue that the blowing of the coolant jet may largely be affected. Significant change of the time
mean and instantaneous film-cooling protection may result, compared to a steady state regime.

**Idealization of large scale pressure fluctuation to the first harmonic**

It is well known that the pressure signal at the blade surface induced by passing shock waves, wakes, and in a lesser extend by potential interactions, is characterized by a large band of frequencies and amplitudes, see for instance Johnson et al. [47]. The frequency spectrum of the pressure signal $\tilde{P}$ is strongly dependent of the blade and casing geometry as well as the flow conditions. In the present study, it is mainly aimed to demonstrate that the CFD-embedded film-cooling jet model can be used in an unsteady flow environment, typical of high-pressure turbine. Thus, as a first step, in order to give a comprehensive validation of the use of the model in such unsteady flows, the large scale fluctuation of the pressure signal $\tilde{P}$ is idealized and modeled by only taking into account its first harmonic, typically a function of blade passing frequency.

**Importance of the pulsating jet problem**

In order to show the importance of coolant jet pulsation, Abhari [2] has performed a two-dimensional unsteady heat transfer prediction of a film-cooled rotor, using an extended version of the two-dimensional film-cooling model proposed by Tafti and Yavuzkurt [87]. In validating the CFD prediction with experimental data, it is found that the film-cooling effectiveness can be locally decreased by as much as 64% in the pressure side when taking into account the pulsation of the jet. At this particular location, it is found that the coolant blowing is periodically shut down due to the very high value of near hole static pressure (see also Fig. 9.1). Few attempts have been made to measure directly pulsating film-cooling jet flows and there is not any computational study of this type of flow problem known by the author. The research group of University of Utah (spread later to Seoul National University) is the only one known by the author that has experimentally measured and analyzed pulsating jet flows, in a flat plate test section, see Ligrani et al. [57] [58], Bell et al. [10] and Seo et al. [84]. A slight deterioration of film cooling protection is found when the pulsation of freestream static pressure is increased to a level where the jet behavior at the hole exit is non-quasi steady. For quasi-steady freestream and coolant condition and long cooling holes (as assumed in this study), hardly any difference of film cooling protection is seen, compared to a steady coolant jet. Meanwhile, only low Mach number ($M^f < 0.04$), relatively
Reynolds number \((Re_d < 6800)\) and moderate blowing ratio \((BR < 1.0)\) flow regimes have been investigated.

9.1.2 Relevant flow regimes of a pulsating jet

**Idealization of the blade surface to a flat plate**

In the present work, the pulsation of coolant jets at blade surface is reduced to the analysis of an infinite row of pulsating jets in a flat plate, so that no curvature and/or mean pressure gradient effects are included. Meanwhile, the same flow regimes of jet pulsation as the ones encountered in turbine shall be approached. In fact, the pulsation of a coolant jet can be characterized by two main values which are the frequency of the pulsation and its amplitude.

**Range of pulsating frequency**

The level of the jet pulsating frequency \(F_j\), set by the frequency of the fluctuation of static pressure \(\bar{P}_s\), induces three main time scales. The first time scale determines the level of unsteadiness of the freestream flow field. In the case of a film-cooled flat plate, the length \(L\) of the plate sets the freestream reduced frequency \(\Omega^f\).

\[
\Omega^f = \frac{F_j L_{fp}}{U_f}
\]

where \(L_{fp}\) is the length of the flat plate. This freestream reduced frequency \(\Omega^f\) determines the quasi \((\Omega^f \ll 1.0)\) or non-quasi \((\Omega^f \gg 1.0)\) steady behavior of the freestream flow. The second important time scale is linked to the time required by the coolant fluid to go through the hole. It is quantified by the coolant reduced frequency \(\Omega^c\)

\[
\Omega^c = \frac{F_j L_c}{U_c}
\]

where \(L_c\) is the length of the hole. As it has been previously mentioned, and also hypothesized in Chapter 3, the temporal variation of the near-hole static pressure \(\bar{P}_s\) should be much smaller than the time required for the coolant to flow through the hole (quasi-steady assumption in the near hole region). This means that the present model can only be used for \(\Omega^c < 1.0\). In addition, the frequency \(F_j\) of the large scale pressure fluctuations should be small enough to drive (modify) directly
9.1. INTRODUCTION TO PULSATING JET IN TURBINES

The coolant mass flow rate\(^2\). To deal with this issue, the near hole static pressure reduced frequency \(\Omega^{a}\) is introduced

\[
\Omega^{a} = \frac{F^j L^c}{\bar{a}^c} = M^c \cdot \Omega^c
\]  

(9.4)

where \(\bar{a}^c\) is the speed of sound based on coolant flow condition\(^3\). In their film-cooled rotor test rig, typical of a high pressure gas turbine, Abhari and Epstein [1] have measured that the near hole pressure reduced frequency \(\Omega^{a}\) is equal to 0.06\(^4\) and must be below 1.0. This demonstrates that the large scale pressure variation \(\bar{P}_s\) is indeed directly modifying the coolant flow rate. In connection to this, Ligrani et al. [57] have proposed that typical value of \(\Omega^{a}\) in operating turbines ranges as follows

\[
0.02 \leq \Omega^{a} \leq 0.1
\]  

(9.5)

Thus, taking into account the range of \(\Omega^{a}\), expressed in Eq. 9.5, and the fact that the mean coolant Mach number can broadly range as \(0.1 \leq M^c \leq 1.0\), the coolant reduced frequency \(\Omega^c\) should broadly range as follows

\[
0.02 \leq \Omega^c \leq 1.0
\]  

(9.6)

This eventually leads to have a scaling of the frequencies that are relevant to turbine flows. It is interesting to note that the coolant frequency can be about 1.0 in some situations, so that the quasi-steady assumption is not valid anymore. But this case occurs only if the coolant Mach number is very low. This type of situation may especially happen in the first half of the pressure side, see Fig. 9.1. A very low mean coolant Mach number could reflect a very low blowing ratio, so that the flow inside the hole might already be reversed.

**Range of pulsating amplitude**

The amplitude of the large scale pressure fluctuation \(\bar{P}_s\) is driving the magnitude of the coolant jet blowing. As already introduced by Abhari and Epstein [1], the normalized magnitude of large pressure fluctuation coefficient \(\Psi\) is used

\[
\Psi = \frac{\bar{P}_{s,\text{max}} - \bar{P}_{s,\text{min}}}{P^c_T - \bar{P}_s}
\]  

(9.7)

\(^2\)If the frequency of \(\bar{P}_s\) is too high, the impact of the variation of the near-hole static pressure may experience an unsteady damping

\(^3\)Pressure waves are radiating at the speed of sound \(\bar{a}\) in compressible flows

\(^4\)They give a value of 0.4, based on the shedding frequency \(\omega_j = 2\pi F^j\)
This fluctuation amplitude coefficient $\Psi$ can dramatically vary as a function of where it is observed at the blade surface. Abhari [2] has calculated that, in a standard industrial gas turbine rotor blade\footnote{He presented his calculation at a $-10^\circ$ incidence angle condition. However, except on the first quarter of the pressure side, the distribution of coolant blowing is found the same at design condition.}, the coefficient $\Psi$ can vary from 0.0 up to 3.0 around the blade surface. In particular, the first half of the suction side surface is where the coefficient $\Psi$ is the highest ($\Psi_{max} \simeq 3.0$) whereas on the second half of the suction side it is negligible ($\Psi < 0.2$): the coolant jet does not pulsate. Over the first 85\% of the pressure side surface, the coefficient $\Psi$ is well above 0.2 (threshold value) but not higher than 2.0. However, the instantaneous value of $\dot{P}_s$ can be higher than the total pressure in the plenum chamber ($\dot{P}_T < \dot{P}_s$). In the last 15\% part of the pressure side surface, the coefficient $\Psi$ is negligible.

9.1.3 Goals of the study

This study essentially focuses on demonstrating that the CFD-embedded film cooling jet model can be used in an unsteady flow environment. The following steps are therefore discussed in this Chapter.

1. Treatment of the boundary conditions to be imposed to the film-cooling jet model when experiencing a quasi-steady pulsation.
2. Computational stability and overhead when using the model for unsteady flow predictions.
3. Verification of flow solutions obtained with the film-cooling model when simulating a quasi-steady pulsating jet.
4. Perspective for further developments.

9.2 Making use of the model for unsteady flows

9.2.1 Instantaneous blowing ratio model

\textbf{Steady and unsteady inputs}

In referring to Fig. 3.19, one can list the flow quantities that need to be input to the film-cooling jet model, that is total coolant fluid condition, incoming freestream flow values, as well as the near-hole static pressure. Since it is assumed that total
9.2. MAKING USE OF THE MODEL FOR UNSTEADY FLOWS

Coolant fluid properties are constant, there are only the near-hole static pressure and freestream, flow properties that can vary in time. The large scale near-hole pressure fluctuation \( \bar{P}_s \) is given by

\[
\bar{P}_s = \bar{P}_s + \bar{P}'_s
\]  

(9.8)

which causes the coolant and freestream flow to fluctuate.

\[
\begin{align*}
\bar{U}^c &= \bar{U}^c + \bar{U}'^c \\
\bar{U}^f &= \bar{U}^f + \bar{U}'^f
\end{align*}
\]  

(9.9)

Thus, a large scale fluctuating blowing ratio \( \bar{BR} \) can be defined.

\[
\bar{BR} = \frac{\bar{\rho}^c\bar{U}^c}{\bar{\rho}^f\bar{U}^f}
\]  

(9.10)

From a designer perspective, it is very unlikely that the instantaneous blowing ratio \( \bar{BR} \) is known; in general, a bulk blowing ratio based on the stagnation properties of the upstream main flow, or on a no-film-cooling flow, is utilized. However, the behavior of a film-cooling jet is ultimately driven by the near-hole surrounding flow field. Thus, the film cooling jet model has to be used in connection to the blowing ratio \( \bar{BR} \) based on near-hole flow values\(^6\).

**Instantaneous blowing ratio \( \bar{BR} \) model**

In order to focus on this issue, it is proposed to derive a simple one-dimensional model of the fluctuation of the blowing ratio \( \bar{BR} \) as a function of the large scale variation of the static pressure field \( \bar{P}_s \). It is assumed that the static pressure \( \bar{P}_s \) is directly modifying the freestream and coolant velocity field (\( \Omega^a \ll 1.0 \)). A quasi-steady assumption is made for both the freestream and coolant flow (\( \Omega^f \ll 1.0 \) and \( \Omega^c \ll 1.0 \)). The fluctuating coolant and freestream velocities and densities are defined as follows.

\[
\begin{align*}
\bar{U}^c &= \bar{C}_d\bar{M}^c\bar{a}^c \\
\bar{U}^f &= \bar{M}^f\bar{a}^f \\
\bar{\rho}^c &= \bar{P}^c_T \left( \frac{T^c}{T^T} \right)^{\gamma - 1} \\
\bar{\rho}^f &= \bar{P}^f_T \left( \frac{T^f}{T^T} \right)^{\gamma - 1}
\end{align*}
\]  

(9.11)

\(^6\)This is not exclusive. For steady injection, the inputs can be provided as experimentalists and/or designers do.
9.2. MAKING USE OF THE MODEL FOR UNSTEADY FLOWS

where \( \tilde{C}_d \) is the fluctuating discharge coefficient. Indeed, it has been shown by Bell et al. [10] that the hole discharge coefficient varies as a function of the frequency of the static pressure fluctuations. In this study, the discharge coefficient \( \tilde{C}_d \) is the same as for steady injection since no measurement of the behavior of \( \tilde{C}_d \) is currently available for the flow regimes investigated. Putting Eqs. 9.11 in Eq. 9.12 and using isentropic relation, one gets

\[
\bar{B}R_{1DM} = \tilde{C}_d \left( \frac{T_f^c}{T_f} \right)^{\frac{1}{2}} \left( \frac{P_c^o}{P_f^o} \right)^{\frac{\gamma+1}{2\gamma}} \left[ \left( \frac{P_f^o}{P_s^o} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right] \left( \frac{P_f^o}{P_s^o} \right)^{\frac{\gamma-1}{\gamma}} - 1 \]

(9.12)

To illustrate this model, a typical variation of blowing ratio \( \bar{BR} \) as a function of the near-hole static pressure \( P_s \) is plotted in Fig. 9.2. In Fig. 9.2, the mean blowing ratio \( BR \) is given by the total flow conditions of the coolant and freestream, as well as the mean near-hole static pressure \( P_s \). A different behavior between a low and high mean blowing ratio case is noticed. In the case of a low mean blowing ratio \( (\bar{BR} = 0.5) \), an increase of the near-hole static pressure leads to a decrease

![Figure 9.2: Theoretical variation of the blowing ratio \( \bar{BR} \) as a function of the near-hole static pressure \( P_s \), for quasi-steady freestream and coolant flow condition. It is given in terms of non-dimensionalized blowing ratio \( \bar{BR}^* = (\bar{BR} - BR) / BR \) and non-dimensionalized static pressure \( P_s^* = (P_s - P_s) / P_s \).](image)
of the blowing ratio, as logically expected. However, in the case of a high mean blowing ratio \( BR = 2.0 \), an increase of the near-hole static pressure theoretically leads to an increase of the blowing ratio! Even if this model idealize the real flow phenomenon, the quasi-steady assumption used are valid in some of the computations presented later in this Chapter. This means that a way of verifying the correct probing of the surrounding near-hole flow values (used as inputs to the film-cooling jet model\(^7\)) shall be to match the instantaneous blowing ratio model presented herein.

### 9.2.2 Computational issues

**Definition of the unsteady flows studied**

The film-cooled flat plate test case of Bernsdorf [13] is selected for setting the film-cooling configuration to be investigated. The coolant injection case S4 (see Table 5.3), that is \( DR = 1.26 \) and \( BR = 2.0 \) sets the reference total freestream and coolant flow conditions. In this context, the computational mesh C2, defined in Chapter 6 and largely utilized in the validation of the CFD-embedded film-cooling jet model, is used. Since the pulsating jet flow is studied in a flat plate configuration, the fluctuation of the near-hole static pressure \( P_s' \) needs to be artificially introduced in the calculation. To this effect, the outlet static pressure\(^8\) is specified through a sinusoidal signal in time \( t \).

\[
\tilde{P}_s^{out} = \bar{P}_s + \Delta P_s \sin (2\pi F_3 t + \phi_p) \tag{9.13}
\]

where \( \Delta P_s \) is the maximum pressure amplitude of the fluctuation of \( \bar{P}_s \), relatively to the mean static pressure \( \bar{P}_s \). The phase shift \( \phi_p \) is set to 0 by default. The large-scale static pressure fluctuation is therefore restricted to one harmonic of frequency \( F_3 \), which radiates from the outlet to the inlet of the computational domain.

**Test cases investigated**

Apart from the reference test case S4, another blowing ratio is studied for comparison with the blowing ratio model given in Eq. 9.12. All the investigated test cases are listed in Table 9.2. Note that all test cases are coupled with a computation without any static pressure fluctuation \( (\Delta P_s = 0.0) \) for a better investigation.

\(^7\)That is incoming freestream density \( \rho^f \), velocity \( u^f \) and temperature \( T_T^f \), as well as near hole static pressure \( P_s \).

\(^8\)This is the outlet boundary condition to be specified for a subsonic flow, see Chapter 2.
of the effect of the jet pulsation. Due to the relaxation of the computed flow field

<table>
<thead>
<tr>
<th>Case</th>
<th>$F_j$ [Hz]</th>
<th>$BR$ [-]</th>
<th>$\Omega^f$ [-]</th>
<th>$\Omega^c$ [-]</th>
<th>$\Omega^a$ [-]</th>
<th>$\Psi$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>U1</td>
<td>250</td>
<td>2.13</td>
<td>0.52</td>
<td>0.02</td>
<td>0.01</td>
<td>0.21</td>
</tr>
<tr>
<td>U2</td>
<td>250</td>
<td>0.50</td>
<td>0.52</td>
<td>0.02</td>
<td>0.01</td>
<td>0.98</td>
</tr>
<tr>
<td>U3</td>
<td>1000</td>
<td>2.13</td>
<td>2.07</td>
<td>0.09</td>
<td>0.05</td>
<td>0.47</td>
</tr>
</tbody>
</table>

Table 9.2: Flow regimes investigated for the quasi-steady pulsating jet problem.

in the unsteady mode, the mean blowing ratio $BR$ is found a bit higher in the $BR = 2.13$ then what is a priori guess ($BR = 2.13$). The test cases U1 and U2 lead to a quasi-steady flow regime for both the incoming freestream boundary layer and the coolant. In the higher frequency case U3, a non-quasi steady behavior of the freestream flow in encountered ($\Omega^f = 2.07$). With this test case, it is aimed to analyze the effect of the freestream unsteadiness upon the model behavior. The fluctuation amplitude coefficient $\Psi$ as given in Table 9.2 is calculated after that the calculation has been carried out. The computed peaks of static pressure near the hole appear to be not exactly equal to the ones specified at the outlet of the domain ($\pm \Delta P_s$). This is due to the finite geometry of the domain and the relative flow damping due to the unsteadiness of the flow. The fluctuation amplitude coefficient $\Psi$ is noticed to be lower than what is usually experienced in film-cooled turbines. The reason of this discrepancy is coming from the fact that, due to the rectilinear channel geometry and mean freestream and/or coolant velocity magnitude, the fluctuation of the static pressure has to stay in a thin range in order not to have flow reversal in the hole and/or at the inlet of the computational domain. This type of flow behavior is not supported by the CFD code and the model used in the present thesis.

Total resolution time and overhead

The computation of the pulsating coolant jet is carried out for all the test cases presented in Table 9.2 without any stability problem. All computations are always started with the same initial flow solution, that is the flow through the channel without any film-cooling jet. The computational procedure necessitates therefore to calculate first the transient regime and then to entirely calculate at least one period of the periodic regime. To this effect, an a priori analysis of the convective time scale $\tau^f$ of the flow through the whole computational domain is carried out.
This is done relatively to the pulsating frequency $F_j$. In this context, a minimal number of period $N_p^{\text{min}}$ to be computed is defined to get a periodic solution over the last computed period.

$$N_p^{\text{min}} = \text{int} \left( \frac{\tau^I}{\tau_j} \right) + 1 = \text{int} \left( \frac{L_{fp}/u^I}{1/F_j} \right) + 1$$ (9.14)

Computing at least $N_p^{\text{min}}$ periods should ensure that when the calculation reaches the iteration number (physical time) where the numerical solution begins to be stored, the startup flow solution is totally evacuated. In reality, it is obvious that many flow time scales are existing. However, it has been checked with the flow case $U3$ that this definition brings effectively a good scaling of the number of period needed to reach the periodic flow regime. The number of periods $N_p$ computed for the two frequencies investigated are given in Table 9.3. The total number of iterations $N_I$ are also given, as well as the total resolution time $\tau_{\text{total}}$.

<table>
<thead>
<tr>
<th>$F_j$ [Hz]</th>
<th>$N_p^{\text{min}}$ [-]</th>
<th>$N_p$ [-]</th>
<th>$N_I$ [-]</th>
<th>$\tau_{\text{total}}$ [h]</th>
</tr>
</thead>
<tbody>
<tr>
<td>250</td>
<td>2</td>
<td>2</td>
<td>358'398</td>
<td>118</td>
</tr>
<tr>
<td>1000</td>
<td>3</td>
<td>8</td>
<td>358'398</td>
<td>118</td>
</tr>
</tbody>
</table>

Table 9.3: Pulsating jet: computational time to reach a periodic solution.

of iterations $N_I$ are also given, as well as the total resolution time $\tau_{\text{total}}$. The time discretization for the high frequency case ($F_j = 1000$ [Hz]) is shown to be coarser than the one of the low frequency case ($F_j = 250$ [Hz]) meanwhile satisfying the stability criterion. In order to demonstrate that the CFD-embedded film-cooling jet model can also be used for predicting unsteady film-cooling jet flows in a relevant time scale, the overhead $\sigma$, as defined in Eq. 6.2 in Chapter 6 is evaluated for the unsteady flow cases simulated in this Chapter. It is tabulated in Table 9.4.

<table>
<thead>
<tr>
<th>Case</th>
<th>$N_u$ [-]</th>
<th>$\sigma$ [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>U1</td>
<td>17'920</td>
<td>4.13</td>
</tr>
<tr>
<td>U2</td>
<td>17'920</td>
<td>4.38</td>
</tr>
<tr>
<td>U3</td>
<td>2'240</td>
<td>4.52</td>
</tr>
</tbody>
</table>

Table 9.4: Overhead obtained for the computed pulsating jet cases.

In this Table, the number of updates per period $N_u$ made for numerically immersing
the three-dimensional jet surface, due to its movement, is also given. As it can be seen, the number of updates per period is very large, so that the overhead $\sigma$ given in Table 9.4 envelops the overhead that should occur for this kind of computation. One can see that the overhead $\sigma$ is about 4% whatever the blowing ratio and frequency are. This is a low overhead since the CFD-embedded film-cooling jet model is used here in an unsteady flow environment. This means that all the arrays that are needed to describe the location of the three-dimensional toroidal jet surface can be very quickly updated when new inputs are fed to the model. This shows that the modeling approach and the strategy chosen for the immersion of the model in a CFD code is relevant for unsteady CFD prediction in terms of computational cost.

9.3.1 Computational history

**Matching the instantaneous blowing ratio model**

In order to verify that the computational results make sense, it is first proposed to compare the computed evolution of the blowing ratio $BR$ and near-hole static pressure $P_s$ with the instantaneous blowing ratio model $BR_{1,Di}$. The computed blowing ratio $BR$ and near-hole static pressure $P_s$ are found by probing, in a cross plane, the calculated flow field at 1 hole diameter upstream of the hole leading edge and at the wall surface$^9$ respectively, as shown in Fig. 4.11 in Chapter 4. The evolution of the computed blowing ratio $BR$ and near-hole static pressure $P_s$ with the instantaneous blowing ratio model $BR_{1,Di}$ are compared in Fig. 9.3. This is shown for the test cases $U2 (BR = 0.5)$ and $U1 (BR = 2.13)$. First of all, in the test case $U2$, a very good agreement of the time evolution of the computed and modeled blowing ratios is found. A low near-hole static pressure $P_s$ leads to a high instantaneous blowing ratio $BR$ and vice-versa, as expected. This result indicates that the quasi-steady assumptions made for the film-cooling jet model are valid, at least in this case. Most importantly, this shows that the behavior of the CFD-embedded film-cooling jet model in an unsteady flow environment tends to make sense. However, when comparing the time history of the fluctuation of the computed blowing ratio $BR$ to the blowing ratio model $BR_{1,Di}$ for the test case $U1$, a noticeable discrepancy arises. A phase shift of $\Delta t / \tau_j = 0.2$ between the computed and modeled blowing ratio is observed, although the amplitude of

---

$^9$ where the area of probing is selected as $-1.5 < X/d < 1.5$ and $-1.0 < Y/d < 1.0$
fluctuation is found very similar in both cases. It is to be underlined that the phase shift is much less than $\Delta t/\tau_j = 0.5$ which therefore confirms that, for a high mean blowing ratio ($\overline{BR} = 2.13$), the time evolution of the blowing ratio $\overline{BR}$ is not similar as for a low mean blowing ratio case ($\overline{BR} = 0.5$); The minimum value of near-hole static pressure $\overline{P}_s$ does not correspond to the maximum value of the blowing ratio $\overline{BR}$ and vice-versa. In addition, a mode of oscillation of $\overline{P}_s$ and $\overline{BR}$ at low amplitude and high frequency is noticed. This comes from a numerical artifact which is created by the use of a central collocated numerical scheme, leading easily to odd-even wiggles.

**Inputs to the model - the phase shift issue**

In order to better understand and explain the phase shift observed for the test case $U1$, Fig. 9.4 plots the time evolution of the freestream averaged momentum (probed in the computational mesh) and coolant momentum (derived from the expansion model). The time evolution of the near-hole static pressure $\overline{P}_s$ is superimposed. Indeed, the instantaneous blowing ratio $\overline{BR}$ is a direct function of these two quantities ($\overline{\rho f} \overline{U_f}$ and $\overline{\rho c} \overline{U_c}$), which are themselves function of the near-hole static pressure $\overline{P}_s$. In the case $U2$ ($\overline{BR} = 0.5$), it is noticed that the influence of the freestream momentum is much smaller than the coolant momentum.
9.3. QUASI-STEADY PULSATING JET RESULTS

Figure 9.4: Evolution of the freestream momentum $\dot{\rho}U_f$, coolant momentum $\dot{\rho}U_c$ and near-hole static pressure $P_s$ as a function of time $t/\tau_j$. On the left, test case U2 and on the right, test case U1.

\[
\Delta \left[ \dot{\rho}U_c \right] \approx 13 \Delta \left[ \dot{\rho}U_f \right].
\]

This can be explained by the relatively small magnitude of the static pressure variation, coupled with a low mean blowing ratio. Given that the coolant momentum $\dot{\rho}U_c$ is instantaneously updated as a function of the probed near-hole static pressure $\ddot{P}_s$, it becomes obvious that the computed blowing ratio $BR$ is very similar to the modeled blowing ratio $\overline{BR}$. Now, for the test case U1 ($\overline{BR} = 2.13$), it is observed that the freestream momentum fluctuation $\dot{\rho}U_f$ is of the same order of magnitude than the one of the coolant momentum $\dot{\rho}U_c$, even a bit larger \(
\Delta \left[ \dot{\rho}U_c \right] \approx 0.44 \Delta \left[ \dot{\rho}U_f \right].
\)

As previously, the evolution of the coolant momentum $\dot{\rho}U_c$ is exactly in phase with the fluctuation of near-hole static pressure $\ddot{P}_s$. Meanwhile, a phase shift between the time evolution of the freestream momentum $\dot{\rho}U_f$ and the near-hole static pressure of about $\Delta t/\tau_j \approx 0.2$ is noticed, which is very similar to the one observed for the time evolution of the computed and modeled blowing ratio. Thus, this phase shift indicates that the freestream momentum needs a time $\Delta t_{lag}/\tau_j$ to reach a value at equilibrium condition corresponding to $\ddot{P}_s$ at time $t/\tau_j$. This phase shift may come from several sources. The probing of the incoming freestream flow is done one diameter upstream of the hole leading edge, which corresponds to a very small
time shift of $\Delta t^{1.5d}/\tau_j \simeq 0.03^{10}$ from the center of the hole. Thus, the relaxation of the freestream momentum reached at time $(t + \Delta \tau^{lag}) \tau_j$ to an equilibrium solution based on the near-hole static pressure $P_s$ at time $t/\tau_j$ is the main reason of the phase shift. This induces that an instantaneous adjustment of the coolant momentum as a function of the near-hole static pressure condition, as proposed in the current model, may be erroneous. As a recommendation for further use of the model in unsteady flow environments, it is propose to modify the expansion model of the coolant flow by taking into account a time lag parameter $\tau^{lag}$ so as to have (see Eq. 9.15 in Chapter 3)

$$u^c_X(t) = \frac{4m_{actual}(t - \tau^{lag})}{\pi d^2}$$ (9.15)

This empirical parameter needs further experimental investigations in order to be characterized.

**Blowing history for non-quasi steady freestream flow**

It has been seen that, for quasi-steady condition of the freestream and coolant flow and for one moderate harmonic of pulsation, the model brings a reasonable answer. The test case $U3$ in which the large-scale static pressure fluctuation frequency is increased 4 times more than for the case $U1$ is now analyzed. This corresponds to a non-quasi steady behavior of the freestream flow. Fig. 9.5 plots the time evolution of the computed and modeled blowing ratios, as well as the time evolution of the freestream and coolant momentum. It is first noticed that the time histories of all quantities plotted are smoother than what has been previously observed. This can be explained by the fact that the time discretization is about 8 time coarser in this case than the test cases $U1$ and $U2$. Even though the coarser time discretization tends not to solve as many time scales as previously, the large scale fluctuation are captured, since only one large harmonic of the pressure fluctuation is fed to the computational domain. As observed in Fig. 9.5, the computed evolution of $BR$ does not correspond anymore to the modeled $BR_{1Dm}$, in terms of both amplitude and phase. In fact, looking at the time evolution of the freestream and coolant momentum, it is surprisingly seen that the absolute magnitude of the fluctuation of freestream momentum is much smaller than in the $U1$ case $\Delta [\varrho U^f]_{U3} \simeq 0.5 [\Delta \varrho U^f]_{U1}$, even though the absolute magnitude of

$^{10}$based on a distance of 1.5 hole diameter and $U^f = 100$ [m/s], for a pulsating frequency of $F_j = 250$ [Hz]
9.3. QUASI-STEADY PULSATING JET RESULTS

Figure 9.5: Evolution of the computed $\tilde{BR}$, modeled $\tilde{BR}$ blowing ratios (left), freestream momentum $\tilde{\rho Uf}$, coolant momentum $\tilde{\rho Uc}$ (right) and near-hole static pressure $\tilde{P_s}$ as a function of time $t/\tau_j$.

near-hole static pressure fluctuation is larger! (see Table 9.2). In fact, for a very high value of the freestream reduced frequency $\Omega^f = 2.07$, the freestream flow field does not manage anymore to reach an equilibrium as a function of the near hole static pressure $\tilde{P_s}$. The quasi-steady adjustment of the freestream flow is verified to be not valid anymore. There is an unsteady damping effect of the fluctuation of the freestream. As such, the assumptions used for the CFD-embedded film-cooling jet model are ensured anymore.

9.3.2 Analysis of the computed flow field

**Boundary layer development**

As shown in Table 9.2, the amplitude $\Psi$ of the fluctuation of the near-hole static pressure $\tilde{P_s}$ is not as high as it occurs in a film-cooled turbine. Furthermore, the relatively parallel fluctuation of the freestream and coolant flow has the tendency to damp the variation of the blowing ratio. This induces a relatively low absolute magnitude of the blowing ratio fluctuation, see previous section. The boundary layer development for the three test cases computed, in terms of the streamwise velocity $U/\tilde{U}^f$, are plotted in Figs. 9.8, 9.7 and 9.8. This is shown at different axial locations ($X/d = 2.0, 4.0, 6.0, 14.0$) for the time-averaged result of the no pulsation and pulsation cases, as well as for two different instantaneous cases, that
is at time steps $t/\tau^i = 0.25$ and $t/\tau^i = 0.75$. In all computed test cases, it

is systematically observed that the time-averaged result of the pulsation case is quasi identical to the no pulsation case. It is also noticed that the variation, in time and space, of the streamwise velocity profile around its time-averaged value is small. This corresponds to a small variation of the fluctuating blowing ratio $\overline{BR}$. 

Figure 9.6: Axial evolution of the boundary layer streamwise velocity $\tilde{U}/\tilde{U}^f$ for the test case U2 ($BR = 0.5$).

Figure 9.7: Axial evolution of the boundary layer streamwise velocity $\tilde{U}/\tilde{U}^f$ for the test case U1 ($BR = 2.13$).
9.3. QUASI-STEADY PULSATING JET RESULTS

In addition, in the test case U3, the spatial evolution of the streamwise velocity, for a given time is phase shifting from $X/d = 2.0$ to $X/d = 6.0$. This is induced by the relatively high frequency, compared to the other cases. In a first approach, it is very surprising to see no real differences in the presented results, between the no pulsation and pulsation cases. However, there are a beam of reasons that can objectively explain these results. First of all, as it has been seen, the variations of the blowing ratio are small, so that it is very unlikely that high differences occur between the no pulsation and pulsation results. Secondly, it has experimentally been shown, see Seo et al. [84], that for a quasi-steady flow condition and long holes, the change in time-averaged heat load, compared to a no pulsation flow case, is negligible. Since these conditions are exactly fulfilled in the computations shown in this Chapter, it is very likely that the time-averaged flow profile is very similar to the no pulsation flow case.

Adiabatic film-cooling effectiveness

The computed laterally-averaged adiabatic film-cooling effectiveness $\bar{\eta}$ is plotted in Figs. 9.9 and 9.10 for all the test cases. This is shown for the time-averaged results of the no pulsation and pulsation cases, as well as for two different instantaneous cases, that is at time steps $t/\tau^j = 0.25$ and $t/\tau^j = 0.75$. In connection to the observation done for the streamwise velocity field $U/\bar{U}^j$, the time-averaged laterally-averaged film-cooling effectiveness of the pulsation case
9.3. QUASI-STEADY PULSATING JET RESULTS

Figure 9.9: Laterally-averaged film-cooling effectiveness $\bar{\eta}$ for the test case $U_2$ ($BR = 0.5$).

Figure 9.10: Laterally-averaged film-cooling effectiveness $\bar{\eta}$ for the test case $U_1$ ($F_j = 250$ [Hz], left) and for the test case $U_3$ ($F_j = 1000$ [Hz], right).

is quasi-identical to the no pulsation case. In the $U_2$ case, the temporal variation of $\bar{\eta}$ is very small, which confirms the very low impact of the large-scale static pressure variation as being simulated here. In the $U_1$ and $U_3$ cases, the temporal variation of $\bar{\eta}$ is large relatively to its small mean value. This shows that even if the time-averaged results are not changing compared to the no pulsation case, it is very likely that at some instant, the cooling protection may be enhanced or reduced.
9.4 Conclusion

A comprehensive introduction of the pulsation of coolant jets in turbines has been provided. The quasi-steady assumption used for the film-cooling jet model covers a large (but not the entire) part of coolant jet flow regimes. It has been shown that for a quasi-steady jet, the CFD-embedded film-cooling jet model delivers a satisfactory solution. In particular, the overhead when using the model in such flow is about 4\%, which shows the relevance of the modeling approach for computing in a reasonable time scale. It is proposed to modify the coolant expansion model by including a time lag \( \tau_{\text{lag}} \) parameter. Further experimental investigations are needed to characterize this parameter. Due to the restrictions upon flow conditions imposed by a flat plate configuration, a complete validation of the use of the model still needs to be done. This can only be carried out in a real rotating film-cooling turbine test case. The film-cooling jet model as it is can only be used for a quasi-steady regime of both the freestream and coolant flow. For quasi-steady pulsating jets, issuing from long holes, the time-mean flow field is shown to be similar to the one encountered in steady injection.
Seite Leer / Blank leaf
Chapter 10

Film Cooling Model Use in an Linear Turbine Cascade

10.1 Introduction

10.1.1 Motivation and goals

This Chapter is dedicated to the utilization of the film cooling model in the context of a film-cooled linear turbine cascade. The main question raised in the Introduction (see section 1.4) has partially been covered in the previous sections. A model that allows computing a multi-scale flow problem in a reasonable time scale has been introduced and validated within several flow configurations in a flat plate. Using the model in a realistic turbine configuration is a natural step toward answering the question underlying this thesis. In particular, the model should allow to predict, in a reasonable time scale, blade loading, film-cooling effectiveness and heat transfer rate at blade surface, as well as losses occurring through the blade passage. These are the key physical values setting the performance of a film-cooled turbine. By showing the relevancy of the use of the model in such representative turbine blade configuration, it is aimed to make a step toward the development of better optimization tools dedicated to the design of film-cooled turbine. In this context, it is eventually intended to show that the model can also predict the aerothermal field resulting from multiple jets interactions, typical of the flow structure of an energy conversion engine.

Main steps of the Chapter

The film-cooled turbine blade investigated in this Chapter is presented. To preserve the computational accuracy, a special mesh clustering technique to be used in the near hole region is discussed. The computational performance of the model in such a flow configuration is evaluated, in terms of convergence history, mass
flow error and computational overhead. The CFD predictions are compared to experimental data, in terms of blade loading, film-cooling effectiveness and Nusselt number. An evaluation of the losses is given. This is done for two blowing ratios and 9 different hole configurations. Eventually, an investigation of the effect of the superposition of different cooling rows on the pressure side upon film-cooling protection is carried out.

10.1.2 Description of the transonic turbine linear cascade

Geometry and freestream flow condition

The Oxford University linear transonic turbine cascade test case is briefly described. The turbine blades are placed in an isentropic light piston cascade, extensively described by Schultz et al. [82]. This is a short duration facility ($\Delta t \simeq 500$ [ms]) which produces uniform (steady) flow conditions for about 300 [ms]. The test gas is compressed by a single-stroke isentropic compressor. A turbulence grid generator is placed at five blade chords upstream of the blades row and produces a freestream turbulence level of about 3%. Just upstream of the blade row, a rotating bars device can be placed in order to simulate shock waves and wakes blade interaction. This option is not considered in this study; only a steady flow through the blade passage is investigated. The linear turbine blades are a two-dimensional representation, at midspan, of typical gas turbine rotor blades. The dimensions of the blade and the passage are given in Fig. 10.1 and Table 10.1.

<table>
<thead>
<tr>
<th>Geometrical dimension</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blade true chord $C_{TC}$ [mm]</td>
<td>41.0</td>
</tr>
<tr>
<td>Blade axial chord $C_{AC}$ [mm]</td>
<td>34.82</td>
</tr>
<tr>
<td>Blade to blade pitch $P_{BB}$ [mm]</td>
<td>34.3</td>
</tr>
<tr>
<td>Inlet metal angle $\theta_{int}$ [°]</td>
<td>48</td>
</tr>
<tr>
<td>Outlet metal angle $\theta_{int}$ [°]</td>
<td>77</td>
</tr>
<tr>
<td>Inlet hub to tip span $H_{HT}^{in}$ [mm]</td>
<td>50.0</td>
</tr>
<tr>
<td>Outlet hub to tip span $H_{HT}^{out}$ [mm]</td>
<td>56.095</td>
</tr>
</tbody>
</table>

Table 10.1: Geometrical dimensions of the linear turbine cascade of Ashworth [6].

The geometry of the blade profile is provided by Ashworth [6]. As the blades are not leaned, the blade profile at hub section is orthogonally stacked up to the casing
10.1. INTRODUCTION

Figure 10.1: Sketch of the linear turbine cascade geometry of Ashworth [6].

Wall (tip of the blade). There is no tip clearance nor shroud as the blades are not moving. The hub to tip height $H_{HT}$ increases from the leading edge to the trailing edge of the blade. Unfortunately, the profile of the endwall section is not known, so that a linear increase, through the axial direction, of the hub to tip height ($H_{HT}$) is assumed. A transonic flow regime is investigated, based on the experimental data provided by Rigby et al. [74] and is listed in Table 10.3.

<table>
<thead>
<tr>
<th>Flow quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Mach number $M_{in}$ [-]</td>
<td>0.38</td>
</tr>
<tr>
<td>Outlet Mach number $M_{out}$ [-]</td>
<td>1.18</td>
</tr>
<tr>
<td>Reynolds number $Re_{out}$, based on blade chord and outlet isentropic condition [-]</td>
<td>$1.0 \cdot 10^6$</td>
</tr>
<tr>
<td>Inlet total temperature $T_{in}$ [K]</td>
<td>460</td>
</tr>
<tr>
<td>Inlet total pressure $P_{in}$ [bar]</td>
<td>2.9</td>
</tr>
</tbody>
</table>

Table 10.2: Experimental flow condition of the linear turbine cascade of Rigby et al. [74].
Film-cooling holes configuration

There are five rows of film-cooling holes drilled through the blade material, as shown in Fig. 10.2. There are two rows on the suction side (set 1 and set 2) and three rows on the pressure side (set 3) among which two are staggered (set 4a and set 4b). Except for set 1, every cooling hole has a cylindrical body from the plenum chamber to the blade surface. The hole diameter is \( d = 0.5 \) [mm]. The cooling row in the first half of the suction side (set 1) contains only shaped-holes. This means that the circular cross section of the holes at the plenum outer surface is expanded up to the blade surface, resulting in a trapezoidal cross sectional shape at hole exit. The width of the hole exit area is larger than for cylindrical holes. Except for set 3, every cooling hole has a streamwise injection angle of \( \alpha_0 = 30^\circ \). The cooling row in the first half of the pressure side (set 3) has a streamwise injection angle of \( \alpha_0 = 60^\circ \). In fact, the orientation of the holes at this location is typical. Due to concave shape of the pressure side, it is more difficult to drill holes with a low streamwise injection angle. Rigby et al. [74] do not report the number of holes per row in their experimental setup. In order to demonstrate the capability of the film-cooling model to simulate coolant jets issuing simultaneously from several dozens of holes, 17 holes per row (16 holes for set 4b) are placed. The hole to hole pitch distance is constant for every row and equal to \( s/d = 4.0 \). This study limits the
validation of the use of the CFD-embedded film cooling jet model near the midspan section only. Thus, there is no hole in the very near proximity of endwall corners. As previously described, the first row in the suction side (set 1) is a collection of shaped-holes. Since this hole configuration is not available in the present film cooling jet model, it is replaced by a cylindrical hole configuration. Furthermore, there is currently no calibration of the model coefficients for a streamwise injection angle of $\alpha_0 = 60^\circ$ (holes in set 3). To solve this issue, the model coefficients are linearly extrapolated from their known values at $\alpha_0 = [30^\circ, 50^\circ]$.

Coolant flow conditions

The coolant flow properties are not all explicitly provided. The freestream to coolant total temperature ratio is given \( \frac{T_{f0}}{T_{f}} = 1.6 \) which results in a coolant total temperature in the plenum of \( T_{f}^c = 287.5 \) [K]. The coolant total pressure \( P_{f}^c \) is not explicitly known. Instead, several blowing ratios BR are given, but no formal indication on how they are calculated is provided. Thus, the definition of blowing ratio employed by many experimentalists is used. In a first step, an uncooled blade computation is carried out in order to determine the near-hole static pressure \( P_s \). Based on the total freestream flow conditions, near-hole static pressure, blowing ratio and total temperature of the coolant, the total pressure \( P_{f}^c \) of the coolant is computed using Eq. 9.12. As it can be seen in Fig. 10.2, there are two plenum chambers, one for set 1 and set 3, the other one for set 2 and set 4ab. In addition, the experiment has been carried out separately for the pressure and suction side. Using these facts, it is decided that the total pressure \( P_{f}^c \) of the coolant could be different for each rows (except for set 4a and set 4b) so that the blowing ratio is matched for each of them. All the different coolant conditions investigated in this study are listed in Table 10.3. The two reference flow regimes

<table>
<thead>
<tr>
<th></th>
<th>BR = 1.0</th>
<th></th>
<th>BR = 1.5</th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>set 1</td>
<td>set 3</td>
<td>set 4ab</td>
<td>set 1</td>
<td>set 3</td>
<td>set 4ab</td>
</tr>
<tr>
<td>Total temper-</td>
<td>287.5</td>
<td>287.5</td>
<td>287.5</td>
<td>287.5</td>
<td>287.5</td>
<td>287.5</td>
</tr>
<tr>
<td>ature ( T_{f}^c ) [K]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Total pressure</td>
<td>2.632</td>
<td>2.841</td>
<td>2.803</td>
<td>3.144</td>
<td>2.961</td>
<td>2.997</td>
</tr>
<tr>
<td>( P_{f}^c ) [K]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 10.3: Coolant flow conditions.
10.1. INTRODUCTION

concern the uncooled blade with adiabatic or isothermal wall. For the isothermal wall computation, the wall temperature $T_w$ is set to $T_w = 320$ [K], approximatively as in the experiment (experimental uncertainty in fixing $T_w$ is $\pm 6$ [K]). The different flow regimes with cooling are coolant injection with the two blowing ratios ($BR = 1.0, 1.5$) combined with different arrangement of rows together ($set 1$, $set 3$ $set 4a$ and $set 4ab$). These different flow regimes are computed twice, for an adiabatic and an isothermal wall boundary condition. It is worth mentioning that $set 2$ is not considered since it is located in a transonic flow region, in which the current model is not available.

Available experimental data

The blade surface pressure $P_s$ and blade surface heat transfer rate $q'_w$ are measured. The heat transfer rate $q'_w$ employs

$$q'_w = -\kappa \left[ \frac{\partial T}{\partial n} \right]_{wall} \tag{10.1}$$

where $n$ is the direction normal to the blade surface. To acquire the surface pressure, pressure taps are placed at several locations on the blade surface, leading to have 10 and 13 measurement points at the pressure and suction side respectively. The blade loading is given in term of the isentropic Mach number $M_{is}$, which reads

$$M_{is} = \sqrt{\frac{2}{\gamma - 1} \left[ \left( \frac{P_T}{P_s} \right)^{\frac{\gamma - 1}{\gamma}} - 1 \right]} \tag{10.2}$$

The surface heat transfer rate $q'_w$ is obtained from surface temperature $T_s$ measurements using high frequency analogues, see Oldfield et al. [72]. The surface heat transfer is given in term of the Nusselt number $Nu$.

$$Nu = \frac{q'_w C_{TC}}{\kappa (T_T - T_s)} \tag{10.3}$$

The film cooling effectiveness $\eta$ is also provided in term of the isothermal adiabatic wall film cooling effectiveness. For more information about the latter, see Rigby et al. [74]. The measurements uncertainties are not reported by Rigby et al. [74].
10.2 Computational issues

10.2.1 Computational mesh for an efficient use of the film-cooling model

**Computational domain**

The computational domain entirely covers the hub to tip span, through the whole blade passage. The inlet plane is placed at a distance of 0.27 axial chord upstream of the leading edge of the blade. Despite the relatively short distance between the inlet plane and the blade, the use of the non-reflecting boundary conditions, as described in Chapter 2, ensures an adequate treatment of the flow characteristics going in and out of the inlet plane. In addition, the short interval between the inlet plane and the leading edge of the blade is typical of a multiple blade rows configuration. The outlet plane is located at a distance of 0.5 axial chord downstream of the trailing edge of the blade. The meshing of the computational domain, displayed in Fig. 10.3, is carried out by using an in-house grid generator developed at ETHZ [65]. Briefly, this grid generator is based upon a multi-block topology.

![Figure 10.3: Computational domain mesh, in the XY-plane at hub (2D grid - left) and in the XZ-plane at the pressure side surface and periodicity plane (right).](image)

Several H-blocks are connected together in the passage from the inlet to the outlet plane. Around the blade surface, an O-block is placed. This allows to arrange and to cluster the grid cells that are located in the boundary layer region without signi-
10.2. COMPUTATIONAL ISSUES

Significantly influencing the grid in the through-flow passage. In particular, it is ensured that the first cells close to the blade walls are strictly hexahedral so that their edges are orthogonal to the blade surface\(^1\). The strategy to get the three-dimensional computational mesh is to pile up two-dimensional blade to blade grids in the spanwise direction, as shown in Fig. 10.3. The 2D grid in Fig. 10.3-left exhibits a refinement near the throat area. This is because a shock is assumed at this location. After the throat, the expansion of the grid cells area does not significantly influence the solution accuracy [65]. In total, the computational mesh contains \(N_{3D} = 2.13 \cdot 10^6\) grid nodes. This is a relatively large grid to those generally employed in RANS-based numerical predictions of turbomachinery flows. A typical three-dimensional grid to predict the flow through an uncooled blade passage (not tip clearance nor shroud) has an order of magnitude of \(5 \cdot 10^5\) grid nodes. There are several reasons that lead to this grid size. Firstly, as a two-layer turbulence model is used (Baldwin-Lomax), it is needed to cluster the first cell near the blade wall so as to ensure a \(Z^+\) value of \(Z^+ \simeq 1\). This leads to place a relatively large number of cells nearby the blade wall. In the current grid, there are approximately 12 to 20 grid nodes in the boundary layer region. Secondly, the use of the CFD-embedded film cooling jet model necessitates to have enough mesh resolution near the cooling holes in order to preserve an acceptable accuracy.

**Special clustering for the computational mesh**

As it has been analyzed and demonstrated in Chapter 6, it is advised to have about \(N_X \simeq 4 - 7\) and \(N_Y \simeq 7 - 11\) near the cooling holes to ensure an acceptable accuracy without excessively increasing the computational time. This means that the mesh density at blade surface needs somehow to be controlled. To this effect, a special mesh clustering technique has been implemented in the current grid generator [65]. This mesh adaption strategy can be described as follows. Given an orthogonal grid at the blade surface, as shown in Fig. 10.4, it is wished to stretch and tilt the mesh lines so that the resulting mesh is clustered near the location of the cooling holes. In order to carry out this procedure, the one-dimensional adaptive grid algorithm proposed by Gnoffo [36] is utilized for the current mesh adaption problem. The algorithm is upgraded for two-dimensional problem by simply solving the problem twice, in the two directions describing the blade surface. The basic idea of the algorithm is to link every grid node with a constant spring element. The value of the spring constant is a function of the mesh density wanted at

---

\(^1\)This orthogonal mesh arrangement near the wall allows to lower the numerical production of entropy, especially near the leading edge of the blade
the considered location. Once all spring elements are defined, a Newton-Raphson procedure is employed to find the equilibrium solution of the system of springs. The equilibrium solution gives the final clustered mesh, as shown in Fig. 10.4. There are $N_x = 5$ and $N_y = 7$ grid nodes per hole diameter near every cooling hole of the computed blade. The fact that $N_y = 7$ leads to have in total $279$ 2D grids to be piled up. This induces an overall grid size of $N_{3D} = 2.13 \cdot 10^6$. Due to computer memory limitation at the time of this work, this is the largest grid size that could be utilized. This special mesh clustering technique shows that the current numerically immersed film-cooling jet model needs to be used with care but allows to compute a full film-cooled turbine blade passage with a reasonable number of grid nodes. As a comparison, in terms of overall grid size, the film-cooled turbine blade computed by Heidmann et al. [42] (see also Fig. 1.5), where the holes and plenum chamber are gridded, contains about $1.2 \cdot 10^6$ grid nodes with only 2 holes per row (only the midspan section is modeled). An extrapolation of this grid to the film-cooled turbine blade computed in this work leads to a mesh having more than 25 to $40 \cdot 10^6$ grid nodes. Based on this benchmark, it is postulated that, the current jet model allows decreasing by a factor of over 10 times the size of the mesh to be used to compute the flow in a film-cooled turbine. In addition, the pre-processing time spent to build up the entire film-cooled blade
10.2. COMPUTATIONAL ISSUES

geometry and grid is significantly reduced because only the blade through-flow region is modeled.

10.2.2 Computational performance

Computational history

The computation of the different flow regimes, using the model, shows a stable behavior. The computational procedure employed is the following. A generic flow solution is first computed with the blade uncooled and having an adiabatic or isothermal wall boundary condition. The numerical solutions of the different flow regimes through the film-cooled blade are obtained by starting the computation with the generic flow solution. Two typical convergence histories, in terms of root-mean square residual $DQ_{rms}$, are plotted in Fig. 10.5. The convergence

![Figure 10.5: Computational history (Root-Mean Square residuals ($DQ_{rms}$)) of the film-cooled linear turbine cascade (adiabatic wall) with set 1, set 3, set 4ab and $BR = 1.0$.](image)

histories concern the uncooled blade and the film-cooled blade with rows set 1, set 3, set 4ab and a blowing ratio of $BR = 1.0$ (adiabatic wall). The other computations get similar convergence histories. The root-mean square residual $DQ_{rms}$ drops by about 1 order of magnitude for both restarted computations. Interestingly, the iterative evolution of the root-mean square residual $DQ_{rms}$ is very similar for both computations, although there is a shift of about $\Delta DQ_{rms} \approx 0.3$ between the uncooled and film-cooled blade calculations, the latter being higher. This can be explained by the fact that the flow near the toroidal jet body readjusts so that a higher residual is found locally. This process is similarly found near the leading
edge or trailing edge of the blade. At this stage, it is worth noting that the total coolant mass flow injected varies from 0.15% to 1.35% of the total freestream mass flow entering the turbine cascade, depending on the coolant injection configuration, see Appendix L.1. Interestingly, the reduction of freestream mass flow due to the blockage of the coolant jets is found to be counter-balanced by the augmentation of the coolant mass flow so that the outlet mass flow remains constant \( \dot{m}_{\text{out}}^f = 0.39 \text{ [kg/s]} \) for one passage. The global mass flow error \( \epsilon_{\text{m}}^G \) ranges from \(-0.26\%\) to \(0.22\%\), see Appendix L.2. In summary, the computational histories for the uncooled and film-cooled blade computations are similar. This shows that the numerical inclusion of the film-cooling jet model does not significantly degrade the convergence rate and mass flow errors. In other words, the number of iterations (10'000 in this case) to find a converged flow solution in a film-cooled blade cascade is of the same order as for an uncooled blade calculation, using the same grid.

**Overhead**

The computational overhead \( \sigma \), as defined in Eq. 6.2 (see Chapter 6), is plotted in Fig. 10.6 as a function of the number of holes \( N_h \). As it can be observed,

![Figure 10.6: Overhead \( \sigma \) as a function of the number of holes \( N_h \) found in the linear film-cooled turbine cascade test case.](image)

\[ \sigma \approx 0.1 \cdot N_h \]  \hspace{1cm} (10.4)
10.3. UNCOOLED BLADE: VALIDATION OF THE COMPUTATION

This means that including one more hole within the blade surface increases the computational time by 0.1%. The mesh employed in this film-cooled blade flow prediction has a typical size for the utilization of the CFD-embedded film cooling jet model. Thus, Eq. 10.4 gives a relevant scaling of the computational cost when using the model in a film-cooled turbine cascade. As the number of iteration needed to get a converged solution is similar in the uncooled and film-cooled blade computations, Eq. 10.4 scales the total amount of extra computational time when calculating a film-cooled turbine, compared to an uncooled blade computation with the same grid. However, as previously mentioned, a typical grid size for an uncooled blade computation contains approximately 4 times less grid nodes. Hence, the real computational cost when using the model is more than 4 times the uncooled blade computational cost.

10.3 Uncooled blade: validation of the computation

In a first step, some key results of the uncooled blade prediction are presented. This is to give to the reader an overview of the flow problem studied and also to validate the computational procedure.

10.3.1 Overall flow field

Midspan flow field - static pressure

The predicted contours of static pressure coefficient $C_{ps}$ at midspan are plotted in Fig. 10.7. The contours of static pressure coefficient $C_{ps}$ exhibit the transonic character of the computed flow field. A shock system occurs near the throat area and blade trailing edge. One branch of the shock system is going from the trailing edge of the blade toward the rear part of the suction side. This provokes a shock-boundary layer interaction at the suction side surface. Meanwhile, no boundary layer separation is identified downstream of the shock. Near the surface of the blade, the static pressure reaches a peak minima just before the shock. Just after the shock, the static pressure suddenly increases. In the freestream, just after the shock, the throat is behind. Thus, a region of relatively low static pressure persists which induces a supersonic zone. The other branch of the shock system is going from the trailing edge of the blade toward the outlet of the computational domain. Schlieren photographs done by Rigby et al. [74] confirm the presence of this shock system.
10.3. UNCOOLED BLADE: VALIDATION OF THE COMPUTATION

Passage flow field - Nusselt number

The predicted contours of Nusselt number $Nu$ at the blade surface, superimposed with the surface flow streamlines, are plotted in Fig. 10.8. The Nusselt number reaches its maximum level at the leading edge of the blade, from hub to tip. Indeed, this is where the boundary layer is the thinnest. On the pressure surface, the Nusselt number is pretty constant along blade axial direction ($750 < Nu < 1000$). The surface flow streamline shows the upward radial migration of fluid due to the increase of the blade to tip passage height $H_{IT}$ and local secondary flow structure. On the suction side surface, the Nusselt number suddenly drops just after the leading edge ($-0.1 < X/C_{AC} < 0.0$). This is a typical feature of a laminar boundary layer thickening. In this context, the increase of the Nusselt number level at $(X/C_{AC} \approx -0.1)$ is a marker of a laminar to turbulent boundary layer transition process [53]. Near the endwalls, the lift-off line of the passage vortex is clearly identified with the surface flow streamlines. As a consequence, the Nusselt number (heat transfer) level reduces on the lift-off line of the passage vortex (locally thick boundary layer) and increases again in between the lift-off line and the blade corners, because of the new thin endwall boundary layer forming. In addition, it is observed that the surface flow streamlines near the lift-off line are...
10.3. UNCOOLED BLADE: VALIDATION OF THE COMPUTATION

Figure 10.8: Nusselt number $Nu$ at the blade surface, superimposed with the surface flow streamlines.

suddenly compressing toward midspan, at the location of the shock. This may indicates the sudden growth of the passage vortex due to the compression/dissipation effect of the shock. Eventually, it is noticed that the contours of Nusselt number shows some wiggling artifacts. This is due to a lack of data of the blade geometry combined with the local refinement of the grid: the blade surface spline contains some small bumps near the refined grid regions.

10.3.2 Design value: comparison to experiment

To validate the computation, the measured and predicted isentropic Mach number $M_{is}$ distributions at the midspan section are compared in Fig. 10.9. In the same Figure, the measured and predicted Nusselt number $Nu$ distributions at midspan section are also compared. The predicted isentropic Mach number $M_{is}$ is found in agreement with the experimental value. On the suction side, the agreement is very good. A peak Mach number of $M_{is}^{\text{max}} = 1.55$ is found near the shock-
boundary layer interaction. On the pressure side, the predicted isentropic Mach number distribution is slightly higher than the measured one (underprediction of the blade loading). This discrepancy can be attributed to the fact that the full cascade geometry is not known. This is especially true for the endwalls, so that the radial migration of the fluid may be predicted erroneously. As it can be noticed in the Nusselt number $N\nu_0$ plot, there are two experimental data sets that are displayed. Indeed, in the uncooled blade measurement set, a plain (no holes) blade is taken. The boundary layer in the suction side exhibits a laminar behavior until the shock-boundary layer-interaction. In the film-cooled measurement set, only the cooling holes in set 2 and set 4ab are utilized. Since no coolant is injected through set 1 and set 3, the boundary layer is tripped at these locations. These induces a transition of the boundary layer to a turbulent character. The turbulence model used in this thesis fails to predict transition, so that it induces a turbulent boundary layer not far downstream of the leading edge of the blade. Thus, the comparison of the measured and predicted Nusselt number should be done with the tripped boundary layer result, at least on the first half of the blade surface. As a consequence, it is observed that the heat transfer on the suction side surface is accurately predicted down to the location of the shock-boundary layer interaction. On the pressure side surface, an underprediction of 20% on average is found. Several reasons may explain this discrepancy. The freestream turbulence (3%) is certainly enhancing the turbulent mixing in the pressure side, as well as the heat transfer. The current
10.4 Film-cooled turbine blade

10.4.1 Blade loading

The measured and predicted blade loading, in terms of isentropic Mach number \( M_{is} \), are plotted in Fig. 10.10 for the two blowing ratios investigated, that is \( BR = 1.0, 1.5 \). The results are presented for coolant injection in set 1, set 3 and set 4ab. Interestingly, the blade loading is found almost identical for both blowing ratios, in the experimental and the CFD prediction. It is also very similar.
10.4. FILM-COOLED TURBINE BLADE

to the blade loading found in the uncooled case. At the locations of the holes, the blade loading could obviously not be measured. The CFD predictions using the CFD-embedded film cooling jet model give solutions that make sense in terms of the aerodynamics occurring near the hole. Firstly, the blade loading level predicted a short distance before and after the holes are always found similar to the measured one. Secondly, a decrease of the isentropic Mach number $M_{is}$ (increase of static pressure) is systematically predicted just upstream of the holes. Furthermore, an increase of the isentropic Mach number $M_{is}$ (decrease of static pressure) just downstream of the holes is also systematically predicted. This result is in agreement with the behavior of the near-hole static pressure analyzed in Chapter 7. It is worth noting that the isentropic Mach number $M_{is}$ distribution exhibits a higher gradient near the holes in the suction side than those in the pressure side. Although no conclusion can be drawn from the present results, it would be interesting to deeper investigate the effect of a high near-hole static pressure gradient (accelerating flow) upon the jet blowing and mixing.

10.4.2 Adiabatic film-cooling effectiveness on suction side

The adiabatic film-cooling effectiveness $\eta$ is analyzed. Since no experimental data is available for adiabatic film-cooling effectiveness on the pressure side, only the suction side surface is analyzed (injection from set 1). The predicted contours of adiabatic film-cooling effectiveness $\eta$ on the suction surface are plotted in Fig. 10.11 for the two blowing ratios investigated. The injection of the coolant with the use of the CFD-embedded film cooling jet model is clearly seen. The downstream evolution of the coolant is observed. In particular, the lateral mixing of the coolant and freestream increases along the surface of the suction side. It is observed that there is some distance (more than 20 hole diameters) before the coolant and freestream are fully mixed. In fact, the flow is accelerating downstream of the hole exit. Thus, the stretching of the Counter-rotating Vortex Pair (CVP) better counter-balanced its diffusion. This can favor a pinching of the coolant jet further downstream, compared to a film cooling flow without streamwise pressure gradient. At this point, it is worth recalling that in the experiment, the holes in set 1 are shaped, whereas in the CFD prediction, the holes are cylindrical. In the rear part of the suction side, the shock-boundary layer interaction provokes a blurring of the film-cooling protection. Indeed, the sudden deceleration of the flow just after the shock induces a intense viscous dissipation, hence a reduction of the film cooling protection. In addition, the spanwise length covered by the coolant suddenly reduces just after the shock. This is in agreement with the trajectory of the
surface flow streamlines shown in Fig. 10.8. As expected, the film-cooling protection appears to be slightly better for $BR = 1.0$ than for $BR = 1.5$. In order to quantify the accuracy of the numerical solution, Fig. 10.12 displays the measured and predicted adiabatic film-cooling effectiveness $\eta$ at midspan, as a function of the non-dimensionalized axial distance $X/d$. The results with the two blowing ratios investigated ($BR = 1.0, 1.5$) are provided. For the coolant injection with the lower blowing ratio case ($BR = 1.0$), the film-cooling protection is measured and predicted better than with the higher blowing ratio case ($BR = 1.5$), before the shock. Not too far downstream of the hole ($X/d < 15$), a relatively large discrepancy is found in the CFD prediction compared to the measurements. In this region,
the numerical solution exhibits a lower adiabatic film-cooling effectiveness $\eta$ than the measured one. Meanwhile, the fact that in the experiment, shaped-holes are used can be a major reason for the discrepancy. Indeed, in Fig. 10.11, it has been observed that the coolant mixes with the freestream not in the near downstream of the hole but far downstream. In general, shaped-holes having a large width, such as the ones in this experiment, induce a better mixing near the hole and also largely decrease the strength of the Counter-rotating Vortex Pair, see for instance Hyams and Leylek [46]. Thus, the discrepancy of the numerical prediction can partially be explained by the fact that the cylindrical holes lead to a lower adiabatic film-cooling effectiveness in the near downstream of the holes. Further downstream ($X/d > 15$), the CFD prediction is better, especially for the low blowing ratio case ($BR = 1.0$). However, an overprediction in the higher blowing ratio case ($BR = 1.5$) remains. After the shock, the CFD prediction tends to blur the result so that the adiabatic film-cooling effectiveness $\eta$ distribution is flat and similar for both blowing ratios.
10.4.3 Nusselt number

The measured and predicted Nusselt number $N_u$ distributions at midspan are compared in Fig. 10.13 for the two blowing ratios investigated ($BR = 1.0, 1.5$) and injection at set 1 and set 3. Near the leading edge, it is found both in the measure-

![Figure 10.13: Measured (EXP) and predicted (CFD) Nusselt number $N_u$ at midspan, for two blowing ratios ($BR = 1.0, 1.5$). Injection at set 1 and set 3.](image)

ments and computations that the Nusselt number $N_u$ is pretty similar. On the suction side, the Nusselt number decreases down to a value lower than 0.0 just at the injection site of set 1. This is because the coolant temperature ($T_{T}^{C} = 287.5$ [K]) is lower than the wall temperature ($T_{wall} = 320$ [K]). Then, just downstream of the injection site, the Nusselt number rapidly increases both in the experiment and the computation. Near this location, the peak of Nusselt number is too highly predicted, by about 20% on average for both blowing ratios. Further downstream, the heat transfer is reduced, compared to the uncooled blade, by about 25% in the experiment and by about 50% in the CFD prediction. The discrepancy of the prediction can be explained by the earlier comments made about the adiabatic film cooling effectiveness prediction: the holes at the suction side are not the same in the experiment and in the computation. Meanwhile, the measured distribution of Nusselt number is qualitatively well predicted when using the film cooling model, compared to other numerical studies involving more advanced turbulence models.
10.5 Evaluation of losses

10.5.1 Definition of losses

The designer is ultimately looking for the best thermal protection while maximizing the turbine efficiency. In this respect, an evaluation of the efficiency of the film-cooled turbine with the different computed configurations of coolant holes is a necessary step. Since the computed turbine case is a stationary cascade, there is no work exchange between the blades and the flow. Hence, the efficiency of the current computed turbine can only be described through a loss coefficient. A good review of losses in turbine cascade is provided by Denton [26]. In this thesis, two loss coefficients are computed. These are the pressure loss coefficient \( Y_p \) and the enthalpy loss coefficient \( Y_H \), see Appendix L.3 for a complete definition. The pressure loss coefficient \( Y_p \) is broadly used because it is easy to measure it from experimental cascade data. The enthalpy loss coefficient is more useful for design purposes because it scales the loss of usable energy through the blade row.

10.5.2 Results

The pressure loss coefficient \( Y_p \) is plotted in Fig. 10.14. The enthalpy loss coefficient \( Y_H \) is plotted in Fig. 10.15. The losses are systematically predicted higher in the high blowing ratio case \( (BR = 1.5) \) than in the low blowing ratio case \( (BR = 1.0) \). In the low blowing ratio case \( (BR = 1.0) \), it is observed that the losses with coolant injection can be lower than the losses without injection. Haller et al. [41] deliver an experimental evaluation of the losses occurring through the same cascade. This is done for a coolant injection at the suction side only, for different hole positions than set 1 and a slightly lower Mach number that in this thesis. They also find that the losses increase with the blowing ratio and, for a low blowing ratio (e.g. \( BR = 1.0 \) or lower), the losses can be lower than without injection. The increase of losses with the blowing ratio can be explained by the fact that when the blowing ratio increases, the jet penetrates more in the freestream so that the blade boundary layer increases in size. This leads to a region of higher skin friction and larger area of boundary layer mixing downstream of the injection site. In Figs.
10.5. EVALUATION OF LOSSES

10.14 and 10.15, it is also found that coolant injection from several rows of holes, with $BR = 1.5$, generally leads to higher losses than a coolant injection from only one row of holes. In the $BR = 1.0$ case, no global trend is found. Meanwhile, the addition of the losses obtained from the different isolated row computations does not end up to the same level of losses than for the computation with all the rows together. Thus, the losses does not linearly change with the addition of rows of holes. For the isolated row cases, the pressure loss coefficient $Y_p$ leads to conclude that the losses are higher in the pressure side than in the suction side. However, with the enthalpy loss coefficient $Y_H$, the inverse statement is made. This observation can be explained by two main reasons. Firstly, the pressure loss coefficient $Y_p$ takes into account the losses in the holes, given by the discharge coefficient $C_d$. The discharge coefficient is systematically lower in the pressure side than in the
10.5. EVALUATION OF LOSSES

suction side\(^2\). Secondly, in the \(BR = 1.0\) case, the specified coolant total pressure \(P_T^c\) inside the plenum chamber of holes in set 1 (suction side) is substantially lower than the one for set 3 and set 4ab, see Table 10.3. Thus, the mass-averaged inlet total pressure \(P_{T,in}\) is somewhat lower when injecting only from set 1 whereas the mixed-out static pressure \(P_{s,out}\) remains constant. This means that the pressure loss coefficient \(Y_p\) tends to be lower when delivering the coolant at a low total pressure. The losses are generally observed to be higher in the suction side [53], and this fact is observed with the enthalpy loss coefficient. This is because the entropy production is proportional to the third power of the boundary layer edge velocity [26]. Indeed, Fig. 10.10 shows that the flow velocity near the suction side surface is higher than near the pressure side surface. The enthalpy loss coefficient conforms to this trend.

\(^2\)for instance, for \(BR = 1.0\), set 1: \(C_d \simeq 0.8\), set 3: \(C_d \simeq 0.70\), set 4ab: \(C_d \simeq 0.73\)
10.6. SUPERPOSITION OF SOLUTIONS ON PRESSURE SIDE SURFACE

10.6 Superposition of solutions on pressure side surface

10.6.1 Definition of the superposition of the adiabatic film-cooling effectiveness

This section presents an investigation of the superposition of cooling rows at the pressure side surface, in particular its impact upon the film-cooling protection. The study aims to show that the model can be used to investigate different cooling schemes because it can handle the flow interactions resulting from the injection of coolant from several rows of holes. Hence, it is intended to show that the model can be used to study the impact of a small scale flow feature upon the global performance of an engine. In fact, in the design of film-cooled turbine, it is often useful to superpose different experimental or numerical results from the adiabatic film cooling effectiveness, as it can shrink the turnover time. This superposition principle is a direct outcome of the linearity of the energy equation. This linearity would indicate that in the absence of property variation of the flow and for the same hydrodynamic flow distribution, any variation in the film temperature could be calculated by superimposing various solutions or experimental results. Given the above stated conditions, then the optimization of the location of one hole can be independently done from the others. The goal of the present investigation is to first check for the limitations of this superposition rule in a typical 3-dimensional flow structure as in a cascade of airfoils. The superposition model of Sellers [83] is selected here for application. This model assumes that the hydrodynamic of a cooling jet is not interacting with the other ones so that the superposition of the measurements or predictions of adiabatic film cooling effectiveness $\eta_r$ for different rows $r$ gives the total adiabatic film cooling effectiveness $\eta_{tot}$. It is given by

$$1 - \eta_{tot} = \sum_{r=1}^{r=R} (1 - \eta_r)$$

(10.5)

where $R$ is the total number of rows. The main issue of the current investigation is to observe, on the pressure side surface, if the superposition of the distribution of the predicted adiabatic film-cooling effectiveness with injection at set 3 only and with injection at set 4a only is the same as with injection at set 3 and set 4a together. If the superposition principle given by Eq. 10.5 applies, then the design of a film-cooled turbine blade can be reduced to the addition of the individual design of each cooling row, i.e. the jet hydrodynamic does not need to be taken into account.
10.6.2 Results of superposition on the pressure side surface

The predicted contours of adiabatic film-cooling effectiveness \( \eta_{34a} \) and \( \eta_3 + \eta_{4a} \) on the pressure side surface are plotted in Fig. 10.16, for a blowing ratio of \( BR = 1.0 \). The same type of result is found for blowing ratio \( BR = 1.5 \), see Appendix L.4. At a first glance, the results seem to be pretty similar. However, a zoom downstream of the second row (4a) shows that a different film-cooling protection is found. The adiabatic film-cooling effectiveness is observed to be better spread over the surface of the pressure side surface, in the midspan region, for the two rows together (set 34a) than for the addition of the individual rows set 3 and set 4a. However, near the hub, the coolant shows to be better spread over the surface in the numerical solution obtained with the addition of the individual rows than for the two rows together. Thus, the coolant jets coming from the holes in set 3 must have an impact upon the evolution of the jets issuing from the holes in set 4a.

10.6.3 Hydrodynamic impact of coolant jets issuing from set 3 upon the aerothermal flow field downstream of set 4a

Definition of the visualization

In order to analyze the hydrodynamic impact of the coolant jets issuing from set 3 upon the aerothermal flow field downstream of set 4a (i.e. the film-cooling protection downstream of set 4a), plots of the predicted flow field on cross planes at three streamwise location, near the pressure side surface, are provided in Figs. 10.17-10.20. These three cross planes are denoted A, B and C. Cross plane A is located about 3 hole diameters upstream of set 4a, cross plane B is located about 1 hole diameter downstream of set 4a and cross plane C is located about 15 hole diameters downstream of set 4a. On all the cross planes, contours of the normalized total temperature \( \theta \) superimposed with the coolant secondary flow vectors \( U_{2nd}^c \) are shown. A cross plane is defined by the plane orthogonal to the flow direction at the streamwise position where the visualization is made. The coolant secondary vectors \( U_{2nd}^c \) are found as follows. The predicted velocity vectors obtained in the cooled case are subtracted by the predicted velocity vectors obtained in the uncooled case. The resulting vectors are projected on the cross planes to give the coolant secondary velocity vectors \( U_{2nd}^c \).

\[
U_{2nd}^c = (U_{cooled} - U_{uncooled}) \cdot (e_b e_b + e_t e_t)
\]  

(10.6)

where \( e_b \) and \( e_t \) are the two unity vectors defining the cross plane. The coolant secondary vectors \( U_{2nd}^c \) represent the hydrodynamic disturbance on the flow field.
Figure 10.16: Predicted contours of adiabatic film-cooling effectiveness $\eta$ at the pressure side surface for a blowing ratio of $BR = 1.0$. Injection at set 3 and set 4a (left) and injection at set 3 only, superposed with injection at set 4a only (right).
10.6. SUPERPOSITION OF SOLUTIONS ON PRESSURE SIDE SURFACE

due to the presence of the coolant jets.

Near midspan flow field

The visualization of the predicted flow field near midspan is given in Fig. 10.17 for injection from set 134a and in Fig. 10.18 for injection from set 14a. These two plots allow to observe the hydrodynamic impact of the upstream jet, issuing from set 3, upon the jet issuing from set 4a. On the cross plane upstream of set 4a (cross plane A), the flow field is undisturbed in the injection case set 14a (Fig. 10.18) whereas the counter-rotating vortices (CVP) and coolant mixing zone are clearly seen in the injection case set 134a (Fig. 10.17). At about 1 hole diameter downstream of the center of the holes pertaining to set 4a (cross plane B), the newly injected coolant jets are clearly seen. In particular, the counter-rotating motion of the vortices and the coolant core are observed. In the set 134a case (Fig. 10.17), the upstream jets are located above, on the right of the new jets. Thus, the counter-rotating vortices of the upstream jets are phase shifted compared to the counter-rotating vortices of the new jets. This induces a downward momentum push of the new jets due to the action of the upstream CVP. The result of this phenomenon is observed on the cross plane further downstream of set 4a (cross plane C). In the injection case set 14a, the evolution of the jets issuing from set 4a are found as it can be expected: the coolant jets are lifting off and pinching due to the action of the CVP. In the injection case set 134a, the flow field appears to be more complicated. The coolant jets issuing from set 4a are much more skewed with the presence of the phase-shifted upstream jets. Indeed, the action of the upstream CVP induces two main flow features. Firstly, a migration of the coolant material toward the hub direction (left side of the newly injected jets) due to the action of the left-leg of the upstream CVP. This means that each newly injected coolant jet better spread toward the hub direction, close to the blade surface. Secondly, the right-leg of the upstream CVP induces a blocking effect against the action of the CVP issuing from the new coolant jets, injected at set 4a. This blocking effect induces a squeezing of the right-leg of the CVP of the new coolant jets toward the blade surface. Thus, the coolant material is forced to stay closer to the surface and the pinching effect is less pronounced. Overall, this explain the better film-cooling protection observed in the midspan region, downstream of set 4a (near midspan), with the presence of the upstream row (set 3).
Figure 10.17: Predicted contours of normalized total temperature $\theta$ superimposed with the coolant secondary flow vectors $U_{2nd}^c$, near the pressure side surface, for a blowing ratio of $BR = 1.0$. Injection at set 3 and set 4a.
10.6. SUPERPOSITION OF SOLUTIONS ON PRESSURE SIDE SURFACE

Figure 10.18: Predicted contours of normalized total temperature θ superimposed with the coolant secondary flow vectors $U_{2nd}$ near the pressure side surface, for a blowing ratio of $BR = 1.0$. Injection at set 4a only.
Figure 10.19: Predicted contours of normalized total temperature $\theta$ superimposed with the coolant secondary flow vectors $U_{2nd}^c$, near the pressure side surface, for a blowing ratio of $BR = 1.0$. Injection at set 3 and set 4a.
Figure 10.20: Predicted contours of normalized total temperature $\theta$ superimposed with the coolant secondary flow vectors $U_{2nd}^\infty$, near the pressure side surface, for a blowing ratio of $BR = 1.0$. Injection at set 4a only.
Near hub flow field

As previously, the flow field upstream of set 4a (cross plane A) is undisturbed in the injection case set 14a (Fig. 10.20) whereas it is disturbed by the coolant jets issuing from set 3 in the injection case set 134a (Fig. 10.19). Just downstream of set 4a (cross plane B), in the injection case set 14a, the coolant jets appear to behave as a collection of single jets, as expected. In the injection case set 134a, the upstream coolant jets are now located directly above the new coolant jets. Further downstream (cross plane C), the flow field for the injection case set 14a is the same as in the midspan region. In the injection case set 134a, the newly injected coolant jets appear to have almost completely de-attached from the wall. This is observed to be due to the presence of the upstream jets, just above. These upstream jets enhance both the upward motion of the CVP in the center of the newly injected jets and the freestream entrainment at their sides. As a consequence, the film-cooling protection becomes poor when the two coolant jets are in phase, as observed near the hub endwall region for the current film-cooled turbine blade.

Summary

A schematic of the previous flow analysis is given in Fig. 10.21. In this section, it has been numerically shown that the standard superposition model of Sellers [83] does not apply in a three-dimensional turbine environment. It still need to be experimentally confirmed. Meanwhile, thanks to the use of the proposed film-cooling jet model, the effect of a small scale flow feature, such as the hydrodynamic interaction of several rows of jets, upon the global performance of an engine can be
observed and explained. In this study, it is shown that the upward radial migration of the fluid particles induced by a non-constant endwall profile and local secondary flow structure has an impact of the trajectory of the coolant jets. The trajectory of the upstream coolant jets leads to a flow configuration at the downstream row of holes that has either a positive or negative impact on the film-cooling protection, as a function of the spanwise position. If the upstream coolant jets are phase-shifted with the new coolant jets, then a positive impact on the film-cooling protection is found. If the upstream jets are in phase with the new coolant jet, then a negative impact on the film-cooling protection is found. Thus, it would be interesting and possible with the model to study more in details the impact of passage flow distortions (due to non-constant endwall profile, incoming vortices, hot streaks, etc...) on the position of the film-cooling holes so that on the film-cooling protection. This could allows to better optimize the design of film-cooled turbine blades.

10.7 Conclusion

The use of the CFD-embedded jet model in a three-dimensional linear turbine cascade, representative of turbine rotor geometry and flow condition at midspan, has been shown. In total, 67 cooling holes have been taken into account. There have been placed on the suction and pressure surface. A computational mesh of $2.13 \cdot 10^6$ grid nodes has been used. The mesh has been clustered near the holes in order to preserve accuracy. This grid size appears to be more than 10 times smaller than if a mesh including the holes and plenum chamber would be used. In addition, the meshing complexity is alleviated. A stable solution has been obtained for different cooling schemes and blowing ratios. It has been shown that the computational overhead in such a flow configuration is about $\sigma = 0.1\%$ per hole. From a restart solution, the same number of iterations are needed to converge the uncooled and film-cooled blade (with model) computation. However, the total number of grid nodes is about 4 times higher than for a standard computational mesh for uncooled blade computation. This is a necessary condition to ensure that the macro flow features of the smallest flow scale to be computed (namely the film-cooling jet) are captured. Predictions of blade loading are in excellent agreement with experimental data. Predictions of adiabatic film-cooling effectiveness and of Nusselt number are in agreement with experimental data. Different arrangements of cooling rows are investigated on pressure side. It appears that the interaction of coolant jets coming from different rows can increase or decrease film-cooling protection, as a function of the arrangement of the rows and passage flow distortion.
Seite Leer / Blank leaf
Chapter 11

Conclusion

11.1 Conclusive summary

This thesis has dealt with multi-scale flows pertaining to an energy conversion engine. Based on the question proposed in the Introduction, see section 1.4, a solution has been proposed and presented step-by-step in this thesis. The challenge that has been raised concerns the modeling of the macro flow features of a small scale flow phenomenon typical of gas turbine technology, that is namely a film-cooling jet. To the author’s knowledge, this is the first time that a fully three-dimensional film cooling jet model has been proposed for use in a RANS solver. The modeling approach and the numerical implementation have shown to be relevant in regards of the prediction results obtained in different geometries and flow configurations. The major breakthrough of this thesis is that the proposed model is computationally efficient:

- The current modeling approach fully alleviates the meshing complexity that arises when the holes and plenum chamber are gridded.

- For a film-cooled turbine blade computation, the current modeling approach leads to a grid size at least 10 times lower than if the holes and plenum chamber were modeled.

- A film-cooled turbine blade with 67 holes can be computed with a mesh having about 2 Million grid nodes.

- The use of the model for a steady flow prediction in a film-cooled turbine leads to a computational overhead of 0.1% per hole.

The key points of this work are summarized in the following.
11.1. CONCLUSIVE SUMMARY

A RANS-based CFD solver

A RANS-based CFD code, so-called MULTI3, has first been developed by the author in the course of this work. The development, optimization and validation of the CFD solver, with different test cases typical of turbomachinery flows [15], [16], took about 1½ years. This "in-house computational tool" has been designed for academic research in turbomachinery flows. The CFD solver is built upon an explicit, time-marching, Ni-Lax-Wendroff algorithm, based on a Finite Volume Method (FVM) for spatial discretization. The computational stencil is vertex-based. This central differencing algorithm is typical to those used for solving compressible flows in industrial configurations. Although its simplicity, robustness and relative accuracy (for a second-order accuracy scheme) are a consequent advantage, its odd-even aliasing behavior, typical of central schemes, is a major source of errors when analyzing flow derivatives.

A feature-based film-cooling jet model

The first part of the core of this thesis is the development of a film-cooling jet model based on the macro flow features occurring in the near hole region of such a flow. The macro flow features are identified through an in-depth review of the literature covering this subject. Due to the rapid jet bending and the re-arrangement of its structure occurring between the hole exit plane and just after the hole trailing edge, the model is developed in the so-called plane of injection, typically located in a jet cross plane just after the hole exit. The macro flow features that are accounted for are the bulk jet trajectory, jet counter-rotating vortex pair (CVP), wake zone and the jet to freestream mixing. The model is partially based on the governing equations of fluid dynamics and has empirical coefficients. The number of inputs to the model is small. Only the hole geometry, as well as the freestream and coolant flow conditions need to be input. The model is only valid for long cylindrical holes.

A numerical immersion of the film-cooling jet model

The numerical implementation of the model in a RANS-based solver represents the second part of the core of this thesis. A discussion of the different existing technique to numerically include the film-cooling jet model is provided. Based on this literature survey, the Immersed Boundary Method (IBM) is chosen due to its relatively low computational cost versus accuracy, easy integration to any CFD code and its locality. The ground of this method is to immerse a three-dimensional surface in a standard mesh through the specification of the flow at ghost-nodes.
11.1. CONCLUSIVE SUMMARY

This specification is done as a function of the surrounding computed flow field and also the boundary conditions representing the effect of the immersed surface upon the surrounding flow field. In between the hole exit and the plane of injection, the assumption of a very thin mixing limit between the coolant and freestream fluids is used. This allows the representation of the jet boundary by a toroidal surface. A slip boundary condition is specified at the jet toroidal surface since no vorticity can be created at this location. The plane of injection cuts the toroidal jet surface, allowing to inject the film-cooling jet model. The full CFD-embedded film-cooling jet model is localized and immersed in the computational mesh through the use of boolean functions that set the character (inside, outside or in between the jet surface) of each grid node and cell. The algorithm is entirely local, working in a so-called "3D film-cooling box".

An experimentally-aided calibration of the model coefficients

This work is part of a wider project about unsteady film-cooling flows. In connection to this work, a film-cooled flat plate experiment [13] has been built to facilitate the comprehension of film-cooling flows and also to provide accurate three-dimensional data for the calibration of the present model. Since a large number of model coefficients (16) need to be tuned, a strategy of calibration is shown but is not unique. It is assumed that the model coefficients are only function of the momentum flux ratio $IR$ and injection angles $\alpha_0$, $\beta_0$. The functional dependencies are either constant or linearly or algebraically varying as a function of the momentum flux ratio $IR$. The calibration is done twice, for two streamwise injection angles $(\alpha_0, \beta_0)$. The circulation $\Gamma$ of the two counter-rotating vortices is redistributed as a function of the lateral injection angle $\beta_0$. The model coefficients describing the toroidal jet surface are tuned so that the mass flow at the plane of injection is conserved. The calibrated model shows an excellent agreement with three-dimensional accurate experimental data (about 200 to 300 measurement points within the jet cross section), for a broad range of momentum flux ratios $(0.6 < IR < 7.3)$ and two streamwise injection angles $(\alpha_0 = 30^\circ, 50^\circ)$.

An evaluation of the computational performance of the model

The computational performance of the CFD-embedded film-cooling jet model is evaluated in a steady film-cooled flat plate configuration. A large spectrum of computational mesh sizes is tested (20 in total). It is proposed to express the mesh refinement $N$ as a function of the hole diameter. As the mesh refinement in the
direction normal to the wall is already set by the boundary layer characteristics, only refinements in the axial (X) and in the lateral (Y) direction are considered. A marginal gain coefficient $C_Q$ is introduced. It allows to quantify the gain in computational accuracy if one is willing to spend one more time unit for the computation. Spending more computational time means a finer mesh. The computational accuracy is perfect when a totally grid-independent solution is found. The computational accuracy is measured through the vortex circulation $\Gamma$ and adiabatic film-cooling effectiveness $\eta$. A mesh with $N_X \approx 4 - 7$ and $N_Y = 7 - 11$ grid nodes per hole diameter is found to be the best suited for an optimal use of the CFD-embedded film-cooling jet model. For this type of mesh resolution, the local mass flow error is of the order of $e_{\text{LOC}} \approx 1\%$. The overhead $\sigma$, which expresses the extra time spent per iteration compared to a no film-cooling case, is of the order of $\sigma \approx 1\%$.

**A study of the impact of the momentum flux ratio $IR$ and streamwise injection angle $\alpha_0$ upon jet aerodynamics**

The use of the CFD-embedded film-cooling jet model is validated against several film-cooled flat plate experiments. In a first step, the capability of the model to predict the aerodynamics of a coolant jet is checked. A clear distinction is done between region-I (in the near downstream of the hole), where flow gradients are high so that convective entrainment is dominant and region-II (in the far downstream of the hole) where flow gradients are low, so that diffusive entrainment is dominant. As in the experiment, it is found that the momentum flux ratio $IR$ is the major parameter that influences the trajectory of the coolant jet. It typically penetrates vertically 50\% more when $IR = 4.0$, compared to a flow regime where $IR = 1.0$. The streamwise injection angle $\alpha_0$ influences the trajectory of the coolant jet much less. Meanwhile, the wake structure shows to differ from the $\alpha_0 = 30^\circ$ case compared to the $\alpha_0 = 50^\circ$ case. In particular, in region-I, the wake velocity deficit is more pronounced in the $\alpha_0 = 50^\circ$ case than in the $\alpha_0 = 30^\circ$ case. This can be explained by the fact that the CVP vorticity in the wake region decays much more quickly in the $\alpha_0 = 30^\circ$ case.

**An evaluation of the predictive capability of the model for aerodynamics and heat transfer values, in an isotropic turbulence context**

Low momentum flux ratio cases ($IR \leq 1.0$) are better predicted than high momentum flux cases ($IR \geq 4.0$). The major discrepancy in the CFD prediction
using the model is found to occur in the wake region. In general, the wake velocity deficit is overpredicted by an order of magnitude of 10%. This discrepancy is shown to emerge from a misprediction of the evolution of the counter-rotating vortex pair. The lateral spreading of the vortices is constantly overpredicted by 15% to 30%, which leads to entrain more low-momentum fluid toward the wake. In connection to this fact, the predicted vorticity in the wake is shown to decay too slowly, especially for the \( \alpha_0 = 30^\circ \) case. Based on previous experimental and computational works, it is expected that turbulence anisotropy, enhancing the lateral diffusion of the coolant, should be taken into account in this region. The circulation of the vortices at the plane of injection is constantly overpredicted by about 10% to 20%. The deflection of the viscous freestream flow, due to the presence of the toroidal jet surface, induces the appearance of vorticity. This vorticity originally comes from the interior of the immersed toroidal shape because of the imposition of the immersed boundary conditions. Thus, the vorticity appearing at the sides of the immersed jet body is effectively coming from the coolant, as experimentally observed. The effect of the misprediction of the coolant jet aerodynamics, especially in the wake, upon the prediction the adiabatic film-cooling effectiveness \( \eta \) is clearly demonstrated. The predicted laterally-averaged film-cooling effectiveness \( \bar{\eta} \) agrees in general very well with experimental data. In particular, it is shown that the predictive error of laterally-averaged surface metal temperature, compared to experimental data, is less than 1% with the model. Meanwhile, the overprediction of the CVP strength, coupled with the lack of the enhancement of the lateral turbulent mixing, results in underpredicting the lateral diffusion of coolant fluid and overpredicting the centerline adiabatic film-cooling effectiveness. The model has been tested in extreme injection case, that is with high lateral injection angles (\( \beta_0 = 30^\circ, 60^\circ, 90^\circ \)). With the increase of the lateral injection angle \( \beta_0 \), the switch of the coolant aerodynamics behavior is reproduced by the CFD prediction using the model; it switches from a symmetrical counter-rotating vortex structure to an asymmetrical structure where only one vortex issuing from the back side of the coolant jet. Up to a lateral angle of \( \beta_0 = 30^\circ \) and moderate blowing ratio (\( BR \leq 0.9 \)), the CFD-embedded film-cooling jet model is capable of giving a reasonably accurate prediction of the laterally-averaged film-cooling effectiveness. Beyond \( \beta_0 = 30^\circ \), the model fails in most cases to accurately predict film-cooling effectiveness.
A comparative study of the model versus other injection strategies

The predictive capabilities of two alternative injection strategies have been compared to the CFD-embedded Film-Cooling jet Model (FCM). They have been chosen to represent the two extreme strategies to inject a coolant jet. The simplest strategy, so-called Wall Injection (WI), consists of specifying the coolant flow condition at the wall cell faces that should be inside the hole exit area. The most complex strategy, so-called Full Injection (FI), consists of explicitly computing the flow in the plenum chamber and hole. The computational mesh for the WI injection is the same as the one used for the model. The computational mesh for the FI strategy, containing about 3 times more grid nodes than for the previous strategies, is shown to be much more complicated, due to the more complex geometry to be meshed. The total resolution time of the steady coolant flow field is the same for the WI and FCM strategies, whereas it is about 6 times bigger for the FI strategy. Overall, the best predictive accuracy is achieved with the FCM strategy. The FI strategy delivers a solution similar to the FCM solution whereas the WI strategy largely mispredicts the aerodynamics of the jet. This is shown to be due to the fact that the specification of the coolant conditions at the wall faces are erroneous. In the FI strategy, it is shown that the complex flow field existing in the hole, exhibiting a vortical system, is certainly a very challenging flow to be accurately predicted using a RANS-based method.

A method for dealing with quasi-steady pulsation of the near-hole flow field

The pulsation of coolant jets in turbines is first introduced and discussed. A state of the art review of the current knowledge about this phenomenon is provided. The large-scale fluctuation of the near-hole static pressure $\bar{P}_s$ is shown to be the major phenomenon to influence the blowing of a coolant jet. Upon this review, a quantification of the range of jet pulsation is done in terms of the pulsating frequency and amplitude. The pulsating jet flow is computed in a flat plate configuration. This limits the jet pulsation regimes to values a bit lower than what is encountered in turbines. Only quasi-steady pulsation is considered. The CFD-embedded film-cooling jet model is shown to work in such an unsteady flow environment, with an overhead of only $\sigma \approx 4\%$. The update of the jet shape and model, in the CFD solver, is automatically carried out as a function of the large scale pulsation of the near-hole static pressure. It is shown that for a low mean blowing ratio ($\bar{BR} = 0.5$), the increase of the near-hole static pressure $\bar{P}_s$ leads to the decrease of the instantaneous blowing ratio $\bar{BR}$. Interestingly, for a high mean blowing ratio
The time mean flow field of a quasi-steady, moderately pulsating jet issuing from a long hole is similar to the corresponding steady coolant injection. Further investigations of a pulsating jet in a turbine configuration are needed to better characterize its influence upon heat transfer rate.

**A relevant demonstration of the use of the model in a film-cooled turbine**

The use of the CFD-embedded film-cooling jet model in a three-dimensional linear turbine cascade, representative of turbine rotor geometry and flow condition at midspan, has been shown. In total, 67 cooling holes have been taken into account. There have been placed on the suction and pressure surface. A computational mesh of \(2.13 \times 10^6\) grid nodes has been used. In particular, the mesh has been clustered near the holes in order to preserve accuracy. This grid size appears to be more than 6 times smaller than if a mesh including the holes and plenum chamber would be used. In addition, the meshing complexity is alleviated. A stable solution has been obtained for different cooling schemes and blowing ratios. It has been shown that the computational overhead in such a flow configuration is about \(\sigma = 0.1\%\) per hole. From a restart solution, the same number of iterations are needed to converge the uncooled and film-cooled blade (with model) computation. However, the total number of grid nodes is about 4 times higher than for a standard computational mesh for uncooled blade computation. This is a necessary condition to ensure that the macro flow features of the smallest flow scale to be computed (namely the film-cooling jet) are captured. Predictions of blade loading are in excellent agreement with experimental data. Predictions of adiabatic film-cooling effectiveness and of the Nusselt number are in agreement with experimental data. Different arrangements of the cooling rows are investigated on the pressure side. It is shown that the interaction of the coolant jets coming from different rows can increase or decrease the film-cooling protection, as a function of the arrangement of the cooling rows and passage flow distortion.

**11.2 Recommendations for future work**

Recommendations for future work with the CFD-embedded film-cooling jet model are given in the following.
Further development and use of the film-cooling jet model

The current film-cooling jet model is available only for long cylindrical holes, with a streamwise injection angle of no more than \( \alpha_0 = 60^\circ \) and for a lateral injection angle of no more than \( \beta_0 = 30^\circ \).

- **Shaped-holes.** There is a tendency of having shaped-holes in film-cooled turbines, due to their better performance. The model should be further develop to be able to reproduce the flow field resulting from such cooling holes. In particular, the counter-rotating vortex (CVP) pair structure seems to be strongly affected by the hole geometry. Shaping the holes tends to reduce the impact of the CVP structure. In addition, coolant jets issuing from shaped holes appear to better stick to the wall compared to coolant jets issuing from cylindrical hole at the same blowing ratio. Thus, a new calibra­tion the coefficients used for the jet trajectory model and the mixing model should be carried out.

- **Lateral injection angle.** Lateral injection is often interesting since it pushes the coolant fluid to better spread over the surface to be cooled. The mixing occurring in the near-hole region is certainly changing. Also the counter-rotating vortex pair is degrading to one big vortex issuing from the back side of the hole when the lateral injection angle is increased.

- **Showerhead.** The cooling of the leading edge of the Nozzle Guide Vane (NGV) and the downstream rotors is critical. Indeed, this is where the heat transfer rate is the highest. Generally, holes inclined at a streamwise injection angle of \( \alpha_0 = 90^\circ \) are placed at this location. They are called showerhead. The coolant flow field at the hole exit may be very different from what is experienced at the suction or pressure side of a blade. Further development of the film-cooling jet model for such hole configuration is strongly recommended.

- **Other jet flow cases in turbine.** Jet flows in turbines are not exclusively used for film-cooling purposes. For example, the design of modern high-pressure rotating turbine blades are more and more often using a tiny tip clearance gap near the outer casing. The flow control (as well as the cooling) of such blade parts, to lower the induced losses, can be done with discrete injections. It is proposed to test and use the model in such configurations.

- **Pulsating jets.** For quasi-steady jets, it has been shown that the film-cooling jet model should be upgraded so as to take account of a time lag factor. Fur-
ther investigation are needed to characterize this parameter. For non-quasi steady jets, the model as it is cannot be used. In this respect, further consideration of the main assumptions used for the current model are advised.

- **Turbulence.** The model has been used in the context of an algebraic and isotropic turbulence model (Baldwin-Lomax). As the turbulence in coolant jet is often complex, with an anisotropic behavior, it is recommended to develop a model of the turbulent flow field on the plane of injection. This model should serve for computations involving more complex turbulence models, having better heat transfer prediction capabilities, such as the $k-\omega$ model.

**Further development and use of the Immerged Boundary Method**

The current Immerged Boundary Method (IBM) used to numerically include the film-cooling jet model is based on the assumption that a slip boundary condition is imposed at the jet surface.

- **Thermal boundary conditions.** The thermal boundary condition of the current model assumes a constant temperature on the jet toroidal surface. This temperature is given by the averaged value of the coolant and freestream temperature. It would be worth testing more accurate thermal boundary conditions, such as specifying a heat exchange between the coolant and the freestream. In this context, the systematic discrepancy that is found in the prediction of adiabatic film-cooling effectiveness in the near hole downstream region could be reduced. This is because the thermal mixing at the sides of the jet would be more in agreement with the suddenly imposed thermal mixing on the 2D plane of injection.

- **Turbulence.** If one wishes to use the model with a more complex turbulence model, based on solving partial differential equations for the transport of turbulent quantities, then new boundary conditions on the jet surface should be developed for these turbulent quantities. This should be done in connection with experimental measurements and/or computational predictions (at the level of LES and/or DNS) of such flows.

- **Freestream penetration and permeability.** As it has been discussed in Chapter 9, if a high near-hole static pressure $P_s$ relatively to the plenum coolant total pressure $P_T$ is occurring, then it may happen that the freestream
flow is actually ingested inside the hole. From this point of view, it is advised to develop new boundary conditions at the immersed jet surface so as to allow penetration of the freestream flow inside. In addition, and in connection with the development of more advanced thermal boundary conditions, a permeable boundary condition could also be implemented for the immersed toroidal jet body. Indeed, the permeable boundary condition would allow to transfer not only energy but also mass through the toroidal jet body. This can be important because a non-negligible mass fraction may be ingested between the hole exit and plane of injection due to the action of the nascent CVP structure and freestream momentum. This may induce a better relaxation of the computed near hole flow field so that a better prediction of the near hole adiabatic film cooling effectiveness.

- Using IBM for local geometrical change. In a more general context, the use of the IBM is very promising for flow problems involving local complex geometrical features. For example, in industrial flows, the investigation of the effect of a local geometrical change of a mechanical component upon the general flow behavior is often desired because it could significantly improve the overall efficiency of the component. The use of the IBM allows to modify the geometry in a given grid without having to redo the whole meshing. Significant gain in effort and time in the optimization process could be obtained.

Using the model for optimization

It has been shown that the current model can be used to solve flow through a film-cooled turbines in a reasonable time scale. In the continuation of this work, it would be worth implementing, testing and using the model in a real optimization loop. Since the model appears to be robust for predicting film-cooled turbine, its integration in a design tool seems promising.
11.2. RECOMMENDATIONS FOR FUTURE WORK
Bibliography


ASME paper 90-GT-78, Proceedings of the Gas Turbine and Aeroengines Congress and Exposition, June 11-14, Brussels, Belgium


Appendix A

Toroidal shape

A.1 Mathematical description

A toroidal shape is chosen to represent the jet three-dimensional surface in the near hole region. The mathematical description of the toroidal shape is done in the jet intrinsic frame of reference $(\chi, \xi, \eta)$. It is assumed that the toroidal shape is symmetrical in respect to the $\eta$-direction (lateral direction from the point of view of the jet) where the symmetry plane is lying on the $(\chi, \xi)$-plane, see Fig. A.1. The big circle, which is the curved axis bearing the solid small ring, has a constant radius $R_m$. The center of the big circle is located at point $X_{tc} (\chi_0, \xi_0, 0)$. The big circle goes through the hole exit plane, at the point $X_{ho} (\chi_{ho}, \xi_{ho}, 0)$ and also through the center of the jet $X_{jc} (\chi_{jc}, \xi_{jc}, 0)$ on the plane of injection. The small "circle" of the toroidal shape, which is the circular section that is translated along the big circle axis to create the toroidal surface, is in fact an ellipse. This "small ellipse" represents the cross section of the jet, which may vary in size along the jet path, as explained in the previous section. The two axis of the small ellipse are denoted by $A(\chi, \xi)$ (this is the axis of the small ellipse in the $(\chi, \xi)$-plane) and $B(\chi, \xi)$ (this is the axis of the small ellipse in the $\eta$-direction). They actually are an extension in the third dimension $\chi$ of the two axis of the ellipse which represents the mixing zone, see Chapter 3. Intuitively, the mathematical description of the toroidal surface is given by

\[ T_{intuitive} (\chi, \xi, \eta) = \left( \frac{Y(\chi, \xi) - R_m}{A(\chi, \xi)} \right)^2 + \left( \frac{\eta - \eta_0}{B(\chi, \xi)} \right)^2 - 1 = 0 \]  \hspace{1cm} (A.1)

where

\[ Y(\chi, \xi) = \sqrt{(\chi - \chi_0)^2 + (\xi - \xi_0)^2} \]  \hspace{1cm} (A.2)

\[ R_m = \sqrt{(\chi_{jc} - \chi_0)^2 + (\xi_{jc} - \xi_0)^2} \]  \hspace{1cm} (A.3)
Figure A.1: Description of the toroidal three-dimensional surface mimicking the jet body near the hole.
Obviously, \( \eta_0 \) is equal to zero since the toroidal shape is placed on the \((\chi, \xi)\)-plane, where \( \eta \) is equal to zero. To have a more compact expression of the toroidal three-dimensional surface, it is set that

\[
\Omega (\chi, \xi) = Y (\chi, \xi) - R_m
\]  

(A.4)

To simplify the notation, the spatial components \((\chi, \xi, \eta)\) of the functions are from now on omitted, as they all have been shown. Plugging Eq. A.4 in Eq. A.1 and some algebra, the final mathematical expression for the toroidal surface is derived.

\[
T = \eta^2 A^2 + \Omega^2 B^2 - A^2 B^2 = 0
\]  

(A.5)

The geometrical place of the toroidal surface is found when the function \( T (\chi, \xi, \eta) \) is equal to zero.

**A.2 Axis of the small ellipse - jet cross section**

The axes of the small ellipse are varying as a function of its position on the \((\chi, \xi)\)-plane. To simplify the problem, it is first assumed that the same type of function is describing the two axes \( A (\chi, \xi) \) and \( B (\chi, \xi) \), that is the function \( \Lambda (\chi, \xi) \). A periodic behavior is assumed for the function \( \Lambda \) whose period is given by the aperture angle \( \Delta \theta = \theta_{le} - \theta_{wi} \) for \( A \) and \( \Delta \theta = \theta_{ho} - \theta_{wi} \) for \( B \) between the hole exit and the plane of injection, as shown in Fig. A.2. The function \( \Lambda \) is given by

\[
\Lambda (\chi, \xi) = \Lambda_m + d\Lambda \cos \left( \frac{\theta - \theta_1}{\theta_2 - \theta_1} \right)
\]  

(A.6)

and the angle \( \theta \) reads

\[
\theta = \arctan \left( \frac{\xi - \xi_0}{\chi - \chi_0} \right)
\]  

(A.7)

For the axis \( A \), \( \theta_1 = \theta_{le}, \theta_2 = \theta_{wi} \) and for the axis \( B \), \( \theta_1 = \theta_{ho}, \theta_2 = \theta_{wi} \). Furthermore, the first derivative of the function \( \Lambda \) is zero at \( \theta = \theta_1, \theta_2 \).

\[
\left( \frac{d\Lambda}{d\theta} \right)_{\theta=\theta_{1,2}} = 0
\]  

(A.8)
A.3. SCALING OF THE TOROIDAL SHAPE PARAMETERS

Figure A.2: Variation of the two axis A and B of the jet elliptic cross section ellipse.

In fact, the angle of the toroidal surface\(^1\) is enforced to match the angle of injection at the hole exit (that is \(\alpha_0, \beta_0\) in the \((\chi, \xi)\)-plane and 90° in the \((r, \theta)\)-plane) and on the plane of injection (that is \(\alpha\) and \(\beta\)). As it has been seen in Chapter 3, the axis \(A\) is not the same on the plane of injection whether it is considered above or below the jet center. As it has been assumed that the toroidal shape is symmetrical in respect to the \(\eta\)-direction, the axis \(B\) is the same in both lateral sides of the jet, and varies following the \(\Lambda\) function. Furthermore, the axis \(A\) in its lower side, that is \(A_1\), see Chapter 3, is set constant. Only \(A_2\) varies as the \(\Lambda\) function.

A.3 Scaling of the toroidal shape parameters

Obviously, the parameters of the toroidal shape, contained in the functions \(A(\chi, \xi), B(\chi, \xi)\) and \(\Omega(\chi, \xi)\) are not known a priori. They are listed in details in Table A.1. To find them, it is necessary to take into account the available information, that is the geometry of the hole as well as the characteristics of the jet mixing zone on the plane of injection. These two features allow to constraint the search of the unknown parameters defining the toroidal shape in such a way to get a unique solution. The geometrical characteristics known owing to the knowledge of the hole geometry and jet mixing are given in Table A.2. The center of

\(^1\)this is the local angle \(\alpha\) in respect to the streamwise direction
A.3. SCALING OF THE TOROIDAL SHAPE PARAMETERS

<table>
<thead>
<tr>
<th>Geometrical characteristic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( X_{tc} (\chi_0, \xi_0, 0) )</td>
<td>Center of the toroidal shape</td>
</tr>
<tr>
<td>( R_m )</td>
<td>Radius of the toroidal big circle</td>
</tr>
<tr>
<td>( A_m, dA, A_1 )</td>
<td>Axis of the small ellipse, in ( \xi )-direction, of the toroidal shape</td>
</tr>
<tr>
<td>( B_m, dB, \theta_{ho} )</td>
<td>Axis of the small ellipse, in ( \eta )-direction, of the toroidal shape. Aperture angle on the plane of injection</td>
</tr>
</tbody>
</table>

Table A.1: Unknown parameters to be used as a function \( T (\chi, \xi, \eta) \) which defines the toroidal surface.

<table>
<thead>
<tr>
<th>Geometrical characteristic</th>
<th>Description</th>
<th>Known from</th>
</tr>
</thead>
<tbody>
<tr>
<td>( d )</td>
<td>Hole diameter</td>
<td>Hole geometry</td>
</tr>
<tr>
<td>( X_{le} (\chi_{le}, \xi_{le}, 0) )</td>
<td>Leading edge point of the hole</td>
<td>Hole geometry</td>
</tr>
<tr>
<td>( t_{le}, \theta_{le} )</td>
<td>Coolant flow injection direction</td>
<td>Hole injection angles ( \alpha_0 ) and ( \beta_0 )</td>
</tr>
<tr>
<td>( X_{wi} (\chi_{wi}, \xi_{wi}, 0) )</td>
<td>Top side of the jet on the plane of injection</td>
<td>Located at height ( A_2 ) from the jet center position ( X_{jc} ), in the direction of ( e_\xi )</td>
</tr>
<tr>
<td>( t_{wi}, \theta_{wi} )</td>
<td>Coolant flow direction on the plane of injection</td>
<td>Local jet angles ( \alpha ) and ( \beta )</td>
</tr>
<tr>
<td>( X_{jc} (\chi_{jc}, \xi_{jc}, 0) )</td>
<td>Center of the jet on the plane of injection</td>
<td>Jet trajectory model</td>
</tr>
</tbody>
</table>

Table A.2: Geometrical characteristics of the toroidal jet surface that are known a priori.

The toroidal shape \( X_{tc} \) can be inferred from the leading edge position of the hole \( X_{le} \), its aperture angle \( \theta_{le} \), the windward side position of the jet on the plane of injection \( X_{wi} \) and its aperture angle \( \theta_{wi} \) as shown in Fig. A.3. In the center plane \((\chi, \xi)\), the local angle \( \alpha \) at any point within the toroidal surface can be computed.
A.3. SCALING OF THE TOROIDAL SHAPE PARAMETERS

Figure A.3: Normal vectors to the toroidal surface at $X_{le}$ and $X_{wi}$ to find the center $X_{tc}$ of the toroidal shape

using the normal vector of the surface.

$$\alpha = - \arctan \left( \frac{n_{\chi}}{n_{\xi}} \right) = \theta - \frac{\pi}{2} \quad (A.9)$$

Now the toroidal surface at points $X_{le}$ and $X_{wi}$ in the plane $(\chi, \xi)$ can be simply defined as a circle, since the derivative of the small ellipse axis is zero at these locations, see Eq. A.8. This means that the equation of the "surface" at these locations, in the plane $(\chi, \xi)$, reduces to

$$C(\chi, \xi) = (\chi - \chi_0)^2 + (\xi - \xi_0)^2 - R = 0 \quad (A.10)$$

The normal vector of such a "surface" is given by

$$\mathbf{n} = \begin{bmatrix} n_{\chi} \\ n_{\xi} \end{bmatrix} = \begin{bmatrix} \frac{\partial C}{\partial \chi} \\ \frac{\partial C}{\partial \xi} \end{bmatrix} = \begin{bmatrix} 2(\chi - \chi_0) \\ 2(\xi - \xi_0) \end{bmatrix} \quad (A.11)$$

Using Eqs. A.11 and A.9 at points $X_{le}$ and $X_{wi}$ leads to the following system of equations

$$\begin{align*}
\alpha_{le} &= - \arctan \left( \frac{\chi_{le} - \chi_0}{\xi_{le} - \xi_0} \right) \\
\alpha_{wi} &= - \arctan \left( \frac{\chi_{wi} - \chi_0}{\xi_{wi} - \xi_0} \right)
\end{align*} \quad (A.12)$$
A.3. SCALING OF THE TOROIDAL SHAPE PARAMETERS

The solution of the above system gives the position of the center of the toroidal shape \((x_0, \xi_0)\).

\[
\begin{cases}
  x_0 = \frac{x_{le} - \left( \frac{\tan \alpha_{le}}{\tan \alpha_{wi}} \right) x_{wi} + (\xi_{le} - \xi_{wi}) \tan \alpha_{le}}{1 - \left( \frac{\tan \alpha_{le}}{\tan \alpha_{wi}} \right)} \\
  \xi_0 = \xi_{wi} - \frac{x_0 - x_{wi}}{\tan \alpha_{wi}}
\end{cases} \tag{A.13}
\]

The radius \(R_m\) of the big circle is simply given by the distance between the center of the jet on the plane of injection \(X_{jc}\) and the center of the toroidal shape \(X_{tc}\).

\[
R_m = \sqrt{(x_{jc} - x_0)^2 + (\xi_{jc} - \xi_0)^2} \tag{A.14}
\]

The point \(X_{ho}\) pertaining to the hole exit plane need to be found in order to derive the aperture angle \(\theta_{ho}\). To do so, the point \(X_{lm}\) which is on the toroidal shape cross section where \(X_{le}\) lies, see Fig. A.1, is first computed. Using Eq. A.11 gives the following system of equations.

\[
\begin{cases}
  \alpha_{le} = \alpha_{lm} = - \arctan \left( \frac{x_{lm} - x_0}{\xi_{lm} - \xi_0} \right) \\
  (x_{lm} - x_0)^2 + (\xi_{lm} - \xi_0)^2 - R_m = 0
\end{cases} \tag{A.15}
\]

The solution of the above system of equation is

\[
\begin{cases}
  x_{lm} = x_0 - \left( \frac{R_m \tan \alpha_{le}}{\sqrt{1 + \tan^2 \alpha_{le}}} \right) \\
  \xi_{lm} = \xi_0 + \left( \frac{R_m}{\sqrt{1 + \tan^2 \alpha_{le}}} \right)
\end{cases} \tag{A.16}
\]

In analogy to the previous system of equation A.15, the system of equation to be solved for \(X_{ho}\) is

\[
\begin{cases}
  \alpha_{ho} = \alpha_{le} \arccos \left[ 1 + \sqrt{(x_{ho} - x_{le})^2 + (\xi_{ho} - \xi_{le})^2} \right] = - \arctan \left( \frac{x_{ho} - x_0}{\xi_{ho} - \xi_0} \right) \\
  (x_{ho} - x_0)^2 + (\xi_{ho} - \xi_0)^2 - R_m = 0
\end{cases} \tag{A.17}
\]

This gives the solution for \(X_{ho}\).

\[
\begin{cases}
  x_{ho} = x_0 - \left( \frac{R_m \tan \alpha_{ho}}{\sqrt{1 + \tan^2 \alpha_{ho}}} \right) \\
  \xi_{ho} = \xi_0 + \left( \frac{R_m}{\sqrt{1 + \tan^2 \alpha_{ho}}} \right).
\end{cases} \tag{A.18}
\]
Hence, the aperture angle $\theta_{ho}$ is known, see Eq. A.7. The two coefficients $B_m$ and $dB$ of the function $B$ (see Eq. A.6) are easily found. Indeed, searching their extreme value at the two points $X_{ho}$ and $X_{wi}$ gives

$$\begin{align*}
B (\theta_{ho}) &= B_m + dB = d/2 \\
B (\theta_{wi}) &= B_m - dB = B_{2}^{jet} \\
\Rightarrow \quad B_m &= \frac{d/2 + B_{2}^{jet}}{2} \\
\frac{dB}{2} &= \frac{d/2 - B_{2}^{jet}}{2}
\end{align*} \tag{A.19}$$

where the semi-axis $B_{2}^{jet}$ should normally be equal to the semi-axis $B_{m2}$ of the mixing zone, see Chapter 3. However, in order to ensure mass conservation, see Chapter 5, this variable is set as a model coefficient to be calibrated. The same procedure as above is applied to search $A_m$ and $dA$.

$$\begin{align*}
A (\theta_{le}) &= A_m + dA = \sqrt{(\chi_{lm} - \chi_{le})^2 + (\xi_{lm} - \xi_{le})^2} \\
A (\theta_{wi}) &= A_m - dA = A_{2}^{jet}
\end{align*} \tag{A.20}$$

hence

$$\begin{align*}
A_m &= \frac{\sqrt{(\chi_{lm} - \chi_{le})^2 + (\xi_{lm} - \xi_{le})^2} + A_{2}^{jet}}{2} \\
dA &= \frac{\sqrt{(\chi_{lm} - \chi_{le})^2 + (\xi_{lm} - \xi_{le})^2} - A_{2}^{jet}}{2}
\end{align*} \tag{A.21}$$

where $A_{2}^{jet}$ is also set as a model coefficient for the same reason as for $B_{2}^{jet}$. The semi-axis $A_1$ is also set as a model coefficient.
Appendix B

Immersion of the 3D jet in the computational mesh

B.1 Cell/Ghost-node classification in the 3D film-cooling box

The aim here is to show how the ghost-nodes \( X_p \) and their corresponding surface points \( X_s \) are searched and classified.

B.1.1 3D film-cooling box

\textit{A priori}, the search of the characteristic \( C^1 \) of cells and the related ghost-nodes \( X_p \) and surface points \( X_s \) should be carried out upon all the grid nodes pertaining to the global computational mesh. However the vast majority of grid nodes are certainly not located in the near-hole region. Hence, in order to not waist computational resources, a \textit{3D film-cooling box} is extracted from the global computational domain, as shown in Fig. B.1. This three-dimensional box represents the near-

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{3D_film-cooling_box.png}
\caption{Three-dimensional film-cooling box.}
\end{figure}

\footnote{The characteristic \( C \) of a cell is either physical, interfacial or ghost, see Fig. 4.5}
B.1. CELL/OST-NODE CLASSIFICATION IN THE 3D FILM-COOING BOX

hole computational domain, so-called $B_{fcm}$. Any cell that is not included in the 3D film-cooling box is considered as being a physical cell. Typically, the 3D film-cooling box has a streamwise length of $3 - 4$ hole diameters, a width of $2 - 3$ hole diameters and a height of $1 - 2$ hole diameters. The searching and classification of the cells, grid nodes and points are only performed inside this local computational mesh $B_{fcm}$.

B.1.2 Cell/Ghost-node classification

The ghost-nodes $X_p$ are necessarily located inside the jet displacement body. The jet body is defined by the interior of the toroidal shape, upstream of the plane of injection, as shown in Fig. B.2. The toroidal shape mimicking the jet near the hole is defined by the function given in Eq. A.5,

$$T(X) = 0$$

Furthermore the plane of injection is defined by the function $P(X)$ when it is equal to zero.

$$P(X) = \chi = 0$$

Thus, the ghost-nodes $X_p$, which are located in the interior of the near-hole jet body, formally reads

$$X_p \in [T(X)]_{T \leq 0} \cap [P(X)]_{P \leq 0} \forall X \in B_{fcm}$$
B.2. BOOLEAN FUNCTION AND SURFACE POINT OF THE PLANE OF INJECTION

Although it is a necessary condition for $X_p$, it is not sufficient. Indeed, the ghost-node must also pertain to one interfacial cell at least, see Fig. 4.5. The classification of all cells pertaining to the 3D film-cooling box is therefore required. To do so, each grid node $X$ is characterized by two boolean functions, as shown in Fig. B.2. The first boolean function is $T(X)$, which tells if the grid node is located outside ($T(X) = 0$) or inside ($T(X) = 1$) of the toroidal shape. In analogy, the boolean function $P(X)$ determines if the grid node $X$ is located downstream ($P(X) = 0$) or upstream ($P(X) = 1$) of the plane of injection. Knowing that each cell has $N_v$ vertices\(^2\), its characteristic $C$ can be derived as follows.

$$C = \sum_{n=1}^{N_v} \max \left( P \left( X_n \right), T \left( X_n \right) \right)$$

(B.3)

where the integer number $C$ determines the class of the cell.

$$\begin{cases} 
C = 0 & \iff \text{physical cell} \\
0 < C < N_v & \iff \text{interfacial cell} \\
C = N_v & \iff \text{ghost cell} 
\end{cases}$$

(B.4)

The ghost-nodes $X_p$ are those which satisfy Eq. B.2 and at least one of its adjacent cells is an interfacial cell ($0 < C < N_v$).

B.2 Boolean function and surface point of the plane of injection

It has been shown in the previous section how the ghost-nodes $X_p$, as well as the cells characteristics, are found and classified. However, the boolean function $P(X)^3$, as well as the surface points $X_s$ of the plane of injection are still missing information to be able to numerically immersed the near-hole film-cooling jet model. As shown in Fig. B.2, for any grid node $X$ located within the 3D film-cooling box, the boolean function $P(X)$ is given by

$$P(X) = \text{int} \left\{ \left( \frac{1}{2} \right) \abs \left[ \frac{(X - X_{je}) \cdot e_X}{|X - X_{je}|} - 1 \right] \right\}$$

(B.5)

Furthermore, the surface point $X_s$ of its corresponding grid node $X$ is given by

$$X_s = X_{je} + [(X - X_{je}) \cdot e_X] e_X$$

(B.6)

\(^2\)In this study, all the cells are considered hexahedral so that $N_v = 8$

\(^3\)as well as $T(X)$, see section B.3
B.3 Boolean function and surface point of the toroidal shape

In analogy to the previous section, it is shown here how to find the boolean function $\mathcal{T}(X)$ as well as the surface points $X_s$ of the toroidal shape. This is to fully immerse the three-dimensional film-cooling jet model.

B.3.1 System of equation to be solved

The search of the position of any grid node $X$ relatively to the toroidal surface does not end up with a unique solution since the toroidal surface is a three-dimensional curved surface. In order to find the surface point $X_s$ corresponding to the grid node $X$, there are three informations to be respected.

1. The surface point $X_s$ verifies Eq. A.5.
2. The normal line to the toroidal surface issuing from $X_s$ is also going through the grid node $X$.
3. As several surface points $X_s$ verify 1. and 2., it is added that the selected surface point $X_s$ is the one that gives the shortest distance to the grid node $X$.

The above information can be put into the following system of equations.

\[
\begin{align*}
T(X_s) &= 0 \\
X_s + \lambda n_s &= X
\end{align*}
\]  

(B.7)

where $\lambda$ and $X_s$ are the four unknowns of the system of equations B.7. The normal vector $n_s$ to the toroidal shape at $X_s$ is given by

\[
n_s = \begin{bmatrix}
\frac{\partial T}{\partial x} \\
\frac{\partial T}{\partial y} \\
\frac{\partial T}{\partial z} \\
\frac{\partial T}{\partial \eta}
\end{bmatrix}
\]  

(B.8)

The first order derivatives of $T$ are given in Appendix C. A Newton-Raphson method is used to solve the system of equations B.7. This means that an initial guess $X_s^0$ of the position of the surface point as well as for the coefficient ($\lambda^0$) has to be fed to the Newton-Raphson procedure. Note that the search of the initial guess will determine the boolean function $\mathcal{T}(X)$ as shown here after. Also the initial guess will ensure that the above-cited point 3 is fulfilled.
B.3.2 Toroidal shape initial guess - get the boolean function

In order to get a quick and efficient initial guess, the three-dimensional problem of Eqs. B.7 is reduced to a two-dimensional problem. This subproblem is defined in the plane going through the grid node X and lying onto the cross section of the toroidal shape, as shown in Fig. B.3. The initial guessed surface point $X^0_s$ and coefficient $\lambda^0_s$ are searched in two steps. The first step is to locate a reference surface point $X^r_s$. It is found when solving the following system of equations.

$$\begin{align*}
X^r_s &= X_c + \lambda^r_s (X - X_c) \\
\frac{|X^r_s - X_c|^2}{A^2} + \frac{|X^r_s - X_c|^2}{B^2} - 1 &= 0
\end{align*}$$
(B.9)

where the center $X_c$ of the local ellipse reads

$$X_c = X_{tc} + R_m \frac{X - X_{tc}}{|X - X_{tc}|}$$
(B.10)

and the angle $\theta$ is found using Eq. A.7 to be fed in $A$ and $B$, see Eq. A.6. The solution of Eqs. B.9 is therefore

$$\lambda^r_s =$$
(B.11)

So that $X^r_s$ is found using Eq. B.11 plugged into the first equation of Eqs. B.9. The initial guessed surface point $X^0_s$ is therefore given by solving the following system of equation.

$$\begin{align*}
X^0_s &= X + \lambda^0_s n^r_s \\
\frac{|X^0_s - X_c|^2}{A^2} + \frac{|X^0_s - X_c|^2}{B^2} - 1 &= 0
\end{align*}$$
(B.12)
B.3. BOOLEAN FUNCTION AND SURFACE POINT OF THE TOROIDAL SHAPE

where the normal vector $n^s$ is found by plugging $X^r_s$ into Eq. B.8. The final solution is therefore

$$\begin{align*}
X^0_s &= \lambda^0 \\
\lambda^0 &= \end{align*} \quad (B.13)$$

The boolean function $T(X)$ is simply given by

$$T(X) = \text{int} \left\{ \left( \frac{1}{2} \right) \left[ \frac{\lambda^0}{|\lambda^0|} + 1 \right] \right\} \quad (B.14)$$

B.3.3 Accurate search of the surface point - Newton-Raphson method

The resolution of the system of equations B.7 can rapidly be fastidious and computationally intensive since there are four non-linear equations (among which three are differential) to be solved in a three-dimensional domain. In order to save computational resources, the system of equations B.7 is decoupled to be reduced to one non-linear equation. This equation is to be solved by an iterative process, using a Newton-Raphson method. It is considered that the surface point $X_s$ is known and fixed so that the only unknown is the coefficient $\lambda$. The equation A.5 of the toroidal shape becomes

$$T[X_s(\lambda)] = T(\lambda) = 0 \quad (B.15)$$

where

$$X_s(\lambda) = X - \lambda n_s \quad (B.16)$$

The normal vector $n_s$ is found by plugging explicitly the surface point $X_s$ in Eq. B.8. The procedure to accurately derive the surface point $X_s$ consists of multiple $n$ steps of repetitive Newton-Raphson method to solve Eq. B.16. The Newton-Raphson method applied to Eq. B.16 gives

$$\frac{T(\lambda_n)}{\lambda_n - \lambda_{n+1}} \approx \left( \frac{dT}{d\lambda} \right)_n \Rightarrow \lambda_{n+1} = \lambda_n - \left( \frac{dT}{d\lambda} \right)_n^{-1} T(\lambda_n) \quad (B.17)$$

where the derivative $dT/d\lambda$ is given in Appendix C. Overall, the procedure to get the surface point $X_s$ is given as follows.

1. Get initial guess $X^0_s$. 
2. Set $i = -1$
3. Update $i = i + 1$
4. Set $\lambda^i$ and $n_s^i$ from the knowledge of $X_s^i$.
5. Set $\lambda_0 = \lambda^i$
6. Compute the toroidal function $T(\lambda_0)$ and its derivative $(dT/d\lambda)_0$.
7. Set $n = 0$
8. Update $n = n + 1$
9. Compute the toroidal function $T(\lambda_n)$ and its derivative $(dT/d\lambda)_n$.
10. Compute $\lambda_{n+1}$ using Eq. B.17.
11. If $\lambda_{n+1} - \lambda_n > C_\lambda$ then go back to 8.
12. Update $X_s^i = X - \lambda_{n+1}n_s^{i-1}$
13. If $X_s^i - X_s^{i-1} > C_{X_s}$ then feed back 3.
14. Set finally $X_s = X_s^i$

After some testing, it is proposed to set $C_\lambda = 10^{-10}$ and $C_{X_s} = 10^{-10}$. In average, it takes about $n = 3$ and $i = 4$ iterations to get the final solution $X_s$, provided that the initial guess $X_s^0$ is good enough.

B.4 Numerical treatment of local singularities

B.4.1 Pseudo-mirror point

As shown in Fig. 4.8, the jet toroidal surface is pinching back in its lee side. In certain cases, the search of the mirror point $X_m$ using Eq. 4.7 may not come to an end. Indeed, as it is displayed in Fig. B.4, the mirror point $X_m$ might be located outside the computational domain. Thus, for this local singularity, a pseudo-mirror point $X'_m$ that acts as a standard mirror point has to be defined. To ensure that the pseudo-mirror point $X'_m$ is inside the computational mesh, it is mirrored to the outside in the direction parallel to the neighboring wall. This direction can be formally defined by the lateral unity vector $e_\eta$ of the jet intrinsic
Figure B.4: Pseudo-mirror point in case of mirror point outside of the computational mesh.

frame of reference. This induces that, before searching the pseudo mirror point $X'_m$, a corresponding pseudo wall point $X'_s$ needs to be found. It is easily found by using the procedure to find the reference wall point $X'^r$ shown in section B.3. The normal vector $n_s$ is simply replaced by the unity vector $e_{\eta}$. The pseudo mirror point is therefore found by solving the following equation.

$$X'_m = \lambda (X'_s - X_p) + X'_s \quad (B.18)$$

B.4.2 Corner node

The ghost-nodes that are located near the plane of injection as well as the jet surface might need to be updated to fulfill two different types of immersed boundary condition $Q_{IBC}$, that is the one affected to immerse the jet surface and the one to inject the film-cooling jet model, as shown in Fig. B.5. Such a singular node is denoted a corner node. A blending parameter $f_c$ is used so that the final state vector value $Q^\text{corner}_p$ of the corner node is a mixture of the two state vector value $Q^\text{jet}_p$ and $Q^\text{fcm}_p$ given by the immersed boundary method applied for mimicking the jet surface and for the film-cooling jet model respectively.

$$Q^\text{corner}_p = f_c Q^\text{fcm}_p + (1 - f_c) Q^\text{jet}_p \quad (B.19)$$

At the location of the corner nodes, the jet is in general starting to be as it is described by the film cooling jet model. Therefore, by default, the blending parameter $f_c$ takes a value of 1.
Figure B.5: Ghost node identified as a corner node.
Appendix C

Derivatives of the toroidal shape function

C.1 \((\chi, \xi, \eta)\)-derivatives

\[
\frac{\partial T}{\partial \chi} = 2 \left[ A \frac{\partial A}{\partial \chi} (\eta^2 - B^2) + B \frac{\partial B}{\partial \chi} (\Omega^2 - A^2) + \Omega \frac{\partial \Omega}{\partial \chi} B^2 \right] \quad (C.1)
\]

\[
\frac{\partial T}{\partial \xi} = 2 \left[ A \frac{\partial A}{\partial \xi} (\eta^2 - B^2) + B \frac{\partial B}{\partial \xi} (\Omega^2 - A^2) + \Omega \frac{\partial \Omega}{\partial \xi} B^2 \right] \quad (C.2)
\]

\[
\frac{\partial T}{\partial \eta} = 2\eta A^2 \quad (C.3)
\]

The derivative of the sub-functions of \(T\) are given as follows

\[
\frac{\partial \Omega}{\partial \chi} = \frac{\chi - \chi_0}{\Omega + R_m} \quad (C.4)
\]

\[
\frac{\partial \Omega}{\partial \xi} = \frac{\xi - \xi_0}{\Omega + R_m} \quad (C.5)
\]

\[
\frac{\partial \Omega}{\partial \eta} = 0 \quad (C.6)
\]

\[
\frac{\partial \Lambda}{\partial \chi} = -d\Lambda \sin \left[ \frac{\theta - \theta_1}{\theta_2 - \theta_1} \pi \right] \left( \frac{\partial \theta}{\partial \chi} \right) \left( \frac{\pi}{\theta_2 - \theta_1} \right) \quad (C.7)
\]
C.2. λ-DERIVATIVES

\[ \frac{\partial \Lambda}{\partial \xi} = -d\Lambda \sin \left[ \frac{\theta - \theta_1}{\theta_2 - \theta_1} \pi \right] \left( \frac{\partial \theta}{\partial \xi} \right) \left( \frac{\pi}{\theta_2 - \theta_1} \right) \]  \hspace{1cm} (C.8)

\[ \frac{\partial \Lambda}{\partial \eta} = 0 \] \hspace{1cm} (C.9)

\[ \frac{\partial \theta}{\partial \chi} = -\frac{\chi - \chi_0}{(\Omega + R_m)^2} \] \hspace{1cm} (C.10)

\[ \frac{\partial \theta}{\partial \xi} = \frac{\xi - \xi_0}{(\Omega + R_m)^2} \] \hspace{1cm} (C.11)

\[ \frac{\partial \theta}{\partial \eta} = 0 \] \hspace{1cm} (C.12)

C.2 λ-derivatives

\[ \frac{\partial T}{\partial \lambda} = 2 \left[ A \frac{\partial A}{\partial \lambda} (\eta^2 - B^2) + B \frac{\partial B}{\partial \lambda} (\Omega^2 - A^2) + \Omega \frac{\partial \Omega}{\partial \lambda} B^2 + \eta \frac{\partial \eta}{\partial \lambda} + A^2 \right] \] \hspace{1cm} (C.13)

The derivative of the sub-functions of \( T \) are given as follows

\[ \frac{\partial \Omega}{\partial \lambda} = \frac{(\chi - \chi_0) n_\chi + (\xi - \xi_0) n_\xi}{\Omega + R_m} \] \hspace{1cm} (C.14)

\[ \frac{\partial \Lambda}{\partial \lambda} = -d\Lambda \sin \left[ \frac{\theta - \theta_1}{\theta_2 - \theta_1} \pi \right] \left( \frac{\partial \theta}{\partial \lambda} \right) \left( \frac{\pi}{\theta_2 - \theta_1} \right) \] \hspace{1cm} (C.15)

\[ \frac{\partial \theta}{\partial \lambda} = \frac{(\chi - \chi_0) n_\xi + (\xi - \xi_0) n_\chi}{(\Omega + R_m)^2} \] \hspace{1cm} (C.16)
Appendix D

Calibrated film-cooling jet model versus experiment: velocity field

The three-dimensional velocity field obtained with the calibrated film-cooling jet model on the plane of injection is shown. It is compared to the available PIV experimental measurements [13] for all flow regimes investigated, with a hole stream-wise injection angle $\alpha_0$ of 30° and 50°.

$\alpha_0 = 30^\circ$  \  $DR = 1.0$  \  $BR = 1.0$  \  $IR = 1.0$  \  $X/d = 1.25$
\[ \alpha_0 = 30^\circ \quad DR = 1.0 \quad BR = 2.0 \quad IR = 4.0 \quad X/d = 1.25 \]

\[ \alpha_0 = 30^\circ \quad DR = 1.0 \quad BR = 2.7 \quad IR = 7.3 \quad X/d = 1.25 \]
\[ \alpha_0 = 30^\circ \quad DR = 1.3 \quad BR = 2.0 \quad IR = 3.2 \quad X/d = 1.25 \]

\[ \alpha_0 = 30^\circ \quad DR = 1.5 \quad BR = 1.0 \quad IR = 0.7 \quad X/d = 1.25 \]
\( \alpha_0 = 30^\circ \) \( \text{DR} = 1.6 \) \( \text{BR} = 2.0 \) \( \text{IR} = 2.5 \) \( \text{X/d} = 1.25 \)

**EXPERIMENT**

**FILM-COOLING JET MODEL**

\( \alpha_0 = 30^\circ \) \( \text{DR} = 1.6 \) \( \text{BR} = 2.7 \) \( \text{IR} = 4.6 \) \( \text{X/d} = 1.25 \)
\( \alpha_0 = 50^\circ \) \( \text{DR} = 1.0 \) \( \text{BR} = 1.0 \) \( \text{IR} = 1.0 \) \( X/d = 1.00 \)

**EXPERIMENT**

**FILM-COOLING JET MODEL**

\( \alpha_0 = 50^\circ \) \( \text{DR} = 2.0 \) \( \text{BR} = 2.0 \) \( \text{IR} = 4.0 \) \( X/d = 1.00 \)

**EXPERIMENT**

**FILM-COOLING JET MODEL**
$\alpha_0 = 50^\circ \quad DR = 1.5 \quad BR = 1.0 \quad IR = 0.7 \quad X/d = 1.00$

EXPERIMENT

FILM-COOLING JET MODEL

$\alpha_0 = 50^\circ \quad DR = 1.5 \quad BR = 2.0 \quad IR = 2.6 \quad X/d = 1.00$
Appendix E

Calibrated film-cooling jet model versus experiment: thermal field

The test rig of Rydholm [75] consists of a closed-loop, recirculating wind tunnel. From a secondary flow loop, cool air is injected into the hot mainstream through a row of five holes, each of them having a length of 8.0 hole diameters. A pitch distance of 3.0 hole diameters separates them. Each hole has a diameter of 5 [mm], and a streamwise injection angle $\alpha_0$ of 30°. Four measurement series were carried out for different density ratios and blowing ratios. In each series, the test rig pressure was at atmospheric level; with a freestream velocity of 47.4 [m/s]. Velocity measurements were made with a Laser-Doppler Anemometer (LDA). The uncertainties can be estimated equal to 0.5 [m/s] for streamwise and lateral velocity and 1 [m/s] for vertical velocity. The total temperature was measured by an iron/constantan thermocouple whose global uncertainty is 0.5 [°C]. A comparison of the velocity and thermal field obtained with the measurements and with the calibrated film-cooling model, for a density ratio of 1.17 and a blowing ratio of 1.35 is shown in Fig. E.1. The plane of injection is located at $X/d = 1.0$. 
Figure E.1: Comparison of the measured (left) [75] and modeled (right) coolant jet flow field on the plane of injection.
Appendix F

Numerical values of the model coefficients for $\alpha_0 = 30^\circ$

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>Functional dependency</th>
<th>$c_0$</th>
<th>$c$</th>
<th>$n$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_{n_a}$</td>
<td>power law: $c_0 + c \cdot I R^n$</td>
<td>40.000</td>
<td>0.0191</td>
<td>3.3570</td>
</tr>
<tr>
<td>$C_{n_b}$</td>
<td>power law: $c_0 + c \cdot I R^n$</td>
<td>5.3245</td>
<td>0.2625</td>
<td>0.2673</td>
</tr>
<tr>
<td>$A_{m1}$</td>
<td>constant: $c_0$</td>
<td>0.3200</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$A_{m2}$</td>
<td>constant: $c_0$</td>
<td>0.2600</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$B_{m1} = B_{m2}$</td>
<td>constant: $c_0$</td>
<td>0.4400</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$\xi_w$</td>
<td>constant: $c_0$</td>
<td>-0.1600</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$\eta_w$</td>
<td>constant: $c_0$</td>
<td>0.0000</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$A_{w1}$</td>
<td>linear: $c_0 + c \cdot I R$</td>
<td>0.1800</td>
<td>0.0200</td>
<td>-</td>
</tr>
<tr>
<td>$A_{w2}$</td>
<td>linear: $c_0 + c \cdot I R$</td>
<td>0.4400</td>
<td>0.0500</td>
<td>-</td>
</tr>
<tr>
<td>$B_{w1} = B_{w2}$</td>
<td>linear: $c_0 + c \cdot I R$</td>
<td>0.3933</td>
<td>0.0167</td>
<td>-</td>
</tr>
<tr>
<td>$I_{w1}$</td>
<td>power law: $c_0 + c \cdot I R^n$</td>
<td>0.0000</td>
<td>-0.1600</td>
<td>0.2618</td>
</tr>
<tr>
<td>$I_{w2}$</td>
<td>constant: $c_0$</td>
<td>-0.0100</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$\xi_{1r}$</td>
<td>power law: $c_0 + c \cdot I R^n$</td>
<td>0.0000</td>
<td>-0.1400</td>
<td>0.3260</td>
</tr>
<tr>
<td>$\eta_{1r}$</td>
<td>constant: $c_0$</td>
<td>0.4800</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$\Gamma_{1r}$</td>
<td>power law: $c_0 + c \cdot I R^n$</td>
<td>0.0000</td>
<td>0.6375</td>
<td>-0.5438</td>
</tr>
<tr>
<td>$a_{1r}$</td>
<td>power law: $c_0 + c \cdot I R^n$</td>
<td>0.0000</td>
<td>0.2960</td>
<td>0.0562</td>
</tr>
</tbody>
</table>

Table F.1: Calibrated functional dependencies of the model coefficient for $\alpha_0 = 30^\circ$. 
Appendix G

Numerical values of the model coefficients for \( \alpha_0 = 50^\circ \)

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>Functional dependency</th>
<th>( c_0 )</th>
<th>( c )</th>
<th>( n )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_{n_a} )</td>
<td>power law: ( c_0 + c \cdot I R^n )</td>
<td>42.000</td>
<td>1.000</td>
<td>1.5000</td>
</tr>
<tr>
<td>( C_{n_s} )</td>
<td>power law: ( c_0 + c \cdot I R^n )</td>
<td>5.3245</td>
<td>0.2625</td>
<td>0.2673</td>
</tr>
<tr>
<td>( A_{m1} )</td>
<td>constant: ( c_0 )</td>
<td>0.3200</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>( A_{m2} )</td>
<td>constant: ( c_0 )</td>
<td>0.2600</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>( B_{m1} = B_{m2} )</td>
<td>constant: ( c_0 )</td>
<td>0.4400</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>( \xi_w )</td>
<td>constant: ( c_0 )</td>
<td>-0.1200</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>( \eta_w )</td>
<td>constant: ( c_0 )</td>
<td>0.0000</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>( A_{w1} )</td>
<td>linear: ( c_0 + c \cdot I R )</td>
<td>0.0200</td>
<td>0.4000</td>
<td>-</td>
</tr>
<tr>
<td>( A_{w2} )</td>
<td>linear: ( c_0 + c \cdot I R )</td>
<td>0.0667</td>
<td>0.3333</td>
<td>-</td>
</tr>
<tr>
<td>( B_{w1} = B_{w2} )</td>
<td>linear: ( c_0 + c \cdot I R )</td>
<td>0.0200</td>
<td>0.4900</td>
<td>-</td>
</tr>
<tr>
<td>( \Gamma_{w}^u )</td>
<td>power law: ( c_0 + c \cdot I R^n )</td>
<td>0.0000</td>
<td>-0.2340</td>
<td>0.1548</td>
</tr>
<tr>
<td>( \Gamma_{w}^r )</td>
<td>constant: ( c_0 )</td>
<td>-0.02</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>( \xi_{1r} )</td>
<td>power law: ( c_0 + c \cdot I R^n )</td>
<td>0.0000</td>
<td>-0.1800</td>
<td>0.2653</td>
</tr>
<tr>
<td>( \eta_{1r} )</td>
<td>constant: ( c_0 )</td>
<td>0.4800</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>( \Gamma_{1r}^u )</td>
<td>power law: ( c_0 + c \cdot I R^n )</td>
<td>0.0000</td>
<td>0.7594</td>
<td>-0.5272</td>
</tr>
<tr>
<td>( a_{1r} )</td>
<td>power law: ( c_0 + c \cdot I R^n )</td>
<td>0.0000</td>
<td>0.2960</td>
<td>0.0562</td>
</tr>
</tbody>
</table>

Table G.1: Calibrated functional dependencies of the model coefficient for \( \alpha_0 = 50^\circ \).
Appendix H

Numerical values of the model coefficients for the small ellipse of the toroidal shape

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>Functional dependency</th>
<th>$c_0$</th>
<th>$c$</th>
<th>$n$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_1$</td>
<td>power law: $c_0 + c \cdot IR^n$</td>
<td>0.1200</td>
<td>0.2000</td>
<td>0.0213</td>
</tr>
<tr>
<td>$A_{2\text{jet}}$</td>
<td>power law: $c_0 + c \cdot IR^n$</td>
<td>1.8000</td>
<td>-0.5380</td>
<td>0.0289</td>
</tr>
<tr>
<td>$B_{2\text{jet}}$</td>
<td>power law: $c_0 + c \cdot IR^n$</td>
<td>0.5000</td>
<td>0.0900</td>
<td>0.0760</td>
</tr>
</tbody>
</table>

Table H.1: Calibrated functional dependencies of the coefficients of the small ellipse of the toroidal shape, for $\alpha_0 = 30^\circ$.

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>Functional dependency</th>
<th>$c_0$</th>
<th>$c$</th>
<th>$n$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_1$</td>
<td>power law: $c_0 + c \cdot IR^n$</td>
<td>0.1200</td>
<td>0.1960</td>
<td>0.0566</td>
</tr>
<tr>
<td>$A_{2\text{jet}}$</td>
<td>power law: $c_0 + c \cdot IR^n$</td>
<td>1.8000</td>
<td>-0.5380</td>
<td>0.0289</td>
</tr>
<tr>
<td>$B_{2\text{jet}}$</td>
<td>power law: $c_0 + c \cdot IR^n$</td>
<td>0.5000</td>
<td>0.0800</td>
<td>0.2925</td>
</tr>
</tbody>
</table>

Table H.2: Calibrated functional dependencies of the coefficients of the small ellipse of the toroidal shape, for $\alpha_0 = 50^\circ$. 
Appendix I

Determination of the stretched wall coordinate

The equations shown in this Appendix are given in Schlichting [80]. The first grid node normal position $Z$ from the flat plate surface is found knowing that the stretched wall coordinate $Z^+$ has to be equal to 1. This is due to the fact that a two-layer turbulence model is used in this study.

$$Z^+ = 1 \quad (I.1)$$

The stretched wall coordinate $Z^+$ reads

$$Z^+ = \frac{\rho^f u_T Z}{\mu^f} \quad (I.2)$$

where $u_T$ is the wall friction velocity. It is given by

$$u_T = \sqrt{\frac{\tau_w}{\rho^f}} \quad (I.3)$$

where $\tau_w$ is the wall stress. It is proportional to the freestream kinetic energy.

$$\tau_w = C_f \frac{\rho^f u_T^2}{2} \quad (I.4)$$

where $C_f$ is the skin-friction coefficient. It is assumed that the flow through the rectilinear channel is fully turbulent. The Blasius solution for such a flow over a flat plate gives the relation for the skin-friction coefficient $C_f$.

$$C_f = 0.058 Re_L^{-\frac{1}{5}} \quad (I.5)$$

Plugging Eqs. I.5, I.4 and I.3 into Eq. I.1 gets

$$Z^+ = 0.2408 \frac{\rho^f u_T Z}{\mu^f} Re_L^{-\frac{1}{10}} \quad (I.6)$$
So that the $Z$ coordinate of the first grid node of the mesh in the normal direction is known, using Eq.
1.1 into Eq. 1.7.

$$Z = 4.1523 \frac{\mu^{f}}{\rho^{f} u^{f}} \text{Re}_{L}^{1/11}$$

(I.7)
Appendix J

Wall Injection (WI) injection strategy

The Wall Injection (WI) strategy consists on the following steps to be carried out.

- Select the hole exit grid nodes that pertain to the hole exit area.

- Compute the state variables at hole exit grid nodes, as a function of the specified coolant conditions.

- Specify in the CFD algorithm the coolant state variables.

Fig. J.1 displays the selected hole exit grid nodes that pertains to the hole exit area. This is shown for the mesh utilized in Chapter 7, that is mesh C2. The coolant state variables are computed at each hole exit grid nodes. They have a constant value at each hole exit grid nodes so that they only need to be computed once. In this work, the technique utilized to probe the surrounding flow field in the FCM strategy is
also used for the WI strategy. Furthermore, the computation of the coolant flow conditions at hole exit is done as for the FCM strategy. It is based on the isentropic expansion of the coolant through the hole, see section 3.4.1 in Chapter 3. This is to ensure that the flow values used for the FCM and WI strategies are as close as possible. Vogel [89] used other inputs but the strategy of the injection stays the same. Here, the coolant state variables are specified as follows:

- \( \rho = \rho^e \)
- \( \rho u_x = \rho^e U^e \cos(\alpha_0) \cos(\beta_0) \)
- \( \rho u_y = \rho^e U^e \cos(\alpha_0) \sin(\beta_0) \)
- \( \rho u_z = \rho^e U^e \sin(\alpha_0) \)
- \( \rho E = (P_s / [\gamma - 1]) + 0.5 \rho^e U^e^2 \)
Appendix K

Full Injection (FI) injection strategy

The Full Injection (FI) strategy consists of meshing not only the freestream region but also the hole and the plenum chamber, where the coolant is delivered. Several authors have already used this strategy to investigate film-cooling jets in a flat plate, see for instance Leylek and Zerkle [56], Walters and Leylek [90], Lakehal [52], Martelli and Adami [60]. The meshing of the hole and plenum chamber is obviously more tedious than meshing only the freestream region. Thus, the effort when using this method is not only increased by the higher amount of computational cells, comparatively to the FCM or WI strategies, but also due to the pre-processing of the computational domain. In this thesis, the commercial software CFX5.7 has been used for the Full Injection (FI) strategy. A very detailed explanation of the procedure to carry out the computation shown in this thesis can be found in Christen [20].

K.1 Computational domain

The computational domain used for the FI strategy corresponds to the geometry of the film-cooled flat plate test section of Berndorf [13]. The dimensions of the computational domain are plotted in Fig. K.1. The dimensions of the plenum chamber are identical to the experimental setup. The coolant is delivered at the bottom of the plenum chamber.

K.2 Computational mesh

The computational mesh consists of a combination of tetrahedral and hexahedral cells. This hybrid approach has been chosen because it allows to lower the number of grid nodes in the freestream region (tetrahedral cells) while being well refined and orthogonal near the walls (hexahedral cells). This hybrid grid structure is fully supported by the solver CFX5.7. A center plane cut of the computational mesh is
shown in Fig. K.2. As it can be seen in Fig. K.2, the mesh is refined in the near hole region and downstream of it. Indeed, this is where the coolant and freestream flows interact. Since a two-layer turbulence model is used, the hexahedral cells near the walls have been ensured to be enough clustered so as to have a non-dimensional wall distance of $Z^+ \simeq 1$. There are typically about 10 hexahedral cells in the laminar near wall region and buffer zone. Then, tetrahedral cells are placed. More than 15 cells are lying within the boundary layer in the freestream region. In total, the computational mesh contains $7.7 \cdot 10^5$ grid nodes. This is a typical grid size for this kind of flow problem, see for instance Martelli and Adami [60]. During the meshing process, one of the most complicated task is the connection of the grid blocks pertaining to the hole to the ones that are in the plenum chamber and in the freestream region. To help the user, CFX5.7 has a optional tool to make sure that all blocks are well connected. The mesh near the hole exit is plotted in Fig. K.3. It reveals the changes in grid density near the corners of the hole. This is a

Figure K.1: Dimensions of the computational domain used in the FI strategy, in [mm].
K.3 Boundary conditions

The same boundary condition as for the FCM and WI strategies are given at the freestream inlet plane and at the outlet plane. At the freestream inlet plane, the flow angles are given (velocity perpendicular to the plane) as well as the total pressure and temperature of the freestream, see Table 5.2 in Chapter 5. There is no boundary height. Indeed, the boundary layer starts developing at the inlet of the computational domain, as in the experiment. At the outlet plane, a constant static pressure is specified so as to match the freestream Mach number, see Table 5.2. The specification at the inlet coolant plane is a bit more tricky. Indeed, after some computational runs, it has been discovered that the prediction of losses through the hole is inaccurate, see Christen [20]. This means that if the coolant total pressure is specified in the computation exactly as it is in the experiment, then the computed
K.4. COMPUTATIONAL PROCEDURE

The Reynolds-Averaged Navier Stokes (RANS) equations are solved. The eddy viscousity is computed through the $k-\omega$ based Shear-Stress Transport (SST) model, see Menter [63]. This is a two-equations, isotropic turbulence model. It basically uses the standard $k-\omega$ model in the boundary layer region and switches to the standard $k-\epsilon$ model in the freestream region. The computational procedure consists of two parts. In a first step, a computation is carried out with a blowing ratio
K.4. COMPUTATIONAL PROCEDURE

of $BR = 1.0$ in order to have an initial flow solution that can serve for different blowing regimes. Then, the computation with the blowing regime wanted is carried out. The Root Mean Square (RMS) residuals of the initial guess computation, as well as the final computation, are shown in Fig. K.4. In total, about 200 iterations are needed to get a fully converged solution. The computations have been done on the same hardware as for the FCM and WI strategies, see Chapter 5. Each iteration takes about 7.4 minutes. This means that the whole computational time is 24 hours and 40 minutes.

![Figure K.4: Root Mean Square (RMS) residuals for the initial guess computation (left) and final computation (right).](image-url)
Appendix L

Linear transonic film-cooled turbine cascade

L.1 Injected coolant mass flow

<table>
<thead>
<tr>
<th>Case</th>
<th>BR = 1.0</th>
<th>BR = 1.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>set 3</td>
<td>0.15</td>
<td>0.23</td>
</tr>
<tr>
<td>set 4a</td>
<td>0.21</td>
<td>0.31</td>
</tr>
<tr>
<td>set 1</td>
<td>0.37</td>
<td>0.50</td>
</tr>
<tr>
<td>set 4ab</td>
<td>0.42</td>
<td>0.61</td>
</tr>
<tr>
<td>set 13</td>
<td>0.52</td>
<td>0.73</td>
</tr>
<tr>
<td>set 14a</td>
<td>0.57</td>
<td>0.81</td>
</tr>
<tr>
<td>set 134a</td>
<td>0.73</td>
<td>1.04</td>
</tr>
<tr>
<td>set 14ab</td>
<td>0.79</td>
<td>1.12</td>
</tr>
<tr>
<td>set 134ab</td>
<td>0.94</td>
<td>1.35</td>
</tr>
</tbody>
</table>

Table L.1: Percentage of injected coolant mass flow as a function of the incoming freestream mass flow.
L.2 Global mass flow error

<table>
<thead>
<tr>
<th>Case</th>
<th>BR = 1.0</th>
<th>BR = 1.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>uncooled</td>
<td>-0.13</td>
<td>0.13</td>
</tr>
<tr>
<td>set 3</td>
<td>-0.24</td>
<td>-0.15</td>
</tr>
<tr>
<td>set 4a</td>
<td>-0.15</td>
<td>-0.04</td>
</tr>
<tr>
<td>set 1</td>
<td>-0.06</td>
<td>0.01</td>
</tr>
<tr>
<td>set 4ab</td>
<td>-0.22</td>
<td>-0.05</td>
</tr>
<tr>
<td>set 13</td>
<td>-0.05</td>
<td>-0.01</td>
</tr>
<tr>
<td>set 14a</td>
<td>-0.26</td>
<td>0.06</td>
</tr>
<tr>
<td>set 134a</td>
<td>-0.02</td>
<td>0.12</td>
</tr>
<tr>
<td>set 14ab</td>
<td>-0.05</td>
<td>0.14</td>
</tr>
<tr>
<td>set 134ab</td>
<td>-0.06</td>
<td>0.22</td>
</tr>
</tbody>
</table>

Table L.2: Global mass flow error $\epsilon_m^G$. 
L.3 Definition of loss coefficients

The pressure loss coefficient $Y_p$ is computed as follows (Denton [26]):

$$Y_p = \frac{P_{T,inl} - P_{T,out}}{P_{T,out} - P_{s,out}}$$  \hspace{1cm} (L.1)

where the pressures are mass-averaged at the inlet and outlet planes respectively. The outlet total and static pressure $P_{T,out}$ and $P_{s,out}$ represent the mixed-out pressure condition, given by the computation (or experiment). The inlet total pressure $P_{T,inl}$ is found by mass-averaging the inlet freestream and coolant total pressures $P_{f,inl}^T$ and $P_{c,inl}^T$.

$$P_{T,inl} = \frac{\dot{m}_{inl}^f P_{T,inl}^f + \sum_{i=1}^{n} \dot{m}_{inl}^{ci} P_{T,inl}^{ci}}{\dot{m}_{inl}^f + \sum_{i=1}^{n} \dot{m}_{inl}^{ci}}$$  \hspace{1cm} (L.2)

where $n$ is the total number of cooling holes. The enthalpy loss coefficient $Y_H$ is defined as follows (Denton [26]):

$$Y_H = \frac{H_{s,out} - H_{s,out}^{is}}{H_{out} - H_{s,out}}$$  \hspace{1cm} (L.3)

where $H_{s,out}^{is}$ is the mixed-out, isentropic expansion, static enthalpy. The mixed-out total and static enthalpy ($H_{out}$ and $H_{s,out}$) are mass-averaged at the outlet plane. The mixed-out, isentropic expansion, static enthalpy $H_{s,out}^{is}$ is found from the mass-averaged inlet total pressure and total temperature $P_{T,inl}^f$ and $T_{T,inl}$. The mass-averaged inlet total temperature $T_{T,inl}$ is given by

$$T_{T,inl} = \frac{\dot{m}_{inl}^f T_{T,inl}^f + \sum_{i=1}^{n} \dot{m}_{inl}^{ci} T_{T,inl}^{ci}}{\dot{m}_{inl}^f + \sum_{i=1}^{n} \dot{m}_{inl}^{ci}}$$  \hspace{1cm} (L.4)

The mixed-out, isentropic expansion, static enthalpy $H_{s,out}^{is}$ is therefore given by

$$H_{s,out}^{is} = c_p T_{T,inl} \left( \frac{P_{s,out}}{P_{T,inl}} \right)^{\frac{\gamma - 1}{\gamma}}$$  \hspace{1cm} (L.5)
L.4. Superposition of rows at pressure side surface with set 3 and set 4a

Figure L.1: Predicted adiabatic film-cooling effectiveness $\eta$ at the pressure side surface for a blowing ratio of $BR = 1.0$. Injection at set 3 and set 4a (left) and injection at set 3 only (right).
L.4. SUPERPOSITION OF ROWS AT PRESSURE SIDE SURFACE WITH SET 3 AND SET 4A

![Graphs showing predicted adiabatic film-cooling effectiveness](image)

Figure L.2: Predicted adiabatic film-cooling effectiveness $\eta$ at the pressure side surface for a blowing ratio of $BR = 1.5$. Injection at set 3 and set 4a (left) and injection at set 3 only superposed with injection at set 4a only (right).
L.5. SUPERPOSITION OF ROWS AT PRESSURE SIDE SURFACE WITH SET 3 AND SET 4AB

Superposition of rows at pressure side surface with set 3 and set 4ab

Figure L.3: Predicted adiabatic film-cooling effectiveness $\eta$ at the pressure side surface for a blowing ratio of $BR = 1.0$. Injection at set 3 and set 4ab (left) and Injection at set 3 only superposed with injection at set 4ab only (right).
Figure L.4: Predicted adiabatic film-cooling effectiveness $\eta$ at the pressure side surface for a blowing ratio of $BR = 1.5$. Injection at set 3 and set 4ab (left) and injection at set 3 only superposed with injection at set 4ab only (right).
Curriculum Vitae

Personal data
Name               André Burdet
Birth              January 19th, 1977, Lausanne, Switzerland
Citizenship       Swiss and French

Education
1995               French Baccalauréat with honors, Lycée Anna de Noailles, Evian, France
1995 - 2000        Mechanical engineering studies, Swiss Federal Institute of Technology - Lausanne (EPFL), Switzerland
1997 - 1998        Exchange student at Carnegie-Mellon University (CMU), Pittsburgh, USA
2000 - 2005        Doctoral candidate, Swiss Federal Institute of Technology - Zürich (ETHZ), Switzerland

Professional Experience

Additional Experience
2000 - 2004        Teaching assistant and supervisor of student semester and diploma thesis, Swiss Federal Institute of Technology - Zürich (ETHZ), Switzerland
2004 - 2005        Reviewer for scientific papers on turbomachinery, Swiss Federal Institute of Technology - Zürich (ETHZ), Switzerland