Numerical and experimental study of the Shock-Boundary Layer Interaction in Transonic Unsteady Flow

Olivier BRON

Doctoral Thesis
2004

Department of Energy Technology
Division of Heat and Power Technology
Royal Institute of Technology
SE-100 44 Stockholm, Sweden
Akademisk avhandling som med tillstånd av Kungliga Tekniska Högskolan i Stockholm framlägges till offentlig granskning för avläggande av teknisk doktorsexam i EnergiTeknik fredagen den 23 januari 2004 kl 10.00 i sal B3, Kungliga Tekniska Högskolan, Brinellvägen 60, S 100 44, Stockholm, Sverige.

© Olivier BRON, 2004

Universitetservice US AB, Stockholm, 2004
A mes parents, Serge et Ilda...
Abstract

A prerequisite for aeroelastic stability prediction in turbomachines is the understanding of the fluctuating aerodynamic forces acting on the blades. Unsteady transonic flows are complex because of mutual interactions between travelling pressure waves, outlet disturbances, shock motion, and fluctuating turbulent boundary layers. Complex phenomena appear in the shock/boundary layer region and produce phase lags and high time harmonics, which can give a significant contribution to the overall unsteady lift and torque, and therefore affect flutter boundaries, cause large local stresses, or even severely damage the turbomachine.

The present research work is concerned with the understanding of phenomena associated with travelling waves in non-uniform transonic flows and how they affect the unsteady pressure distribution on the surface as well as the far field radiated sound. In similitude with turbomachines potential interaction, the emphasis was put on the unsteady interaction of upstream propagating acoustic waves with an oscillating shock in 2D and 3D nozzle flows. Both numerical and experimental studies are carried out and compared with each other.

Results show that the unsteady pressure distribution, both on the bump surface and within the channel, results from the superposition of upstream and downstream propagating waves. It is believed that outlet pressure perturbations propagate upstream in the nozzle, interact in the high subsonic flow region according to the acoustic blockage theory, and are partly reflected or absorbed by the oscillating shock, depending on the frequency of the perturbations and the intensity of the SBHI. Furthermore the shock motion amplitude is found to be related to the mean flow gradients and local wave length of the perturbations rather than to the shock boundary layer interaction. The phase angle between incoming pressure perturbations and the shock motion increases with the perturbation frequency but also depends on the intensity of the SBHI. Additionally the phase angle "shift" observed underneath the shock location linearly increases with the perturbation frequency and the shock strength. Such phase shift is critical regarding aeroelastic stability and might have a significant impact on the phase angle of the overall aerodynamic force acting on the blade and shift the aerodynamic damping from stable to exciting.

Keywords

Résumé

La prédiction des fluctuations de pression sur les aubages de turbomachines est une condition nécessaire afin d’assurer la stabilité par rapport aux problèmes aéroélastiques. La compréhension des phénomènes instationnaires en turbomachines est une tâche extrêmement délicate, en particulier lorsque l’écoulement devient transsonique. De multiples interactions se créent alors entre les ondes acoustiques, ondes de choc et battements de couches limites. Des phénomènes de couplage complexes apparaissent alors entre l’onde de choc et la couche limite, produisant des amplifications de pression importantes et des déphasages critiques en terme de stabilité. Les vibrations mécaniques qui s’en suivent engendrent des fatigues anticipées et dans certains cas extrêmes la rupture brutale des aubages.

Le présent travail de recherche a pour objectif d’améliorer la compréhension des phénomènes instationnaires liés à la propagation d’ondes acoustiques en écoulement transsonique non-uniforme. Plus particulièrement, en similitude avec les interactions potentielles en turbomachine, l’étude porte sur l’interaction entre perturbations de pression aval et onde de choc en tuyère 2D et 3D. Les aspects numériques et expérimentaux ont tous deux été considéré afin d’obtenir, par le biais d’une comparaison viable, une validation des résultats.

Les résultats obtenus montrent que la distribution instationnaire de pression, aussi bien sur la paroi qu’au centre de la veine d’essai, correspond à la superposition d’ondes incidences et rétrogrades. Les perturbations de pression aval se propagent dans à l’amont de la tuyère, jusque dans les régions d’écoulement haut subsonique où s’opère le phénomène de blockage acoustique, puis sont partiellement réfléchies ou absorbées par l’onde de choc, en fonction de la fréquence des perturbations de pression et de l’intensité de l’interaction onde de choc-couche limite. D’autre part, il est montré que l’amplitude d’oscillation de l’onde de choc dépend plus des gradient de pression moyens et de la longueur d’onde des perturbations que de l’interaction avec la couche limite. D’un point de vue général, le déphasage entre les perturbations de pression incidentes et le mouvement de l’onde de choc augmente avec la fréquence des perturbations mais également dépend de l’intensité de l’interaction onde de choc-couche limite. De plus, le saut de phase observé sous la zone d’interaction augmente linéairement avec la fréquence des perturbations et l’intensité du saut de pression à travers l’onde de choc. Un tel changement de phase de la distribution de pression instationnaire se traduire par une modification importante du moment résultant de la force aérodynamique sur la structure qui peut éventuellement se répercuter sur les limites de stabilité et conduire à des problèmes de type aéroélastiques.
Sammanfattning

En förutsättning för uppskattning av aeroelastisk stabilitet är förståelsen av den fluktuerande aerodynamiska kraften som verkar på bladprofilen. Instationära transoniska strömmar är komplexa eftersom det finns gemensam växelverkan mellan utbredande tryckvågor, utloppsstörningar, stötvägsoscillationer och fluktuerande turbulenta gränsskikt. Komplexa fenomenen uppträder i interaktionsområdet mellan stötväg och gränsskikt (ISG) och genererar fasförskjutning och högfrekventa harmoniska svängningar. Detta kan ge ett betydelsefullt bidrag till den totala lyftkraften och momentet och därmed påverka fladdernas gränserna, orsaka höga lokala spänningar eller till och med skada turbomaskinen allvarligt.

Det följande doktorandarbetet rör förståelsen av utbredande tryckvågor i inhomogena transoniska strömmar och hur de påverkar den instationära tryckfördelningen på ytan. I likhet med potentialinteraktion i turbomaskiner är betonningen lagd på instationär växelverkan mellan uppströms utbredande ljudvågor och en vibrerande stötväg i ett 2D och 3D myntstycke. Både experimentella och numeriska studier undersöks och jämförs med varandra. Resultat visar att den instationära tryckfördelningen, både på ytan och i kanalen, resulterar från superposition av upp- och nedströms utbredande tryckvågor. Det misstänks att utloppsstörningar fortpolar sig uppströms och växelverkar i de högsubsoniska strömningssunderlagda områden i enlighet med ”acoustic blockage” teorin, och partiellt reflekteras eller absorberas av den oscillerande stötvägen beroende på störningsfrekvens och intensitet av ISG. Stötvågens utbredningsamplitude fanns vara besläktad med huvudströmmens gränder och tryckstörningssväglangden snarare än ISG. Fasvinkel mellan tryckstörningarna och stötvägen ökar med frekvensen men beror också på intensiteten av ISG. Dessutom observeras att fasvinkelnänvändning under stötvägen ökar linjärt med störningsfrekvensen och styrkan av stötvägen. Sådan fasvinkelnändring är kritisk beträffande aeroelastisk stabilitet och kan ha en betydelsefull inverkan på den aerodynamiska kraften vilket skulle ändra den aerodynamiska dämpningen från stabilt till ostabilt.
Acknowledgments

I first would like to express my gratitude to both my supervisors at KTH and ECL, Professor Torsten Fransson and Pascal Ferrand for their support, patience and trust all along those years.

I also would like to thank Professor Roger Ainsworth from the University of Oxford (UK) and Professor Li He from the University of Durham (UK) for being opponents of this work. Many thanks as well the jury members, Professor A. Böcs from EPFL (Switzerland), J.P. Grisval from ONERA (France) and Professor U. Häll from the Chalmers Technical Highschool (Sweden).

I wish to express a very special thanks to all my colleagues and friends I met during the different working periods at both the Ecole Centrale de Lyon and the Royal Institute of Technology: Laurent, Stephane O., Sabine, Olivier, Guilherme, Fabien O., Xavier, Stanislas, Athem, Fabien G., Stephane A., Joëlle, Jérôme, Wolfgang, Eloi, Samuel, Sebastien, FX, Sven-Hendrik, Damian, Anna, Olga, Yacine, Davy, Catharina, Fridh, Vital, Birute, Nathalie, Thomas, Julien C., Reza, Jiasen, Jeewan, Markus, Alexandros, Björn, Moosa, Mikkel, Nalin, Miroslav, Julien R., Susanne, Kai, Marianne, Arturo, yacine, and Ivan. Thanks to all of you for the motivating working atmosphere, the fruitful discussions and the high quality time I had.

Some special thanks also to the technicians Stellan, Christer, Rolf, Bernt, Jacque and Patrick for their help and precious advices. Thanks for the Swedish lessons too...

I shall not forget the computer support team Tony, Jan, and Birger. Thanks for your help and patience.

I would like to thank the Swedish Energy Agency (STEM), the French ministry of education as well as the Rhône-Alpes region for their financial support, the Parallel Dator Centrum (PDC/KTH) and the Centre Informatique National de l’Enseignement Supérieur (CINES) for granting me access to supercomputers.

Finally, I would like to deeply thank my family, Serge, Ilda and Jessica, for their support and encouragements during my education and most particularly during my PhD.
Contents

Abstract ............................................................... v
Acknowledgment ....................................................... ix
Table of contents ...................................................... xviii
List of figures ......................................................... xxix
List of tables .......................................................... xxxii
List of publications .................................................... xxxiii
Nomenclature ............................................................ xxxv

1 Introduction

2 Background
2.1 Unsteady phenomena in axial turbomachinery flow ................. 5
2.2 Aeroelasticity ....................................................... 6
   2.2.1 General introduction ........................................ 6
   2.2.2 Dynamic aeroelasticity ...................................... 8
      2.2.2.1 Basic description of flutter phenomenon .............. 8
      2.2.2.2 Flutter classification in axial flow turbomachine .... 9
   2.2.3 Forced response .............................................. 11
2.3 Acoustic blockage theory ......................................... 13
2.4 Shock/boundary layer interaction .................................. 15
   2.4.1 General introduction ....................................... 15
   2.4.2 Structure of turbulent boundary layers ...................... 15
      2.4.2.1 Equilibrium turbulent boundary layers ............... 15
      2.4.2.2 Historical effects .................................... 17
      2.4.2.3 APG boundary layer effect ......................... 18
   2.4.3 Shock/turbulence interaction ................................ 20
   2.4.4 Phenomenological description of transonic viscous interaction on airfoils ........................................ 22
   2.4.5 Shock-boundary layer interaction without separation ........ 24
   2.4.6 Incipient shock induced separation in transonic flows .......... 28
   2.4.7 Transonic interaction with boundary-layer separation .......... 30
   2.4.8 Short review on unsteady SBLI in nozzle flow ................ 35
2.5 Conclusion ....................................................... 37

3 Objectives and method of attack .................................. 39
3.1 Objectives ....................................................... 39
3.2 Method of attack ................................................. 39
CONTENTS

4 Numerical Model ........................................ 41
   4.1 Introduction ........................................ 41
   4.2 Governing equations .................................. 41
   4.3 Numerical methods .................................... 43
   4.4 Conclusion ........................................ 47

5 Experimental Model ...................................... 49
   5.1 Introduction ........................................ 49
   5.2 Experimental apparatus ............................... 49
      5.2.1 Air supply wind tunnel facility ................. 49
      5.2.2 Modular test section ............................. 51
      5.2.3 Convergent-divergent nozzles .................... 52
         5.2.3.1 Three-dimensional nozzle .................... 52
         5.2.3.2 Two dimensional nozzle .................... 56
      5.2.4 Perturbation generator ........................... 58
   5.3 Instrumentation ...................................... 60
      5.3.1 Atmospheric conditions ........................... 60
      5.3.2 Wind tunnel instrumentation ...................... 61
         5.3.2.1 Settling chamber ............................. 61
         5.3.2.2 Test section ................................ 62
         5.3.2.3 Air flow quality ............................. 63
      5.3.3 Orifice plate .................................... 65
      5.3.4 Test objects instrumentation ..................... 65
         5.3.4.1 Common pressure instrumentation .............. 65
         5.3.4.2 3D bump instrumentation ...................... 66
         5.3.4.3 2D sliding bump specific instrumentation .... 66
            5.3.4.3.1 Pressure instrumentation ................. 66
            5.3.4.3.2 Hotfilm instrumentation ................. 67
            5.3.4.3.3 Traverse mechanism ....................... 69
            5.3.4.3.4 Side wall sealing method .................. 69
      5.3.4.4 Steady state pressure instrumentation ......... 70
      5.3.4.5 Unsteady flow instrumentation ................. 72
         5.3.4.5.1 High frequency response transducers ....... 72
         5.3.4.5.2 Protection of transducers ................ 73
         5.3.4.5.3 Static calibration of transducers .......... 74
         5.3.4.5.4 Sound propagation related limitations .... 75
      5.3.5 Perturbation generator instrumentation .......... 76
         5.3.5.1 High speed motor instrumentation .............. 76
         5.3.5.2 Electronic counter ............................ 77
         5.3.5.3 Rotating speed accuracy ....................... 77
      5.3.6 Measuring techniques ............................. 79
         5.4.1 Steady state pressure measurements .............. 79
            5.4.1.1 Data acquisition system .................... 79
            5.4.1.2 Data acquisition method .................... 80
               5.4.1.2.1 Two-dimensional sliding test object .... 81
               5.4.1.2.2 Measurement procedure for the 3D test object 82
            5.4.1.3 Data reduction ............................. 83
CONTENTS

6.2 Numerical investigation

6.2.3.2 Schlieren visualization ........................................... 139
6.2.3.2.3 Oil-paint visualizations .................................... 140
6.2.3.2.4 Pressure measurements ..................................... 140

6.3 Numerical investigation

6.3.1 Introduction ........................................................... 154
6.3.2 Computation initialization: Rectangular duct simulation .... 155
6.3.2.1 Computational grid and boundary conditions .............. 155
6.3.2.2 Rectangular duct simulation results ......................... 157
6.3.3 Two dimensional nozzle flow simulations ....................... 158
6.3.3.1 Introduction ....................................................... 158
6.3.3.2 2D RANS computations ....................................... 159
6.3.3.2.1 Computational meshes .................................... 159
6.3.3.2.2 Global flow description ................................... 160
6.3.3.3 3D RANS computations ....................................... 162
6.3.3.3.1 Computational meshes .................................... 162
6.3.3.3.2 Global flow description ................................... 163
6.3.3.4 Experimental-Numerical comparison ......................... 167
6.3.3.4.1 Weak shock configuration case .......................... 167
6.3.3.4.2 Strong shock configuration case ......................... 169
6.3.4 Three dimensional nozzle flow simulation ....................... 170
6.3.4.1 Computational meshes ....................................... 170
6.3.4.2 Global flow description .................................... 171
6.3.4.3 Influence of outlet static pressure .......................... 173
6.3.5 Conclusion ............................................................. 175

7 Unsteady Results and Discussion ........................................ 179

7.1 Introduction ............................................................. 179
7.2 Two-dimensional nozzle .............................................. 179
7.2.1 Experimental results .............................................. 179
7.2.1.1 Introduction ..................................................... 179
7.2.1.2 Analysis of unsteady pressure distribution at mid-channel 180
7.2.1.2.1 Quasi-steady description of the streamwise unsteady pressure repartition ..... 180
7.2.1.2.2 Influence of perturbation frequency ................... 182
7.2.1.2.3 Influence of shock location .............................. 183
7.2.1.2.4 Influence of perturbation amplitude .................... 184
7.2.1.2.5 Concluding remarks ....................................... 185
7.2.1.3 Side wall influence on unsteady pressure distribution .... 185
7.2.1.3.1 Basic quasi-steady influence of spanwise location ..... 185
7.2.1.3.2 Influence of perturbation frequency on spanwise pressure amplification .... 186
7.2.1.3.3 Influence of shock location on spanwise pressure amplification ................. 187
7.2.1.3.4 Influence of perturbation amplitude on spanwise pressure amplification .... 188
7.2.1.3.5 Concluding remarks ....................................... 188
7.2.1.4 Analysis of unsteady shock motion .......................... 189
7.2.1.4.1 Influence of mean flow gradients.................................. 189
7.2.1.4.2 Influence of perturbation frequency............................... 189
7.2.1.4.3 Influence of shock location......................................... 190
7.2.1.4.4 Influence of perturbation amplitude............................... 192
7.2.1.4.5 Concluding remarks.................................................. 193
7.2.1.5 Analysis of shock motion and unsteady pressure perturbation correlation............................... 193
7.2.1.6 Analysis of unsteady pressure phase shift underneath shock............................... 195
7.2.1.7 Analysis of time-averaged versus steady state pressure distribution............................... 196
  7.2.2 Numerical results and comparison with experiments............................... 196
  7.2.2.1 Introduction......................................................... 196
  7.2.2.2 Analysis of unsteady pressure distribution on bump............................... 197
  7.2.2.2.1 Influence of shock location...................................... 197
  7.2.2.2.2 Influence of perturbation frequency............................... 200
  7.2.2.2.3 Comparison with experiments..................................... 201
  7.2.2.2.4 Concluding remarks................................................ 205
  7.2.2.3 Analysis of unsteady pressure distribution in channel............................... 206
  7.2.2.3.1 Influence of shock location...................................... 206
  7.2.2.3.2 Influence of perturbation frequency............................... 208
  7.2.2.3.3 Concluding remarks................................................ 211
  7.2.2.4 Analysis of unsteady shock motion.................................... 212
  7.2.2.4.1 Influence of shock location...................................... 212
  7.2.2.4.2 Influence of perturbation frequency............................... 214
  7.2.2.4.3 Comparison with experiments..................................... 216
  7.2.2.4.4 Concluding remarks................................................ 218
  7.2.2.5 Analysis of shock motion and pressure perturbations correlation............................... 220
  7.2.2.6 Analysis of the Time-Averaged Versus steady state pressure distribution on bump............................... 221
  7.2.2.7 Analysis of the separated region motion................................ 221
  7.2.2.8 Analysis of acoustic propagation in channel............................... 223
  7.3 Three-dimensional Nozzle.................................................. 226
  7.3.1 Experimental results.................................................. 226
  7.3.1.1 Introduction......................................................... 226
  7.3.1.2 Analysis of unsteady pressure distribution through the channel width............................... 226
  7.3.1.3 Analysis of the unsteady pressure phase shift underneath shock........................................... 229
  7.3.1.4 Concluding remarks.................................................. 229
  7.3.2 Numerical results and comparison with experiments............................... 230
  7.3.2.1 Introduction......................................................... 230
  7.3.2.2 Analysis of the unsteady pressure distribution through the channel width............................... 231
  7.3.2.3 Analysis of unsteady pressure distribution in channel............................... 232
  7.3.2.4 Analysis of the unsteady shock motion................................ 233
  7.3.2.5 Concluding remarks.................................................. 236
## CONTENTS

7.4 Conclusion .......................................................... 237

8 Conclusion and perspectives ........................................... 241

Appendices ................................................................. 245

**A Numerical Model** .................................................. 247

A.1 Introduction .......................................................... 247
A.2 Governing equations ................................................ 247
  A.2.1 Instantaneous Navier-Stokes equations in conservative form . 247
  A.2.2 Reynolds-Averaged Navier-Stokes equations .................... 249
A.2.3 Turbulence closure ............................................... 250
  A.2.3.1 Introduction ............................................... 250
  A.2.3.2 Simplification of the RANS ................................ 252
  A.2.3.3 Turbulent variables transport equations ................. 253
A.2.4 Non dimensional formulation ................................... 255
A.2.5 Conclusion ....................................................... 257

A.3 Numerical methods .................................................. 257

A.3.1 Introduction ....................................................... 257
A.3.2 Finite volume formulation ...................................... 257
A.3.3 Time discretization .............................................. 263
  A.3.3.1 Runge-Kutta scheme ....................................... 264
  A.3.3.2 Time step calculation ..................................... 264
  A.3.3.3 Numerical smoothing ...................................... 265
    A.3.3.3.1 Explicit residual smoothing ......................... 265
    A.3.3.3.2 Implicit residual smoothing ....................... 266
A.3.4 Spatial discretization .......................................... 267
  A.3.4.1 Discretization of the convective terms ................. 267
    A.3.4.1.1 Upwind Roe scheme .................................. 268
    A.3.4.1.2 Higher order scheme ............................... 269
    A.3.4.1.3 Spatial accuracy limiters ......................... 270
  A.3.4.2 Discretization of the diffusive terms ................... 271
  A.3.4.3 Discretization of the turbulent equations ............... 272
    A.3.4.3.1 Turbulent convective fluxes ....................... 272
    A.3.4.3.2 Turbulent diffusive fluxes ....................... 273
    A.3.4.3.3 Numerical filters ................................. 273
A.3.5 Boundary discretization ....................................... 274
  A.3.5.1 Extension of the computational domain ................... 275
  A.3.5.2 Geometrical boundary conditions ......................... 276
  A.3.5.3 Physical boundary conditions ............................ 277
    A.3.5.3.1 Mono-directional formulation ...................... 277
    A.3.5.3.2 Different types of physical boundary conditions 278
A.3.6 Parallel computation ............................................ 281
A.4 Conclusion ......................................................... 281

**B Redesigned test section** ......................................... 283

B.1 Design specifications ............................................. 283
B.2 Boundary layer cut off device .................................... 287
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>Settling chamber instrumentation</td>
<td>289</td>
</tr>
<tr>
<td>C.1</td>
<td>Total pressure probe</td>
<td>289</td>
</tr>
<tr>
<td>C.2</td>
<td>Total temperature probe</td>
<td>291</td>
</tr>
<tr>
<td>D</td>
<td>Air flow quality investigation</td>
<td>295</td>
</tr>
<tr>
<td>D.1</td>
<td>Flow quality in settling chamber</td>
<td>295</td>
</tr>
<tr>
<td>D.2</td>
<td>Flow quality in test section</td>
<td>296</td>
</tr>
<tr>
<td>D.3</td>
<td>Measurements repeatability and reproducibility</td>
<td>300</td>
</tr>
<tr>
<td>D.4</td>
<td>Start up and transient regime of the compressor</td>
<td>303</td>
</tr>
<tr>
<td>E</td>
<td>Principle of mass flow measurements</td>
<td>305</td>
</tr>
<tr>
<td>E.1</td>
<td>Working principle</td>
<td>305</td>
</tr>
<tr>
<td>E.2</td>
<td>Determination of mass flow</td>
<td>306</td>
</tr>
<tr>
<td>E.3</td>
<td>Uncertainty of flow rate measurement</td>
<td>307</td>
</tr>
<tr>
<td>F</td>
<td>Static calibration of fast response transducers</td>
<td>309</td>
</tr>
<tr>
<td>G</td>
<td>Sound propagation in closed pipes</td>
<td>313</td>
</tr>
<tr>
<td>G.1</td>
<td>The organ pipe model</td>
<td>313</td>
</tr>
<tr>
<td>G.2</td>
<td>The Helmotz resonator model</td>
<td>313</td>
</tr>
<tr>
<td>H</td>
<td>Elements of statistical analysis</td>
<td>315</td>
</tr>
<tr>
<td>I</td>
<td>Steady state pressure measurement system accuracy analysis</td>
<td>319</td>
</tr>
<tr>
<td>J</td>
<td>Dynamic calibration of capillarity tubes</td>
<td>323</td>
</tr>
<tr>
<td>J.1</td>
<td>Introduction and objectives</td>
<td>323</td>
</tr>
<tr>
<td>J.2</td>
<td>A non-exhaustive overview on dynamic calibration procedure</td>
<td>324</td>
</tr>
<tr>
<td>J.3</td>
<td>Dynamic calibration methodology</td>
<td>329</td>
</tr>
<tr>
<td>J.4</td>
<td>Practical considerations</td>
<td>330</td>
</tr>
<tr>
<td>J.5</td>
<td>Dynamic calibration acquisition procedure</td>
<td>336</td>
</tr>
<tr>
<td>J.6</td>
<td>Dynamic calibration data reduction</td>
<td>337</td>
</tr>
<tr>
<td>J.7</td>
<td>Error and accuracy of dynamic calibration</td>
<td>338</td>
</tr>
<tr>
<td>K</td>
<td>Conventional Schlieren flow visualization</td>
<td>339</td>
</tr>
<tr>
<td>K.1</td>
<td>Principle of shock visualization method</td>
<td>339</td>
</tr>
<tr>
<td>K.2</td>
<td>Optical set-up</td>
<td>340</td>
</tr>
<tr>
<td>K.3</td>
<td>Image acquisition system</td>
<td>342</td>
</tr>
<tr>
<td>K.4</td>
<td>Error and accuracy</td>
<td>343</td>
</tr>
<tr>
<td>K.4.0.1</td>
<td>Schlieren method accuracy</td>
<td>343</td>
</tr>
<tr>
<td>K.4.0.2</td>
<td>Some practical considerations</td>
<td>345</td>
</tr>
<tr>
<td>K.4.0.3</td>
<td>Geometrical and aerodynamical resolution</td>
<td>346</td>
</tr>
<tr>
<td>L</td>
<td>Laser-Two-Focus working principle</td>
<td>347</td>
</tr>
<tr>
<td>M</td>
<td>Steady State results</td>
<td>349</td>
</tr>
<tr>
<td>M.1</td>
<td>Introduction</td>
<td>349</td>
</tr>
<tr>
<td>M.2</td>
<td>Two dimensional RANS computations on 2D nozzle</td>
<td>350</td>
</tr>
<tr>
<td>M.3</td>
<td>Experimental and numerical results in 2D nozzle</td>
<td>354</td>
</tr>
</tbody>
</table>
## CONTENTS

M.4 Experimental and numerical results on 3D nozzle .................................. 373

N  Unsteady results ............................................................................................. 397
   N.1 Introduction .............................................................................................. 397
   N.2 Unsteady results in two-dimensional nozzle ............................................. 397
      N.2.1 Unsteady pressure measurements ....................................................... 397
      N.2.2 Unsteady simulations ........................................................................ 422
   N.3 Unsteady results on three-dimensional nozzle ......................................... 434
      N.3.1 Unsteady pressure measurements ....................................................... 434
      N.3.2 Three-dimensional RANS calculation ................................................ 442

Bibliography ....................................................................................................... 445
List of Figures

2.1 Unsteady phenomena in axial turbomachines (Oliveira, 1999) .................. 6
2.2 Collar’s triangle for axial turbomachines (Collar, 1946) ......................... 7
2.3 Classical flutter .............................................. 9
2.4 Structure of an equilibrium boundary layer (Schlichting, 1979) ............... 16
2.5 Illustration of APG effect, modification of the “log” and outer region (with $P^t = \nu \frac{\partial P}{\partial x}/\rho u^3$ is the dimensionless pressure gradient parameter) . 18
2.6 Illustration of APG effects on BL profiles (LDV measurements (Angele, 2003)) 19
2.7 Experimental profiles of turbulent and total shear-stress on a flat plate BL (Cousteix, 1989) .......................................................... 19
2.8 Distribution of normalized rms value of thermodynamic variables (left) and normal Reynolds shear stresses (right) through a shock wave (Lee et al., 1993) ................................................................. 21
2.9 Schematic representation of SBLI on airfoils (Pearcey, 1955) .................. 23
2.10 Variants of type B flow in transonic interaction (Pearcey, 1968) ............ 24
2.11 Schematic representation of transonic SBLI without separation (Delery and Marvin, 1986) ................................................................. 25
2.12 Mach number streamwise distributions in the outer flow (Delery, 1977) .... 26
2.13 Influence of Reynolds number on wall pressure distribution in transonic interaction (Laurent, 1977) ......................................................... 26
2.14 Influence of Reynolds number on interaction zone extend (Laurent, 1977) . 27
2.15 Influence of initial conditions on the supersonic interaction length (Delery, 1980) ................................................................................. 27
2.16 Correlation of supersonic interaction length (Delery, 1980) .................... 27
2.17 Definition of incipient separation from wall shear stress distributions .. 28
2.18 Experimental shock induced incipient separation limit in transonic flow (Delery, 1980) ................................................................. 29
2.19 Streamwise Mach number distribution in a transonic interaction with a shock induced separation (Delery, 1978) ............................... 31
2.20 Lambda shock system above an extended separation in a transonic channel flow (Delery, 1978) ................................................................. 31
2.21 Description of the lambda shock system pattern (Delery, 1978) .............. 32
2.22 Illustration of flow field structure in a transonic interaction with extended separation (Seddon, 1976) ......................................................... 33
2.23 Illustration of angle influence on shock induced separation bubble shape (Green, 1970) ................................................................. 34
4.1 Correspondence of the physical and generic coordinate domains .............. 44
LIST OF FIGURES

5.1 Overall air supply facility at HPT .......................... 50
5.2 Operating scheme of the overall facility ..................... 50
5.3 Detailed drawing of the VM100 test section ................... 51
5.4 Desired flow configuration over the 3D bump ............... 54
5.5 Description of the three dimensional bump shape ........... 55
5.6 Three dimensional bump on its rectangular base ............. 56
5.7 Location of the beginning of the three-dimensional bump shape ..... 57
5.8 Large two-dimensional bump ................................ 59
5.9 Location of the beginning of the two-dimensional bump profile .. 59
5.10 Upstream and downstream propagating acoustic waves .......... 60
5.11 Stagnation probes location in the settling chamber .......... 61
5.12 Optical glass (left) and plexiglas (right) side windows ....... 62
5.13 Set of removable plugs to be inserted in the plexiglas side windows ... 63
5.14 Pressure taps design ....................................... 66
5.15 Pressure taps layout on 3Dbump ................................ 67
5.16 General pressure taps and hotfilm layout on 2D bump .......... 68
5.17 A typical hotfilm gage, Nickel sensor and copper leads, deposited onto a polyamide substrate foil ....................... 68
5.18 Two dimensional bump sliding mechanism .................... 69
5.19 Large cassettes located inside the plexiglas side window ....... 70
5.20 Large cassette with inflated O-ring sealing mechanism ....... 70
5.21 Working principle of the inflated O-ring sealing technique ... 71
5.22 A set of male and female quick-connectors to the PSI system .... 71
5.23 Steady state pressure instrumentation on 2D bump ............ 72
5.24 Steady state pressure instrumentation on 3D bump .......... 72
5.25 XCQ-002 Kulite transducer ................................. 73
5.26 Illustration of the locking mechanism for unsteady pressure instrumentation 74
5.27 Basic block-diagram of the servo amplifier for speed control with encoder . 76
5.28 ................................................................. 77
5.29 Overview of the 8400 steady state pressure measurement system .... 80
5.30 Experimental steady state pressure measurement layout on 2D test object .... 81
5.31 Experimental steady state pressure measurement layout on three dimension test object ........................................ 82
5.32 Experimental mesh for steady state pressure measurements on 2D test object 84
5.33 Experimental mesh for steady state pressure measurements on 3D test object 84
5.34 Isentropic Mach number uncertainty for the ±100kPa pressure range scanners 86
5.35 Functional overview of the K8000 system ........................ 87
5.36 Functional overview of one of the four available channel in a MBA101 DC bridge amplifier module ...................... 88
5.37 Unsteady pressure measurement system KT8000 software illustration .... 89
5.38 Overview of the dynamic calibration principle ................. 90
5.39 Dynamic calibration unit - Calibration head .................. 90
5.40 Steady state pressure distribution for extreme rod position on unsteady OP No1 .......................................... 93
5.41 Illustration of data reduction process during unsteady pressure measurements on 2D bump .............................. 95
5.42 Illustration of Ensemble Averaging technique .................. 96
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.43</td>
<td>Illustration of the unsteady pressure measurement chain</td>
</tr>
<tr>
<td>5.44</td>
<td>Overview of the IFA300 Constant Temperature Anemometer System</td>
</tr>
<tr>
<td>5.45</td>
<td>Functional overview of the IFA300 Constant Temperature Anemometer</td>
</tr>
<tr>
<td>5.46</td>
<td>Hotfilm measurement principle</td>
</tr>
<tr>
<td>5.47</td>
<td>Illustration of steady state Schlieren post treatment</td>
</tr>
<tr>
<td>5.48</td>
<td>Illustration of unsteady Schlieren post treatment</td>
</tr>
<tr>
<td>5.49</td>
<td>Experimental setup for oil visualizations</td>
</tr>
<tr>
<td>6.1</td>
<td>Quasi-one dimensional nozzle flow theory</td>
</tr>
<tr>
<td>6.2</td>
<td>Two dimensional nozzle flow analysis</td>
</tr>
<tr>
<td>6.3</td>
<td>Experimental steady state pressure distribution at mid-channel for all operating points</td>
</tr>
<tr>
<td>6.4</td>
<td>Experimental steady state shock location at mid-channel for all operating points</td>
</tr>
<tr>
<td>6.5</td>
<td>Experimental steady state pressure measurements at different width locations for all operating points</td>
</tr>
<tr>
<td>6.6</td>
<td>Standard deviation of experimental steady state pressure measurements</td>
</tr>
<tr>
<td>6.7</td>
<td>Skewness of experimental steady state pressure measurements</td>
</tr>
<tr>
<td>6.8</td>
<td>Flatness of experimental steady state pressure measurements</td>
</tr>
<tr>
<td>6.9</td>
<td>Schlieren visualizations of lambda foot shock structure for steady state operating points</td>
</tr>
<tr>
<td>6.10</td>
<td>Schlieren visualizations of steady state shock positions for all operating points</td>
</tr>
<tr>
<td>6.11</td>
<td>High speed Schlieren visualizations of steady state shock oscillations</td>
</tr>
<tr>
<td>6.12</td>
<td>Oil visualizations of steady state shock induced separation for all operating points</td>
</tr>
<tr>
<td>6.13</td>
<td>Separated flow pattern with steady state oil visualization</td>
</tr>
<tr>
<td>6.14</td>
<td>Heat transfer coefficient fluctuations over 2D bump</td>
</tr>
<tr>
<td>6.15</td>
<td>Boundary layer measurements performed with a Laser-Two-Focus system</td>
</tr>
<tr>
<td>6.16</td>
<td>Inlet boundary layer profiles on lower wall, Hotwire measurements (Sigfrid, 2003)</td>
</tr>
<tr>
<td>6.17</td>
<td>Experimental steady state pressure measurements on 3D bump (weak shock configuration)</td>
</tr>
<tr>
<td>6.18</td>
<td>Experimental flow visualization on 3D bump (weak shock configuration)</td>
</tr>
<tr>
<td>6.19</td>
<td>Steady state isentropic Mach number from pressure measurements in 3D nozzle</td>
</tr>
<tr>
<td>6.20</td>
<td>Steady state conventional Schlieren visualizations in 3D nozzle</td>
</tr>
<tr>
<td>6.21</td>
<td>Steady state oil-paint visualization in 3D nozzle</td>
</tr>
<tr>
<td>6.22</td>
<td>Steady state pressure distribution for all operating points at different y locations</td>
</tr>
<tr>
<td>6.23</td>
<td>Standard deviation (RMS) from steady state pressure measurements in 3D nozzle</td>
</tr>
<tr>
<td>6.24</td>
<td>Skewness from steady state pressure measurements in 3D nozzle</td>
</tr>
<tr>
<td>6.25</td>
<td>Flatness from steady state pressure measurements in 3D nozzle</td>
</tr>
<tr>
<td>6.26</td>
<td>Mesh grid in 3D duct</td>
</tr>
<tr>
<td>6.27</td>
<td>Outlet plane flow in 3D rectangular duct</td>
</tr>
<tr>
<td>6.28</td>
<td>Outflow boundary layer profile in 3D rectangular duct</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>6.29</td>
<td>Dimensionless vertical coordinate $y^+$ for 2D RANS calculation (case with $P_{out}=110kPa$)</td>
</tr>
<tr>
<td>6.30</td>
<td>Shock position - Comparison between experiments and 2D RANS</td>
</tr>
<tr>
<td>6.31</td>
<td>Separated region location - Comparison between experiments and 2D RANS</td>
</tr>
<tr>
<td>6.32</td>
<td>2D RANS steady state pressure distribution on 2D bump</td>
</tr>
<tr>
<td>6.33</td>
<td>Computational meshes for 3D RANS calculations of the 2D bump</td>
</tr>
<tr>
<td>6.34</td>
<td>Shock position - Comparison between experiments, 2D RANS and 3D RANS</td>
</tr>
<tr>
<td>6.35</td>
<td>Separation region location - Comparison between experiments, 2D RANS and 3D RANS</td>
</tr>
<tr>
<td>6.36</td>
<td>Streamlines on 3D RANS steady state calculation on 2D bump ($P_{out}=108kPa$)</td>
</tr>
<tr>
<td>6.37</td>
<td>Skin friction elementary singularities (Legendre, 1977)</td>
</tr>
<tr>
<td>6.38</td>
<td>3D RANS steady state pressure distribution on 2D bump</td>
</tr>
<tr>
<td>6.39</td>
<td>Steady state weak shock structure in 2D nozzle</td>
</tr>
<tr>
<td>6.40</td>
<td>Experimental and numerical comparison of pressure distribution on 2D bump (strong shock configuration $P_{out}=112kPa$)</td>
</tr>
<tr>
<td>6.41</td>
<td>Steady state strong shock structure in 2D nozzle</td>
</tr>
<tr>
<td>6.42</td>
<td>Experimental and numerical comparison of pressure distribution on 2D bump (strong shock configuration $P_{out}=106kPa$)</td>
</tr>
<tr>
<td>6.43</td>
<td>Numerical mesh for 3D RANS calculations</td>
</tr>
<tr>
<td>6.44</td>
<td>Streamlines on 3D RANS steady state calculation on 3D bump ($P_{out}=116kPa$)</td>
</tr>
<tr>
<td>6.45</td>
<td>Experimental visualization of the corner separated flow pattern</td>
</tr>
<tr>
<td>6.46</td>
<td>3D path for flow particle within separated region at $P_{out} = 106kPa$</td>
</tr>
<tr>
<td>6.47</td>
<td>Evolution of sonic pocket and streamlines within separated flow region with decreasing back pressure</td>
</tr>
<tr>
<td>7.1</td>
<td>Experimental shock motion for strong shock configuration and high perturbation amplitude ($A_p = \pm 2.12kPa$)</td>
</tr>
<tr>
<td>7.2</td>
<td>Experimental shock motion for strong shock configuration and high perturbation amplitude ($A_p = \pm 2.12kPa$)</td>
</tr>
<tr>
<td>7.3</td>
<td>Experimental shock motion for strong shock configuration and low perturbation amplitude ($A_p = \pm 1.14kPa$)</td>
</tr>
<tr>
<td>7.4</td>
<td>Experimental phase-lag between unsteady pressure perturbation downstream of shock and shock motion for different operating conditions</td>
</tr>
<tr>
<td>7.5</td>
<td>Experimental phase shift underneath shock location for different operating conditions</td>
</tr>
<tr>
<td>7.6</td>
<td>Numerical phase shift underneath shock location for different operating conditions</td>
</tr>
<tr>
<td>7.7</td>
<td>Experimental-Numerical comparison of unsteady pressure amplification for the weak shock configuration at 100Hz and 500Hz</td>
</tr>
<tr>
<td>7.8</td>
<td>Experimental-Numerical comparison of unsteady pressure amplification for the strong shock configuration at 100Hz and 500Hz</td>
</tr>
<tr>
<td>7.9</td>
<td>Experimental-Numerical comparison of unsteady pressure phase-angle for the weak shock configuration at 100Hz and 500Hz</td>
</tr>
<tr>
<td>7.10</td>
<td>Experimental-Numerical comparison of unsteady pressure phase-angle for the strong shock configuration at 100Hz and 500Hz</td>
</tr>
<tr>
<td>7.11</td>
<td>Experimental-Numerical comparison of unsteady pressure phase-shift underneath shock location</td>
</tr>
</tbody>
</table>
7.12 Experimental-Numerical comparison of the unsteady shock motion amplitude for the weak shock configuration at 100Hz and 500Hz ........................................ 217
7.13 Experimental-Numerical comparison of the unsteady shock motion amplitude for the strong shock configuration at 100Hz and 500Hz ........................ 217
7.14 Experimental-Numerical comparison of the unsteady shock motion phase-angle for the weak shock configuration at 100Hz and 500Hz ......................... 218
7.15 Experimental-Numerical comparison of the unsteady shock motion phase-angle for the strong shock configuration at 100Hz and 500Hz ......................... 219
7.16 Numerical phase-lag between shock motion and unsteady pressure perturbations downstream of shock location for different operating conditions ...... 221
7.17 Numerical unsteady separation motion for the strong shock configuration at 100Hz ........................................................................................................ 222
7.18 Numerical unsteady separation motion for the strong shock configuration at high frequencies .............................................................................................. 223
7.19 Acoustic decomposition of the unsteady pressure distribution on 2D bump surface for the subsonic flow configuration at 100Hz ................................. 224
7.20 Acoustic decomposition of the unsteady pressure distribution on 2D bump surface for the weak shock configuration at 100Hz .................................... 224
7.21 Acoustic decomposition of the unsteady pressure distribution on 2D bump surface for the strong shock configuration at 100Hz .................................... 225
7.22 Acoustic decomposition of the unsteady pressure distribution on 2D bump surface for the choked nozzle flow configuration at 100Hz ......................... 225
7.23 Experimental phase shift underneath shock location for different spanwise positions in 3D nozzle ................................................................................. 229
7.24 Numerical and experimental phase shift underneath shock location for different spanwise positions in 3D nozzle ................................................. 232
7.25 Unsteady motion of sonic pocket and streamlines for fluctuating back pressure ................. 235
A.1 Correspondence of the physical and generic coordinate domains ...................... 258
A.2 Cell vertex formulation .................................................................................. 259
A.3 Flux Difference Splitting method .................................................................. 268
A.4 High order interpolation, MUSCL approach ................................................. 269
A.5 Three dimensional computing molecule for a) 1st order inviscid scheme, b) 2nd order inviscid scheme, and c) 2nd order viscous scheme ....................... 275
A.6 Supplementary planes and edges .................................................................. 276
A.7 parallel computation diagram ........................................................................ 282
B.1 Already existing supersonic wind tunnel ....................................................... 284
B.2 Overall drawing of the newly designed VM100 facility ............................... 285
B.3 Drawing of the new VM100 test section ...................................................... 286
B.4 Drawing of the boundary layer cut off device ............................................. 287
B.5 Effect of the outgoing mass flow on streamlines in free stream ..................... 288
B.6 Location of pressure taps on cutting noses .................................................. 288
C.1 Total pressure probe located in settling chamber ........................................ 289
C.2 Total pressure probe sensitivity towards incoming flow angle ....................... 290
C.3 Total temperature probe ............................................................................. 291
C.4 Basic thermocouple circuitry ...................................................................... 292
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>D.1</td>
<td>Velocity component in settling chamber</td>
<td>296</td>
</tr>
<tr>
<td>D.2</td>
<td>Time dependent pressure signal in wind tunnel test section with non rotating cam</td>
<td>297</td>
</tr>
<tr>
<td>D.3</td>
<td>FFT of fluctuating pressure in wind tunnel test section with non rotating cam (normalized)</td>
<td>298</td>
</tr>
<tr>
<td>D.4</td>
<td>Illustration of a cross correlation on two filtered pressure signals from the settling chamber and the test section</td>
<td>298</td>
</tr>
<tr>
<td>D.5</td>
<td>Time dependent pressure signal in wind tunnel test section with rotating cam at 37Hz (2220rpm)</td>
<td>299</td>
</tr>
<tr>
<td>D.6</td>
<td>FFT of fluctuating pressure in wind tunnel test section with rotating cam at 37Hz (2220rpm)</td>
<td>299</td>
</tr>
<tr>
<td>D.7</td>
<td>Schlieren visualizations of measurement repeatability</td>
<td>302</td>
</tr>
<tr>
<td>D.8</td>
<td>Transient regime of the</td>
<td>304</td>
</tr>
<tr>
<td>E.1</td>
<td>Drawing of the orifice plate device</td>
<td>305</td>
</tr>
<tr>
<td>F.1</td>
<td>Typical linear transfer function of a fast response Kulite transducer</td>
<td>311</td>
</tr>
<tr>
<td>G.1</td>
<td>Working principle of an organ pipe resonator</td>
<td>314</td>
</tr>
<tr>
<td>G.2</td>
<td>Working principle of an Helmholtz resonator</td>
<td>314</td>
</tr>
<tr>
<td>H.1</td>
<td>Illustration of the Skewness and Flatness coefficients</td>
<td>317</td>
</tr>
</tbody>
</table>
| J.1    | Illustration of theoretical model for dynamic system characteristic determi- 
|       | nation (Bergh and Tijdeman, 1965)                                           | 325  |
| J.2    | Overview of the dynamic calibration principle                               | 329  |
| J.3    | Dynamic calibration unit - Calibration head                                 | 330  |
| J.4    | Dynamic calibration unit - Fluctuating pressure generator                   | 331  |
| J.5    | Overall static calibration apparatus                                        | 332  |
| J.6    | Static calibration drift of Kulite transducers                              | 332  |
| J.7    | Signal quality characteristics of the fluctuating pressure source used during dynamic calibration | 333  |
| J.8    | Influence of perturbation frequency on Damping and Phase-lag through shortest and longest capillarity tubes on 2D and 3D test objects | 335  |
| J.9    | Influence of perturbation amplitude on Damping and Phase-lag through a 20mm long capillarity tubes on 3D bump | 336  |
| K.1    | Basic layout of a conventional Schlieren system                             | 340  |
| K.2    | Optical set-up of a conventional Schlieren system                           | 341  |
| K.3    | Displacement of the deflected light in the knife edge plane                 | 344  |
| L.1    | Working principle of a 2D Laser-Two -Focus system                           | 347  |
| L.2    | Working principle of a 3D Laser-Two -Focus system                           | 348  |
| M.1    | Two-dimensional steady state Navier-Stokes simulation of 2D Nozzle          
| Operating conditions: Pt=160kPa Tt=303K Psout=120-112kPa                    | 351  |
| M.2    | Two-dimensional steady state Navier-Stokes simulation of 2D Nozzle          
| Operating conditions: Pt=160kPa Tt=303K Psout=112-106kPa                     | 352  |
M.3 Two-dimensional steady state Navier-Stokes simulation of 2D Nozzle
Operating conditions: Pt=160kPa Tt=303K Psout=104kPa 352

M.4 Two-dimensional steady state Navier-Stokes simulation of 2D Nozzle
Operating conditions: Pt=160kPa Tt=303K Psout=102kPa 353

M.5 Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈118kPa 354

M.6 Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈118kPa 355

M.7 Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈116kPa 356

M.8 Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈116kPa 357

M.9 Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈118kPa 358

M.10 Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈118kPa 359

M.11 Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈116kPa 360

M.12 Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈116kPa 361

M.13 Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈110kPa 362

M.14 Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈110kPa 363

M.15 Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈108kPa 364

M.16 Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈108kPa 365

M.17 Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈106kPa 366

M.18 Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈106kPa 367

M.19 Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈104kPa 368

M.20 Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈104kPa 369

M.21 Experimental steady state results and self comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈102kPa 370

M.22 Experimental steady state results and self comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈100kPa 371

M.23 Experimental steady state results and self comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈098kPa 372

M.24 Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈128kPa 373

M.25 Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈126kPa 374
M.26 Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 124 kPa . . . . . . . . 375
M.27 Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 122 kPa . . . . . . . . 376
M.28 Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 120 kPa . . . . . . . . 377
M.29 Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 118 kPa . . . . . . . . 378
M.30 Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 118 kPa . . . . . . . . 379
M.31 Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 116 kPa . . . . . . . . 380
M.32 Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 116 kPa . . . . . . . . 381
M.33 Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 114 kPa . . . . . . . . 382
M.34 Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 114 kPa . . . . . . . . 383
M.35 Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 112 kPa . . . . . . . . 384
M.36 Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 112 kPa . . . . . . . . 385
M.37 Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 110 kPa . . . . . . . . 386
M.38 Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 110 kPa . . . . . . . . 387
M.39 Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 108 kPa . . . . . . . . 388
M.40 Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 108 kPa . . . . . . . . 389
M.41 Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 106 kPa . . . . . . . . 390
M.42 Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 106 kPa . . . . . . . . 391
M.43 Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 104 kPa . . . . . . . . 392
M.44 Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 104 kPa . . . . . . . . 393
M.45 Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 102 kPa . . . . . . . . 394
M.46 Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 100 kPa . . . . . . . . 395
M.47 Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 098 kPa . . . . . . . . 396

N.1 Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: Pt ≈ 160 kPa Tt ≈ 303 K Psout ≈ 106 kPa F_p = 50Hz A_p = ± 2.12 kPa 398
N.2 Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=50\text{Hz}$ $A_p = ±2.12\text{kPa}$ 399
N.3 Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=100\text{Hz}$ $A_p = ±2.12\text{kPa}$ 400
N.4 Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=100\text{Hz}$ $A_p = ±2.12\text{kPa}$ 401
N.5 Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=250\text{Hz}$ $A_p = ±2.12\text{kPa}$ 402
N.6 Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=250\text{Hz}$ $A_p = ±2.12\text{kPa}$ 403
N.7 Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=50\text{Hz}$ $A_p = ±2.12\text{kPa}$ 404
N.8 Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=50\text{Hz}$ $A_p = ±2.12\text{kPa}$ 405
N.9 Unsteady pressure measurement results in 2D Nozzle
Expe-OP2: Pt≈160kPa Tt≈303K Psout≈112kPa $F_p=50\text{Hz}$ $A_p = ±1.25\text{kPa}$ 406
N.10 Unsteady pressure measurement results in 2D Nozzle
Expe-OP2: Pt≈160kPa Tt≈303K Psout≈112kPa $F_p=50\text{Hz}$ $A_p = ±1.25\text{kPa}$ 407
N.11 Unsteady pressure measurement results in 2D Nozzle
Expe-OP2: Pt≈160kPa Tt≈303K Psout≈112kPa $F_p=100\text{Hz}$ $A_p = ±1.25\text{kPa}$ 408
N.12 Unsteady pressure measurement results in 2D Nozzle
Expe-OP2: Pt≈160kPa Tt≈303K Psout≈112kPa $F_p=100\text{Hz}$ $A_p = ±1.25\text{kPa}$ 409
N.13 Unsteady pressure measurement results in 2D Nozzle
Expe-OP2: Pt≈160kPa Tt≈303K Psout≈112kPa $F_p=250\text{Hz}$ $A_p = ±1.25\text{kPa}$ 410
N.14 Unsteady pressure measurement results in 2D Nozzle
Expe-OP2: Pt≈160kPa Tt≈303K Psout≈112kPa $F_p=250\text{Hz}$ $A_p = ±1.25\text{kPa}$ 411
N.15 Unsteady pressure measurement results in 2D Nozzle
Expe-OP2: Pt≈160kPa Tt≈303K Psout≈112kPa $F_p=50\text{Hz}$ $A_p = ±1.25\text{kPa}$ 412
N.16 Unsteady pressure measurement results in 2D Nozzle
Expe-OP2: Pt≈160kPa Tt≈303K Psout≈112kPa $F_p=50\text{Hz}$ $A_p = ±1.25\text{kPa}$ 413
N.17 Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=50\text{Hz}$ $A_p = ±1.14\text{kPa}$ 414
N.18 Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=50\text{Hz}$ $A_p = ±1.14\text{kPa}$ 415
N.19 Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=100\text{Hz}$ $A_p = ±1.14\text{kPa}$ 416
N.20 Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=100\text{Hz}$ $A_p = ±1.14\text{kPa}$ 417
N.21 Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=250\text{Hz}$ $A_p = ±1.14\text{kPa}$ 418
N.22 Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=250\text{Hz}$ $A_p = ±1.14\text{kPa}$ 419
N.23 Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=500\text{Hz}$ $A_p = ±1.14\text{kPa}$ 420
N.24 Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: Pt≈160kPa Tt≈303K Psout≈106kPa $F_p=500\text{Hz}$ $A_p = ±1.14\text{kPa}$ 421
LIST OF FIGURES

N.25 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP1: Pt≈160kPa Tt≈303K Psout≈120kPa F_p=100Hz A_p = ±2.4kPa 422
N.26 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP1: Pt≈160kPa Tt≈303K Psout≈120kPa F_p=500Hz A_p = ±2.4kPa 423
N.27 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP1: Pt≈160kPa Tt≈303K Psout≈120kPa F_p=1000Hz A_p = ±2.4kPa 424
N.28 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP2: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=100Hz A_p = ±2.32kPa 425
N.29 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP2: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=500Hz A_p = ±2.32kPa 426
N.30 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP2: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=1000Hz A_p = ±2.32kPa 427
N.31 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP3: Pt≈160kPa Tt≈303K Psout≈110kPa F_p=100Hz A_p = ±2.2kPa 428
N.32 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP3: Pt≈160kPa Tt≈303K Psout≈110kPa F_p=500Hz A_p = ±2.2kPa 429
N.33 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP3: Pt≈160kPa Tt≈303K Psout≈110kPa F_p=1000Hz A_p = ±2.2kPa 430
N.34 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP4: Pt≈160kPa Tt≈303K Psout≈102kPa F_p=100Hz A_p = ±2.04kPa 431
N.35 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP4: Pt≈160kPa Tt≈303K Psout≈102kPa F_p=500Hz A_p = ±2.04kPa 432
N.36 Unsteady 2D RANS numerical results in 2D Nozzle
  Num-OP4: Pt≈160kPa Tt≈303K Psout≈102kPa F_p=1000Hz A_p = ±2.04kPa 433
N.37 Unsteady pressure measurement results in 3D Nozzle
  Expe-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=50Hz A_p = ±2.10kPa 434
N.38 Unsteady pressure measurement results in 3D Nozzle
  Expe-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=50Hz A_p = ±2.10kPa 435
N.39 Unsteady pressure measurement results in 3D Nozzle
  Expe-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=100Hz A_p = ±2.10kPa 436
N.40 Unsteady pressure measurement results in 3D Nozzle
  Expe-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=100Hz A_p = ±2.10kPa 437
N.41 Unsteady pressure measurement results in 3D Nozzle
  Expe-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=250Hz A_p = ±2.10kPa 438
N.42 Unsteady pressure measurement results in 3D Nozzle
  Expe-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=250Hz A_p = ±2.10kPa 439
N.43 Unsteady pressure measurement results in 3D Nozzle
  Expe-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=250Hz A_p = ±2.10kPa 440
N.44 Unsteady pressure measurement results in 3D Nozzle
  Expe-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=500Hz A_p = ±2.10kPa 441
N.45 Unsteady 3D RANS numerical results in 3D Nozzle
  Unsteady pressure distribution on 3D bump
  Num-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=500Hz A_p = ±2.32kPa 442
N.46 Unsteady 3D RANS numerical results in 3D Nozzle
  Unsteady pressure distribution on 3D bump
  Num-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=500Hz A_p = ±2.32kPa 443

xxviii
N.47 Unsteady 3D RANS numerical results in 3D Nozzle
Unsteady pressure distribution in 3D nozzle channel
Num-OP: \( P_t \approx 160\text{kPa} \), \( T_t \approx 303\text{K} \), \( P_{\text{out}} \approx 116\text{kPa} \), \( f_p = 500\text{Hz} \), \( A_p = \pm 2.32\text{kPa} \)

N.48 Unsteady 3D RANS numerical results in 3D Nozzle
Unsteady shock motion in 3D nozzle channel
Num-OP: \( P_t \approx 160\text{kPa} \), \( T_t \approx 303\text{K} \), \( P_{\text{out}} \approx 116\text{kPa} \), \( f_p = 500\text{Hz} \), \( A_p = \pm 2.32\text{kPa} \)
LIST OF FIGURES
## List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1</td>
<td>Design parameters for the 3D bump</td>
</tr>
<tr>
<td>5.2</td>
<td>3D bump profile description at different transversal locations in the channel</td>
</tr>
<tr>
<td>5.3</td>
<td>Two-dimensional test object profile coordinates</td>
</tr>
<tr>
<td>5.4</td>
<td>Profile description of the 2D bump geometry</td>
</tr>
<tr>
<td>5.5</td>
<td>Dimensions of elliptical cams</td>
</tr>
<tr>
<td>5.6</td>
<td>Motor fluctuations at different rotating speeds</td>
</tr>
<tr>
<td>5.7</td>
<td>Operating conditions for steady state pressure measurements in 2D nozzle</td>
</tr>
<tr>
<td>5.8</td>
<td>Operating conditions for steady state pressure measurements in 3D nozzle</td>
</tr>
<tr>
<td>5.9</td>
<td>Operating conditions during unsteady pressure measurements</td>
</tr>
<tr>
<td>5.10</td>
<td>Breakdown of precision errors during static calibration procedure</td>
</tr>
<tr>
<td>5.11</td>
<td>Breakdown of precision errors during dynamic calibration procedure</td>
</tr>
<tr>
<td>5.12</td>
<td>Operating conditions for hotfilm measurements on 2Dbump</td>
</tr>
<tr>
<td>5.13</td>
<td>Operating conditions for steady state Schlieren visualizations</td>
</tr>
<tr>
<td>5.14</td>
<td>Operating conditions and sampling parameters for unsteady Schlieren visualizations</td>
</tr>
<tr>
<td>5.15</td>
<td>Operating conditions for oil visualizations on 2Dbump</td>
</tr>
<tr>
<td>5.16</td>
<td>Operating conditions for oil visualizations on 3Dbump</td>
</tr>
<tr>
<td>6.1</td>
<td>Separation and reattachment location based on pressure measurements</td>
</tr>
<tr>
<td>6.2</td>
<td>Operating conditions during Hotwire and L2F measurements in the empty test section</td>
</tr>
<tr>
<td>6.3</td>
<td>Experimental boundary layer characteristics</td>
</tr>
<tr>
<td>6.4</td>
<td>Extent of corner separation on 3D bump</td>
</tr>
<tr>
<td>6.5</td>
<td>Operating conditions for 2D RANS calculations in 2D nozzle</td>
</tr>
<tr>
<td>6.6</td>
<td>Operating conditions for 3D RANS calculations in 2D nozzle</td>
</tr>
<tr>
<td>6.7</td>
<td>Operating conditions for 3D RANS calculations in 3D bump</td>
</tr>
<tr>
<td>7.1</td>
<td>Unsteady operating conditions for 2D RANS calculations in 2D nozzle</td>
</tr>
<tr>
<td>7.2</td>
<td>Characteristics of the reattachment point motion for the strong shock configuration</td>
</tr>
<tr>
<td>7.3</td>
<td>Unsteady operating conditions for 3D RANS calculations in 3D nozzle</td>
</tr>
<tr>
<td>D.1</td>
<td>Operating conditions during repeatability measurements</td>
</tr>
<tr>
<td>D.2</td>
<td>Mean shock positions at y=30mm (arrow location in Figure D.2)</td>
</tr>
<tr>
<td>H.1</td>
<td>Confidence interval for a normal distribution</td>
</tr>
<tr>
<td>I.1</td>
<td>Breakdown of the bias and precision errors of the PSI 8400 system</td>
</tr>
</tbody>
</table>
LIST OF TABLES

K.1 Performance parameters for the MotionScope 8000S PCI imaging system . 343
List of publications

The present thesis work has led to a few publications and internal reports, which are enumerated below:

① Ferrand, P.; Aubert, S.; Smati, L.; Bron, O.; Atassi, H.
"Non linear interaction of upstream propagating sound with transonic flows in nozzle"

② Bron, O.
"Rapport d’avancement de thèse en cotutelle entre l’ECL et le KTH"

③ Smati, L.; Ferrand, P.; Bron, O.
"Simulation numériques des interactions onde de choc couche limite sous l’effet d’une fluctuation de pression statique aval"
AAAF Conference, 1999, Marseille, France.

④ Bron, O.; Freudenreich, K.; Fransson, T. H.
"Redesign and taking into service of a new test facility for nozzle flow experiment”

⑤ Bron, O.; Ferrand, P.; Fransson, T. H.; Atassi, H.M.
"Non Linear Interaction Of Acoustic Waves With Transonic Flows In Nozzles”

⑥ Bron, O.; Ferrand, P.; Fransson, T. H.; Aubert, S.; Caro, J.
"Numerical study of Non Linear Interaction In 2D NOZZLE Transonic Flow”
The 9th of International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, Honolulu, Hawaii, February 10-14, 2002.

⑦ Bron, O.
"Investigation of the air flow quality in the VM100 wind tunnel. Part 1: Comparison of acoustic signature versus imposed perturbations”

⑧ Fransson, T. H.; Bron, O.; Allegret-Bourdon, D.
"Experimental and numerical study of non-linear interactions in transonic nozzle flow”

⑨ Bron, O.; Fransson, T.H.; Ferrand, P.
"Experimental and numerical study of non-linear interactions in two dimensional transonic nozzle flow”
## Nomenclature

### Latin Letters

<table>
<thead>
<tr>
<th>letter</th>
<th>signification</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>⃗a</td>
<td>Counter-varying basis vector associated to the direction $\xi^i$</td>
<td>-</td>
</tr>
<tr>
<td>⃗a_i</td>
<td>Co-varying basis vector associated to the direction $\xi^i$</td>
<td>-</td>
</tr>
<tr>
<td>A</td>
<td>Scalar or vectorial variable</td>
<td>-</td>
</tr>
<tr>
<td>a, b</td>
<td>Sensitivity and offset coefficients</td>
<td>[*]</td>
</tr>
<tr>
<td>B</td>
<td>Vectorial or tensorial variable</td>
<td>-</td>
</tr>
<tr>
<td>B(q)</td>
<td>Real value of the physical property imposed on numerical boundaries</td>
<td>-</td>
</tr>
<tr>
<td>$B^*(t)$</td>
<td>Physical property imposed on numerical boundaries</td>
<td>-</td>
</tr>
<tr>
<td>c</td>
<td>Sound velocity</td>
<td>[m/s]</td>
</tr>
<tr>
<td>$C_f$</td>
<td>Friction coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Specific heat capacity at constant pressure</td>
<td>[J/Kg/K]</td>
</tr>
<tr>
<td>$C_v$</td>
<td>Specific heat capacity at constant volume</td>
<td>[J/Kg/K]</td>
</tr>
<tr>
<td>$C_k$, $C_{\omega_1}$, $C_{\omega_2}$</td>
<td>$k - \omega$ turbulence model constants</td>
<td>[-]</td>
</tr>
<tr>
<td>D, d</td>
<td>Various distance or diameter</td>
<td>[m]</td>
</tr>
<tr>
<td>e</td>
<td>Specific internal energy</td>
<td>[J]</td>
</tr>
<tr>
<td>E</td>
<td>Total energy</td>
<td>[J]</td>
</tr>
<tr>
<td>F, f</td>
<td>Frequency</td>
<td>[Hz]</td>
</tr>
<tr>
<td>f</td>
<td>Focal length</td>
<td>[m]</td>
</tr>
<tr>
<td>F, ($\gamma_2$, $\mu_4$)</td>
<td>Flatness ($4^{th}$ order central moment)</td>
<td>[-]</td>
</tr>
<tr>
<td>$F_p$</td>
<td>Perturbation frequency</td>
<td>[Hz]</td>
</tr>
<tr>
<td>$F_c$</td>
<td>Critical (or cut) frequency</td>
<td>[Hz]</td>
</tr>
<tr>
<td>$F^*$</td>
<td>Numerical flux associated to the direction $\xi^i$</td>
<td>-</td>
</tr>
<tr>
<td>$F^+, F^-$</td>
<td>Left and right contributions in the convective flux term</td>
<td>-</td>
</tr>
<tr>
<td>$F_c^*$</td>
<td>Overall convective flux (associated to the direction $\xi^i$)</td>
<td>-</td>
</tr>
<tr>
<td>$F_v^*$</td>
<td>Overall diffusive flux (associated to the direction $\xi^i$)</td>
<td>-</td>
</tr>
<tr>
<td>$\sqrt{g}$</td>
<td>Jacobian of the geometrical transformation</td>
<td>-</td>
</tr>
<tr>
<td>h</td>
<td>Enthalpy</td>
<td>[J]</td>
</tr>
<tr>
<td>H or $H_{12}$</td>
<td>Boundary layer shape factor</td>
<td>[-]</td>
</tr>
<tr>
<td>I</td>
<td>Identity matrix</td>
<td>[-]</td>
</tr>
<tr>
<td>I</td>
<td>Electrical current</td>
<td>[A]</td>
</tr>
<tr>
<td>Letter</td>
<td>Signification</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>---------------</td>
<td>--------------------</td>
</tr>
<tr>
<td>k</td>
<td>Turbulent kinetic energy</td>
<td>[J/kg]</td>
</tr>
<tr>
<td></td>
<td>or Reduced frequency</td>
<td>[-]</td>
</tr>
<tr>
<td></td>
<td>or Thermal conductivity</td>
<td>[W/m/K]</td>
</tr>
<tr>
<td>K</td>
<td>Von Karman constant</td>
<td>[-]</td>
</tr>
<tr>
<td></td>
<td>or Stiffness matrix</td>
<td>[N/m]</td>
</tr>
<tr>
<td>$K'$</td>
<td>Jacobian matrix of the overall convective flux term</td>
<td>-</td>
</tr>
<tr>
<td>L</td>
<td>Characteristic length</td>
<td>[m]</td>
</tr>
<tr>
<td>$L^*$</td>
<td>SBLI interaction length</td>
<td>[m]</td>
</tr>
<tr>
<td>M</td>
<td>Mach number</td>
<td>[-]</td>
</tr>
<tr>
<td>$\vec{n}$</td>
<td>Normalized normal vector (to the surface $S(t)$)</td>
<td>-</td>
</tr>
<tr>
<td>p, P</td>
<td>Pressure</td>
<td>[Pa]</td>
</tr>
<tr>
<td>$P_k$</td>
<td>Turbulent kinetic energy production term</td>
<td>-</td>
</tr>
<tr>
<td>$P_r$</td>
<td>Prandtl number</td>
<td>[-]</td>
</tr>
<tr>
<td>q, Q</td>
<td>Unknown conservative variables</td>
<td>-</td>
</tr>
<tr>
<td>Q</td>
<td>Heat flux</td>
<td>[J/Kg/K]</td>
</tr>
<tr>
<td>r</td>
<td>Perfect gas constant</td>
<td>[J/Kg/K]</td>
</tr>
<tr>
<td>R</td>
<td>Residual variable</td>
<td>-</td>
</tr>
<tr>
<td>$R_e$</td>
<td>Reynolds number</td>
<td>[-]</td>
</tr>
<tr>
<td>s</td>
<td>Entropy</td>
<td>[J/Kg/K]</td>
</tr>
<tr>
<td>S</td>
<td>Scalar or vectorial variable</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>or Root Sum Square on precision error</td>
<td>-</td>
</tr>
<tr>
<td>$S_1$</td>
<td>Sutherland’s law constant</td>
<td>[-]</td>
</tr>
<tr>
<td>S</td>
<td>Surface of the numerical integration cell</td>
<td>[m²]</td>
</tr>
<tr>
<td></td>
<td>or Conservative equation source term</td>
<td>-</td>
</tr>
<tr>
<td>$S_{ij}$</td>
<td>Stress tensor</td>
<td>[N/m²]</td>
</tr>
<tr>
<td>$S_{ij}$</td>
<td>Turbulent kinetic energy equation source term</td>
<td>[J]</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
<td>[s]</td>
</tr>
<tr>
<td>T</td>
<td>Period</td>
<td>[s]</td>
</tr>
<tr>
<td>u</td>
<td>Characteristic velocity scale</td>
<td>[m/s]</td>
</tr>
<tr>
<td>$u_r$</td>
<td>Characteristic friction velocity scale</td>
<td>[m/s]</td>
</tr>
<tr>
<td>$U_i$</td>
<td>Velocity vector components (in cartesian coordinates)</td>
<td>[m/s]</td>
</tr>
<tr>
<td>(U, V, W)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$V_{x}$, $V_{y}$, $V_{z}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$V$</td>
<td>Volume of the integration cell</td>
<td>[m³]</td>
</tr>
<tr>
<td>$V_i$</td>
<td>Co-varying flow velocity associated to the direction $\xi^i$</td>
<td>[-]</td>
</tr>
<tr>
<td>$V^*$</td>
<td>Counter-varying flow velocity associated to the direction $\xi^i$</td>
<td>[-]</td>
</tr>
<tr>
<td>$V^*$</td>
<td>Velocity vector</td>
<td>[m/s]</td>
</tr>
<tr>
<td>$x_{i} ou(x, y, z)$</td>
<td>cartesian coordinates</td>
<td>[m]</td>
</tr>
<tr>
<td>Y</td>
<td>Passive Scalar variable</td>
<td>-</td>
</tr>
<tr>
<td>$y^+$</td>
<td>Dimensionless normal coordinate to the wall</td>
<td>[-]</td>
</tr>
</tbody>
</table>
## Greek Letters

<table>
<thead>
<tr>
<th>letter</th>
<th>signification</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha_1, ..., \alpha_5$</td>
<td>Runge-Kutta scheme coefficients</td>
<td>[-]</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Numerical smoothing parameter</td>
<td>[-]</td>
</tr>
<tr>
<td>$\chi$</td>
<td>Chi-merit function</td>
<td>[-]</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Boundary layer thickness</td>
<td>[m]</td>
</tr>
<tr>
<td>$\delta_i$</td>
<td>Spatial operator (centered scheme) associated to the direction $i$</td>
<td></td>
</tr>
<tr>
<td>$\delta_{ij}$</td>
<td>Kronecker symbol</td>
<td>-</td>
</tr>
<tr>
<td>$\Delta t$</td>
<td>Time step</td>
<td>[s]</td>
</tr>
<tr>
<td>$\delta^*$</td>
<td>Boundary layer displacement thickness</td>
<td>[m]</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Turbulent dissipation</td>
<td>[1/s]</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Boundary layer momentum thickness</td>
<td>[m]</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Specific heat capacity ratio</td>
<td>[-]</td>
</tr>
<tr>
<td>or Intermittent parameter</td>
<td>[-]</td>
<td></td>
</tr>
<tr>
<td>$\kappa$</td>
<td>Thermal conductivity</td>
<td>[W/m/K]</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Laminar viscosity</td>
<td>[kg/m/s]</td>
</tr>
<tr>
<td>or Wave length</td>
<td>[m]</td>
<td></td>
</tr>
<tr>
<td>$\lambda_i$</td>
<td>eigen values</td>
<td>-</td>
</tr>
<tr>
<td>$\Lambda_i$</td>
<td>Diagonal eigen matrix</td>
<td>-</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
<td>[kg/m/s]</td>
</tr>
<tr>
<td>$\mu_1$</td>
<td>Arithmetic mean (1st order central moment)</td>
<td></td>
</tr>
<tr>
<td>$\mu_n$</td>
<td>$n^{th}$ order central moment</td>
<td></td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
<td>[kg/m/s]</td>
</tr>
<tr>
<td>$\xi^l$</td>
<td>Curvilinier spatial variables</td>
<td>[m]</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>[kg/m$^3$]</td>
</tr>
<tr>
<td>$\rho_{spect}$</td>
<td>Spectral radius of Jacobian matrix</td>
<td>[m]</td>
</tr>
<tr>
<td>$\sigma_k, \sigma_\omega$</td>
<td>$k - \omega$ turbulence model constant</td>
<td>[-]</td>
</tr>
<tr>
<td>$\sigma, \mu_2$</td>
<td>Standard deviation (square root of variance)</td>
<td></td>
</tr>
<tr>
<td>$\tau$</td>
<td>Shear stress (tensor)</td>
<td>[kg/m/s$^2$]</td>
</tr>
<tr>
<td>$\Phi$</td>
<td>Heat flux</td>
<td>[J]</td>
</tr>
<tr>
<td>or Spatial accuracy limitor</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>$\omega$</td>
<td>Specific turbulent dissipation</td>
<td>[1/s]</td>
</tr>
<tr>
<td>$\Omega_{ij}$</td>
<td>Rotational tensor</td>
<td>-</td>
</tr>
<tr>
<td>$\chi$</td>
<td>Numerical permeability coefficient</td>
<td>[-]</td>
</tr>
</tbody>
</table>

## Subscripts

<table>
<thead>
<tr>
<th>letter</th>
<th>signification</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Value on the upstream face of the shock</td>
</tr>
<tr>
<td>1</td>
<td>Value on the downstream face of the shock</td>
</tr>
<tr>
<td>$\infty$</td>
<td>Free stream or infinite upstream value</td>
</tr>
<tr>
<td>c</td>
<td>Convective term</td>
</tr>
</tbody>
</table>
LIST OF TABLES

<table>
<thead>
<tr>
<th>letter</th>
<th>signification</th>
</tr>
</thead>
<tbody>
<tr>
<td>cbl</td>
<td>Cable value</td>
</tr>
<tr>
<td>δ</td>
<td>Based on boundary layer thickness</td>
</tr>
<tr>
<td>exp</td>
<td>Explicit</td>
</tr>
<tr>
<td>imp</td>
<td>Implicit</td>
</tr>
<tr>
<td>ISO</td>
<td>Isentropic value</td>
</tr>
<tr>
<td>l</td>
<td>Laminar value</td>
</tr>
<tr>
<td>le</td>
<td>Leading edge</td>
</tr>
<tr>
<td>ref</td>
<td>Reference value for Sutherland’s law</td>
</tr>
<tr>
<td>s</td>
<td>Static value</td>
</tr>
<tr>
<td>t</td>
<td>Turbulent value</td>
</tr>
<tr>
<td>te</td>
<td>Trailing edge</td>
</tr>
<tr>
<td>throat</td>
<td>Value at the throat</td>
</tr>
<tr>
<td>t</td>
<td>Total (stagnation) value</td>
</tr>
<tr>
<td>v</td>
<td>Diffusive term</td>
</tr>
<tr>
<td>w</td>
<td>Value at the wall</td>
</tr>
<tr>
<td></td>
<td>Hotfilm or hotwire sensor value</td>
</tr>
</tbody>
</table>

Superscripts

<table>
<thead>
<tr>
<th>letter</th>
<th>signification</th>
</tr>
</thead>
<tbody>
<tr>
<td>+</td>
<td>Non dimensionalized value</td>
</tr>
<tr>
<td>*</td>
<td>Non dimensionalized value</td>
</tr>
<tr>
<td>img</td>
<td>Image (optical) value</td>
</tr>
<tr>
<td>in</td>
<td>Inlet</td>
</tr>
<tr>
<td>t</td>
<td>Transposed</td>
</tr>
<tr>
<td>obj</td>
<td>Object (optical) value</td>
</tr>
<tr>
<td>out</td>
<td>Outlet</td>
</tr>
</tbody>
</table>

Symbols

<table>
<thead>
<tr>
<th>letter</th>
<th>signification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Δ( )</td>
<td>Difference</td>
</tr>
<tr>
<td>()</td>
<td>Reynolds averaged mean value</td>
</tr>
<tr>
<td>( )</td>
<td>Favre averaged mean value</td>
</tr>
<tr>
<td>(′)</td>
<td>Reynolds averaged fluctuating value</td>
</tr>
<tr>
<td>(′′)</td>
<td>Favre averaged fluctuating value</td>
</tr>
</tbody>
</table>

Abbreviations

<table>
<thead>
<tr>
<th>letter</th>
<th>signification</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D</td>
<td>Two dimensional</td>
</tr>
<tr>
<td>letter</td>
<td>signification</td>
</tr>
<tr>
<td>--------</td>
<td>---------------</td>
</tr>
<tr>
<td>3D</td>
<td>Three dimensional</td>
</tr>
<tr>
<td>AD</td>
<td>Analog-Digital</td>
</tr>
<tr>
<td>AAC</td>
<td>Analog Analog Conversion</td>
</tr>
<tr>
<td>AC</td>
<td>Alternate Current</td>
</tr>
<tr>
<td>ADC</td>
<td>Analog Digital Conversion</td>
</tr>
<tr>
<td>ADI</td>
<td>Alternating Direction Implicit</td>
</tr>
<tr>
<td>APG</td>
<td>Adverse Pressure Gradient</td>
</tr>
<tr>
<td>BL</td>
<td>Boundary Layer</td>
</tr>
<tr>
<td>CI</td>
<td>Confidence Interval</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Friedrichs-Levy Criteria</td>
</tr>
<tr>
<td>CTA</td>
<td>Constant Temperature Anemometer</td>
</tr>
<tr>
<td>DFS</td>
<td>Discrete Fourier Serie Decomposition</td>
</tr>
<tr>
<td>DC</td>
<td>Direct current</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>ESP</td>
<td>Electronic Pressure Scanner</td>
</tr>
<tr>
<td>F</td>
<td>Flatness</td>
</tr>
<tr>
<td>FFT</td>
<td>Fast Fourier Transform</td>
</tr>
<tr>
<td>FT</td>
<td>Fourier Transform</td>
</tr>
<tr>
<td>FS</td>
<td>Full Scale</td>
</tr>
<tr>
<td>GPIB</td>
<td>General Purpose Interface Bus (Standard)</td>
</tr>
<tr>
<td>HCF</td>
<td>High Cycle Fatigue</td>
</tr>
<tr>
<td>HPT</td>
<td>Heat and Power Technology</td>
</tr>
<tr>
<td>HSD</td>
<td>High Speed Data</td>
</tr>
<tr>
<td>IMP</td>
<td>Isolated Measurement Pod</td>
</tr>
<tr>
<td>IEEE</td>
<td>Institute of Electrical and Electronics Engineers (Standard)</td>
</tr>
<tr>
<td>IS</td>
<td>Incipient Separation</td>
</tr>
<tr>
<td>ITS</td>
<td>Intermittent Transitory Separation</td>
</tr>
<tr>
<td>L2F</td>
<td>Laser Two Focus (Anemometry)</td>
</tr>
<tr>
<td>LDA</td>
<td>Laser Doppler Anemometry</td>
</tr>
<tr>
<td>LE</td>
<td>Leading Edge</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>LIA</td>
<td>Linear Interaction Approach</td>
</tr>
<tr>
<td>LMFA</td>
<td>Laboratoire de Mécanique des Fluides et d’Acoustique</td>
</tr>
<tr>
<td>MUSCL</td>
<td>Monotone Upstream-centered Scheme for Conservation Laws</td>
</tr>
<tr>
<td>OP</td>
<td>Operating Point</td>
</tr>
<tr>
<td>PIV</td>
<td>Particule Image Velocimetry</td>
</tr>
<tr>
<td>PCU</td>
<td>Pressure Calibration Unit</td>
</tr>
<tr>
<td>PSI</td>
<td>Pressure System Incorporation</td>
</tr>
<tr>
<td>PSU</td>
<td>Pressure Standard Unit</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier-Stokes</td>
</tr>
<tr>
<td>RH</td>
<td>Rankine-Hugoniot</td>
</tr>
<tr>
<td>RDT</td>
<td>Rapid Distorsion Theory</td>
</tr>
<tr>
<td>RMS</td>
<td>Root Mean Square</td>
</tr>
<tr>
<td>letter</td>
<td>signification</td>
</tr>
<tr>
<td>--------</td>
<td>----------------------------------------------------</td>
</tr>
<tr>
<td>RPM</td>
<td>Revolution Per Minute</td>
</tr>
<tr>
<td>RSS</td>
<td>Root Sum Square</td>
</tr>
<tr>
<td>SBLI</td>
<td>Shock Boundary Layer Interaction</td>
</tr>
<tr>
<td>S</td>
<td>Skewness</td>
</tr>
<tr>
<td>SP</td>
<td>System Processor</td>
</tr>
<tr>
<td>TE</td>
<td>Trailing Edge</td>
</tr>
<tr>
<td>TF</td>
<td>Transfer Function</td>
</tr>
<tr>
<td>TS</td>
<td>Transitory Separation</td>
</tr>
<tr>
<td>TVD</td>
<td>Total Variation Diminishing</td>
</tr>
<tr>
<td>TTL</td>
<td>Transistor-Transistor Logic (Standard)</td>
</tr>
<tr>
<td>ZPG</td>
<td>Zero Pressure Gradient</td>
</tr>
</tbody>
</table>
Chapter 1
Introduction

Nowadays, evolution in axial turbomachinery tends towards highly loaded and lower weight engines. This is especially true for modern transonic compressors with high compression ratio, in which the axial gaps and aspect ratio have been highly reduced. Those geometrical constrains, combined with flow phenomena like shock waves, wake passages, secondary flows, or potential interactions confer to axial turbomachinery flow an inherent unsteady character.

Several aspects concerning the performance, safety and efficiency of the whole turbomachine are affected by the presence of unsteady phenomena. For instance, the aerodynamic force on the blades, the efficiency of the stage, the noise generation, or the energy transfers are directly dependent on unsteady flow phenomena (Aubert et al., 1995). The basic mechanisms concerning generation, development, transport, interaction, and dissipation of these unsteady phenomena are still not enough understood and all complex effect are therefore not taken into account during the early design process.

As a result, the development of new turbomachines comes up against a deeper understanding of unsteady phenomena, which reduce the overall efficiency of the engine, prematurely reduce the lifetime or even sometimes lead to fatigue failure of certain components of the engine. In some extreme cases, the interaction between unsteady phenomena and the structural response can lead to cracks or sudden and brutal ruptures of some parts of the turbomachine. Financially, the discovery of potential risks of failure has critical consequences for a manufacturer, especially when the engine is already at a later stage of development and fails the qualification test. Undoubtedly, the worst type of failures are those that occur although the engine has successfully passed the qualification tests and is already in production or service. Such was the case for instance when two in-flight failures occurred on F101’s General Electric engine and grounded the B1’s fleet in December 1990 (Aerospace-America, 1991). As a result, the interest to understand unsteady flows and, more important, their interaction with steady state flows has thus tremendously increased during the last decennia.

The aim of the present work is to participate in the overall understanding of unsteady flow phenomena and their interactions with mean flow structures in axial turbomachines.
CHAPTER 1. INTRODUCTION

Among the vast possibilities of configurations, this study focuses on the interaction between upstream propagating pressure waves and a fully three-dimensional transonic flow structure. With the aim at understanding realistic configurations such as potential interactions in axial turbomachines, the test section of a wind tunnel was instrumented with test objects, so called "bumps", in order to reproduce three-dimensional flow structures similar to ones encountered in axial transonic turbomachines. In such configuration, at a particular operating point, a complex three dimensional shock wave impacts on the boundary layer of the suction side of the blades and eventually spreads itself over the whole interblade passage and reaches the pressure side of adjacent blades as well. Considering the natural characteristics of boundary layers, the interaction is non-linear and often leads to a low amplitude shock oscillation, which also corresponds to an oscillation of the aerodynamical resultant force acting on the blade. Furthermore, pressure fluctuations, which originate the relative passing motion of downstream blade rows, propagate upstream in the blade passage and put the shock wave into large amplitude oscillating motion with possible time lags between unsteady flow structures and the resulting aerodynamical force on the blade. At this point, the mechanical response of the structure is crucial and a too low structural damping for a critical phase lag could have dramatic consequences on the stability.

Therefore, the present study has been organized as following: The second chapter presents the general background by introducing and successively description the different phenomena and problems of interest. First, a description of unsteady flow phenomena in axial turbomachines is given together with an introduction to aeroelasticity. Then a brief description and classification of a particular type of aeroelastic problems in relation with the present work is presented. The general properties of turbulent boundary layers are thereafter described as an introduction towards Shock/turbulent Boundary Layer Interaction (SBLI). Considered as the main part of the second chapter, different types of viscous interaction, whether it occurs on airfoils or flat plate and with or without separation, are presented and discussed. A brief introduction of the acoustic blockage theory, an amplification phenomenon of upstream propagating acoustic waves in near sonic flow conditions first identified by Atassi et al. (1994b), is thereafter presented. And finally, a short literature survey on unsteady SBLI in transonic nozzle is presented as a summary of previous work on the subject.

The third chapter is dedicated to the formulation of the different motivations for the present work, logically followed by the method of attack presenting the overall perspective of the study and the different steps taken to reach the objectives formulated above.

As the present research work was conducted both on an experimental and numerical levels, chapters four and five are dedicated to the description of the Computational Fluid Dynamic (CFD) tool and the experimental facility respectively. Briefly, the analysis was performed on simple models such as convergent-divergent nozzles, which have the advantage to avoid leading and trailing edges influences, as well as inter-rows region interactions. Instead, the study can focus the analysis on the essential flow features described in the first part of the introduction.

Numerical simulations were performed using the CFD tool, so called PROUST, devel-
oped at Ecole Centrale de Lyon, France, by Aubert (1993), and Smati (1997). The code solves the fully unsteady three-dimensional (3D) compressible Reynolds Averaged Navier-Stokes (RANS) equations, using a finite volume formulation and a linear two-equations turbulent model originally developed by Wilcox and Traci (1988). A brief description of the code is presented in chapter 4 and more complete presentation of the different the numerical methods and computational options used in the present work are further described in Appendix A.

Experiments were performed in the VM100 wind tunnel at the Royal Institute of Technology, Sweden. In order to comply to the needs of the present study, the test section was entirely redesigned in a modular way and now features easy accesses for instrumentation, flow visualizations, and test object exchange. The air flow supply consists of a 1MW continuously running screw compressor with a maximum mass flow capability up to 4.7 kg/s at 4 bar and 30°C, an air cooler, and a set of valves which allows the control of both the Reynolds and Mach numbers within the test section.

Chapter 5 presents a complete description of the experimental apparatus, including the redesign of the test section, the test object design objectives and instrumentation, the measuring techniques, acquisition methods, data reduction, and a discussion on measurement accuracy and source of error.

Experimental and numerical steady state results are presented in Appendix M and analyzed in chapter 6. Different measuring techniques were used to monitor the quasi-steady shock position, the pressure distribution, separation region and heat exchange coefficient on the test object surface. Both simulations and experiments were performed over a wide range of operating conditions, from subsonic to transonic and fully choked flows, and compared with each others. Different shock strengths were therefore studied from a steady state point of view together with the overall flow structure in the wind tunnel channel.

Unsteady results are then presented in Appendix N and analyzed in chapter 7. Although a limited number of configurations both for numerics and experiments were investigated, a parametric study was performed regarding the amplitude and frequency of the perturbations. Basically, different steady state flow fields were identified as interesting regarding the propagation and amplification of upstream propagating pressure waves and therefore used as mean flow conditions for unsteady investigations. The unsteady pressure distribution and shock wave motion were monitored and cross analyzed for perturbation frequencies up to 500Hz. Numerical results were also analyzed and compared with experimental data. Harmonic analysis was mainly used in the present study so that both the amplitude and phase of the unsteady pressure distribution and shock motion could be investigated as a function of the amplitude and frequency of the downstream perturbations.

Finally, chapter 8 summarizes the main results of the thesis and presents the recommendations for future work, both on a numerical and experimental levels in order to further understand the complex SBLI in transonic flows.
Chapter 2

Background

Current and future designs of rotating machines with higher aero-loading, more compact configuration compounded by cost reduction and other non-aerothermal constrains require enhanced understanding of unsteady flow effects as well as their interaction with other flow phenomena in axial turbomachines.

In the following sections a non exhaustive overview of the different flow phenomena and interaction related to the present study is presented.

2.1 Unsteady phenomena in axial turbomachinery flow

Unsteadiness is a natural phenomenon, inherent to the turbomachine, and many aspects concerning the performances can directly be influenced by unsteady flow phenomena and their interactions with other flow structures. Namely, aerodynamic load on blades, stage efficiency, heat transfers, noise generation, or energy transfers are directly influenced by unsteady flows.

Basically, unsteady flows can be divided into periodic and non-periodic phenomena, as illustrated in figure 2.1. Non periodic phenomena can originate from the transient regime of the turbomachine (rotating speed variations or inlet distortions) or the inherent chaotic nature of the flow (wakes, jets, trembling shock waves or boundary layers). Similarly, unsteady periodic flows can be distinguished depending on whether they can be correlated to the rotating speed of the engine or not. As an example, Von Karman vortex shedding, flutter or surge phenomena are decoupled from any passing blade frequency and rather originate some flow properties or complex interactions between different parts of the turbomachine. The majority of unsteady periodic flows investigated nowadays are, however, directly correlated to the rotating speed of the engine. Phenomena like rotor-stator interactions, potential interactions, wakes passages, secondary and 3D flows, as well as oscillating shock waves are characterized by stable operating conditions and easier to observe, than for instance rotating stall.

Among these phenomena, we will be interested in potential interactions, and how acoustic waves propagate, reflect and interact with non linearities present in the flow such as shock waves. The lambda shock region, where the shock impacts the boundary layer
CHAPTER 2. BACKGROUND

Figure 2.1: Unsteady phenomena in axial turbomachines (Oliveira, 1999)

on the surface of a blade, is also of high importance since it is source of strong interaction which is still not well understood and therefore impossible to model and predict. As an example, potential interactions in a transonic flow result in travelling acoustic waves which propagate and interact with the shock wave. The oscillating shock thereafter generates boundary layer fluctuations and results in unsteady loads on the blade and higher losses which lower the efficiency of the whole engine.

Additionally unsteady phenomena often interact with other flow structures and sometimes become part of a complex coupling which drives the blades into vibrations. It is, however, not really clear where the initial perturbation comes from (structural vibration or flow perturbation). Such kind of phenomena thus depend both on the flow unsteadiness and on the mechanical properties of the structure, and can lead either to fatigue failure, or to a brutal destruction of the blades whether the vibration is damped, sustained, or amplified.

Those latter phenomena, extremely complex, result from the strong coupling of forces issued from the dynamic and cinematic properties of the flow and the structure. They belong to the class of aeroelastic phenomena.

2.2 Aeroelasticity

2.2.1 General introduction

Vibration-related failures of structures are of a large importance in many aspects of mechanical engineering. Problems may appear as a sudden destruction or long-term fatigue of the structure. They can often be related to an eventual flow of fluid around the struc-
2.2. AEROELASTICITY

ture (called for example "flow induced vibrations", "flow-structure interaction" or "fluid-elasticity") or be purely structural. It is widely recognized that flow induced vibrations are of major concern in the design of modern engineering structures, as the continuous interaction between a vibrating structure and the changing flow characteristics can cause high stresses leading to high cycle fatigue (HCF) failure. A non-optimal development of a structure can, especially in aeronautics, where development costs are extremely high, have a significant influence on the viability of the final product.

Flow induced vibrations appear in many circumstances in nature and in different engineering concepts. Civil engineering structures, such as bridges and tall buildings, are typical constructions where flow induced vibrations must be taken into account. Flow induced vibrations are of major concern in the design of modern tube and shell heat exchangers (the problem is especially critical in nuclear steam generators that often are designed to last 30 years or more). Blades in hydraulic and thermal turbomachines (both axial and centrifugal flow machines) are subject to large time-dependent variations in the oncoming flow. In many cases unsteady flow effects and induced vibrations lead to high noise levels, which can today be of major environmental concern.

"Aeroelasticity" is an engineering terminology that defines an inter-disciplinary field which combines aerodynamic, inertia and elastic forces in such a way that the structure and the flow around it interacts with each other. Collar Collar (1946) defined a triangle of forces in which each of the inertial, elastic and aerodynamic forces occupies a vertex (Figure 2.2).

![Figure 2.2: Collar’s triangle for axial turbomachines (Collar, 1946)](image)

In the case when only the aerodynamic and inertia forces act together the static aspects of the loading on the structure are considered ("rigid-body aerodynamics"). When the aerodynamic and elastic forces are taken into consideration, the resulting problem is defined as a "static aeroelasticity", whereas the domain "mechanical vibrations" considers the relationship between the inertial and elastic forces. Finally, a fourth domain takes into account all three forces (aerodynamic, inertia and elastic forces), and is usu-
ally called "dynamic aeroelasticity". In a general sense the term "aero-elasto-dynamics" (Platzer, 1990) would thus describe the phenomena involving all three forces more accurately, whereas the term "aero-elastic" should be reserved for what is generally today known as "static aeroelasticity". However, "aeroelasticity" is the generally acknowledged terminology today.

As stated by Fransson (1999), as long as the turbomachine is "flutter-free" the designer is generally not very interested in such a problem. But when a failure does arrive a considerable project is usually started, but gradually gets a lower priority if the failure does not repeat itself in another machine. Although the situation has considerably improved the last decade, the statement still holds although the field is continuously of large technical, scientific and commercial interest. Indeed, over 90% of the potential HCF problems are uncovered during development testing, the remaining few problems account for nearly 30% of the total development cost and are responsible for over 25% of all engine distress events. It is also mentioned (Kielb, 1998) that every new development program for jet engines have about 2.5 serious HCF problems. Estimates (Kielb, 1998) indicate that billions of $US will be spent on HCF problems in the US Air Force only up to year 2020.

Aerodynamically induced vibrations in turbomachines can be classified into two general categories, namely flutter and forced vibrations (Forsching, 1974). In the first case the unsteady aerodynamic forces are dependent on the structural motion and the flow, i.e. the unsteady pressure distribution supplies energy to sustain the motion. In the latter case the aerodynamic forces are independent of that motion and are periodic in time.

### 2.2.2 Dynamic aeroelasticity

The terminology "dynamic aeroelasticity" covers, as stated above, the interaction of aerodynamic forces, inertial forces and elastic forces in Collar’s triangle of forces (see figure 2.2).

#### 2.2.2.1 Basic description of flutter phenomenon

The term "flutter" is the generic name of a wide range of instabilities caused by the coupling between aerodynamic, elastic and inertia forces. Flutter is basically a self excited vibration of the structure by aerodynamic. This phenomenon strongly depends on the flexibility of the structure and appears when the mechanical work is lower than the aerodynamic work, i.e. when the mechanical damping is small to overcome the aerodynamic excitations.

In external flow, classical flutter appears when there is an interaction between two vibrating modes (usually bending and torsion) at distinct frequencies. In turbomachines, where the stiffness of the blades is much higher, flutter is more often due to the interaction between a vibrating mode (bending or torsion) and an unstable aerodynamic behavior, like a boundary layer separation, a strong shock motion, or a shock/boundary layer inter-
action. It is however really difficult to predict which phenomenon originates flutter.

The above explanations are nevertheless non exhaustive in the sense that the aerodynamic phenomena described above are not required to generate flutter. It can indeed appear whereas there is no aerodynamic instability but an interblade phase lag favorable to an amplification of any structural vibration. Here, it is the flow configuration over the whole stage that can create a force in phase with the vibrating motion of the blades.

In any case, flutter exists because of a strong interaction (or coupling) between the instant motion of the blade and the instant aerodynamic forces. The unsteady aerodynamic perturbations can be excessively small but they might act like a catalyst.

### 2.2.2.2 Flutter classification in axial flow turbomachine

The terminology found in the classical literature on aeroelasticity englobes a wide variety of terms. The underlying physical reasons for a specific phenomena may be similar, although the terminology is diversified. As an example, an operating map of a multistage compressor is schematically illustrated in Figure 2.3. A few operating characteristics are shown, together with the surge and choke lines. If an attempt to operate the compressor above the surge line (i.e. at high positive incidence angles) is made, high pressure amplitudes of low-frequency will result as the compressor goes into surge. Decreasing the pressure level at a certain mass flow, on the other hand, will result in low positive or negative incidence angles with an associated choking of the flow. The compressor will then not accept, at a given mass flow, a further decrease of pressure ratio.

![Figure 2.3: Classical flutter](image)

Subsonic/transonic (at positive incidence) flutter and the classical supersonic flutter
are considered to be the most important types, which can be found in the literature. The first one occurs usually when a compressor is operating near the surge or stall-limit line at partial speed/design speed. Fans and the front stages of the compressor section of a gas turbine are affected. The flow conditions are characterized by high incidences and separated flow. The governing parameters are Mach number, reduced frequency and the angle of incidence. The vibratory modes are bending, torsion as well as coupled modes (Srinivasan, 1997).

Classical flutter (potential-flutter) occurs for small angles of incidence. The reason is the phase shift between the blade motion and the pressure on the blade. The phase-angle between the blade motion and the integrated force resulting from the unsteady pressure on the blade is greater than 180° and smaller than 360°. For these conditions energy is transferred from the fluid towards the blade and the oscillation of the blade is amplified (Böles and Suter, 1986). Classical flutter can happen at the operating point.

Negative incidence flutter, also referred as to choke flutter, is encountered during part speed operation when the blades are operating at negative angle of incidence. Choke flutter can affect mid and aft stages of compressors. The governing parameters are the Mach number, the reduced frequency and the incidence angle. The vibratory modes are bending or torsion modes (Srinivasan, 1997).

Classical supersonic flutter generally occurs in thin fan bladings when the outer span of each rotor blade is operating at a supersonic relative but subsonic axial inlet Mach number. It can impose a limit on the high-speed operation of the machine.

Shock flutter occurs in transonic flows. Transonic flow conditions can be reached in compressors by increasing the inlet Mach number and for high angles of incidence. In turbines shock flutter can be reached for high outlet Mach numbers. For such flow conditions, shock boundary layer interaction can induce separation.

Supersonic shock flutter (at low back pressure) is placed near the operating line on the compressor map and can also impose a limit on high-speed or over-speed capability. It occurs in fans in which the outer sections operate at supersonic relative Mach number and results in high stresses. This flow condition, where the flow is supersonic and attached, occurs on fan blades. The dominating parameters are the Mach number, the reduced frequency, the interblade phase angle and shock position. The vibratory modes are bending and pitching. For supersonic flutter the turbine is usually choked.

Supersonic high-incidence or stalled flutter (at high back pressure) occurs during high speed operation. The outer part of the blades is then operating supersonically and the stage is operating near the surge line. The affected components are usually the fan blades of the compressor section of a gas turbine. The blades are highly loaded and strong shocks occur. The important parameters are the Mach number and the reduced frequency.

Potential flutter in turbines occurs near the operating point, for small angles of incidence. The reason is the phase shift between the blade motion and the pressure on the blade.
As already mentioned, when flutter is addressed in turbomachines, it is often argued that due to a large difference in densities between the blade material and the fluid, the effect of the aerodynamic force on the mode and the frequency of the blade motion is negligible (Verdon, 1986). Therefore, the structural part and the fluid dynamic part may be considered to be decoupled. Hence, the problem can be investigated by finding the aerodynamic forces due to prescribed blade motions.

### 2.2.3 Forced response

Another cause of turbomachinery blade failure is HCF due to excessive resonant response, namely forced response. Unsteady aerodynamic forcing functions emanating from inlet distortion, wake-rotor or potential interactions between adjacent blade rows are the main source of aerodynamic excitations. Excessive resonant response occurs when the frequency (and circumferential shape) of the excitation matches the frequency (and mode shape) of a natural mode of a single blade or the entire bladed-disk assembly.

The unsteady aerodynamic forcing functions emanating from the core flow of a turbomachine have been tediously described by Kielb and Chiang (1992) and may be grouped into three classes:

- **Flow unsteadinesses due to off-design operating flow conditions:**
  
  The design procedures usually consider nominal operating conditions. However, aeroengines operate at varying mass flow and pressure ratio and in particular at off-design flow conditions, often for a short instant of time (at takeoff for example). These “critical operating conditions” result in the apparition of “critical” flow phenomena, including surge, rotating stall, cavitation, flutter, and aeroacoustic phenomena.

- **Flow unsteadinesses resulting from transient operating flow conditions, which may appear due to:**
  - Variation in the spinning rotor speed
  - Variation in the inlet/outlet aerodynamic conditions including inlet gust, transition in the combustion process leading to variation at the inlet of the HP turbine stage, etc...

- **Flow unsteadinesses caused by rotor-stator interactions, including:**
  - Potential interaction: The potential distortions are linear, isentropic, irrotational perturbations of the steady state flow field. This interaction is important when the stator rotor gap is small. The effects of the potential interaction can propagate about one chord upstream and downstream of a blade row.
  - The wake/rotor interaction: The creation of wakes is essentially a visous process, however the interaction between the wakes and the downstream blade rows is essentially an inviscid process. The wakes propagate downstream considerably much more than the potential interaction does.
  - The von Karman vortices/rotor interaction: For sufficiently high Reynolds number, vortices appear at the trailing edges of the blades. Those downstream propagating
vortices are known as the von Karman vortices. The time and length scale of these vortices depend on the trailing edge radius and free stream velocity.

- The shock/rotor interaction: Namely, the shocks propagate and reflect on the surface of the blades.
- Stator secondary flow/rotor interaction.

Most commonly, the Campbell diagram (Campbell, 1924) is used to predict the occurrence of resonances in the operating range of a turbomachine. However, this approach has two main drawbacks. First, it is only adequate if all resonances can be avoided which is most generally not the case. Secondly, it does not quantify the amplitude of the blade response. There is therefore a lack to quantify the severity of a resonance which can not be avoided. A more adequate prediction scheme is to predict the amplitude of the aerodynamic forcing function using an unsteady aerodynamic model, and the amplitude of the blade response using a structural dynamic model.

According to Srinivasan (1997) a rational procedure to calculate flutter and resonant characteristics of blades requires advances in:

• assessment of flow deflects at the location they originate and as they are transported along,

• unsteady aerodynamics of cascades under a wide variety of flow conditions expected in the operating range,

• structural vibration frequencies and mode of interest over the operating range,

• quantification of damping in the system due to non aerodynamical sources,

• estimation of material properties (fatigue strength, ultimate strength, modulus of elasticity, etc.) for the configuration at the temperatures expected in the operating range, including the influence of processing, defects, etc., leading to the calculation of structural integrity,

• and finally, identification of dissimilarities in aerodynamic parameters (gap/chord, stagger, incidence, etc.) and accounting for their influence in a statistical sense.

The unsteady aerodynamics needed are of two types: the first type leads to calculations of aerodynamic damping and the second type predicts pressures that act on blades (assumed to be not vibrating) due to flow variations in the stream leading to forcing functions on the blade. The nature and extent of aerodynamic damping, superposed on the corresponding contributions from non-aerodynamic damping, determines the stress levels expected in resonant vibration and establishes susceptibility to flutter.

Clearly, the calculation of the aerodynamic forces depends upon the geometry (i.e. profile of blades, cascade solidity, stagger angle, incidence, etc.) and the flow conditions (i.e. subsonic, transonic or supersonic). Already quite complex in simple configurations (such as, for example, a flat plate cascade at zero incidence in a subsonic flow), the complexity of computational aspects further increases when considering actual blade profiles.
and fully unsteady 3D turbulent flow conditions in which shocks and separations can occur within a passage.

2.3 Acoustic blockage theory

Transonic flows about streamlined bodies are strongly affected, particularly near the shock location, by unsteady excitations. Experimental and computational studies (David and Malcom, 1979; Atassi et al., 1994a) have shown that the unsteady pressure distribution along the surface of an airfoil or of a cascade blade in unsteady transonic flow exhibits a significant bulge near the shock location. For a single airfoil, this phenomenon was examined by Tijdeman (1977) and David and Malcom (1980). Experiments have shown that the pressure rise can be much more important than in quasi-steady case. As a result, it gives a significant contribution to the overall unsteady lift and moment. It also affects the flutter boundary of the airfoil or blade system and causes large local stresses which may result in HCF failure. Tijdeman and Seebass (1980) have also reported that the unsteady pressure bulge and its phase variation result from non-linear interaction between the mean and unsteady flow. This non-linear interaction causes a shift in the shock position which produces the observed large bulge in the unsteady pressure. Moreover, some other studies (Ferrand, 1986) on shock flutter have shown that, in unsteady transonic flows around a single airfoil, the shock motion, and then the pressure distribution along the surface, can be critical regarding to the self-exciting oscillations of the airfoil. It was also found that the mean flow gradients are of high importance regarding the imaginary part, and then to the time response, of the unsteady pressure distribution on the airfoil surface.

For internal flows, Atassi et al. (1994a) has reported results from linear and non-linear computations for two-dimensional cascades in oscillatory motion. It was found that in transonic flows, the unsteady surface pressure always exhibits a strong bulge near the shock location. However, Atassi et al. have shown that for subsonic flows this surface pressure bulge also occurs when the blade surface mean flow approaches the critical sonic velocity and that it is located near the maximum mean velocity. It was also found that the level of the pressure bulge is significantly reduced as a downstream propagating mode cuts on, i.e., when perturbations do not propagate any longer for a certain frequency range. They, therefore, suggested that this sharp rise in the unsteady pressure is due to the near sonic condition. The near-sonic velocity acts as a barrier they called acoustic blockage preventing acoustic disturbances from propagating upstream in a similar way to the shock in transonic flows.

A transonic convergent-divergent nozzle experimentally investigated by Ott (1991); Ott et al. (1993) was thereafter used as a model to investigate the non-linear acoustic blockage effect. The reasons to investigate such simple geometry were to separate the different mechanisms and avoid other problems like acoustic emission at the leading and trailing edges, or acoustic interactions and reflexions in complex geometries. Analytical and numerical computations (Ferrand et al., 1995) using unsteady non-linear Euler CFD tool were then carried out to analyze and quantify the upstream and downstream propagation of acoustic disturbances in the nozzle. The results confirmed the sharp rise of
the upstream propagating pressure disturbances at the nozzle throat as a result of the acoustic blockage. It was also found that in a choked case, outlet pressure perturbations are amplified but blocked by the strong shock. In the other hand, inlet perturbations are convected without amplification through the shock wave. However, in both cases of inlet and outlet perturbations, the shock motion was found purely harmonic.

An analytical analysis was carried out by Ferrand et al. (1995) to determine the sensitivity coefficients of the acoustic blockage. A special treatment was applied to the 1D, unsteady Euler equations in their conservative form, so that it was possible to determine the amplification factors in case of inlet or outlet pressure perturbations. It was thereafter found that upstream propagating perturbations are strongly amplified when the local Mach number is close to one. At a lower level, both inlet and outlet disturbances are amplified when the local Mach number is close to zero, which is a typical situation for a leading and trailing edge. The perturbation’s frequency has a negligible influence on the amplification factors but a slight effect on the phase lag between the pressure fluctuations and the velocity fluctuations. Moreover, the analysis of the analytical expressions showed that the streamwise velocity gradients have the same effect as the inverse of the frequency. On the other hand, when the velocity becomes sonic at the throat, the differential system distorts itself into an algebraic equation between the unsteady variables. Therefore, when the flow stays subsonic, the velocity gradient is zero at the throat and the amplification factor between the pressure fluctuations and the velocity fluctuations is real. But when the flow becomes transonic, the velocity gradient is not zero any more and the amplification factor is complex, introducing a filter and a phase lag at the throat. As a concluding remark towards the turbomachines, Ferrand assumed that the existence of a circumferential periodicity in the turbomachine might be the link between the reduce frequency and the circumferential acoustic propagation modes. Indeed, depending on the reduce frequency, some acoustic modes propagates and some are damped.

Further numerical studies were carried out by Smati et al. (1996); Smati (1997) to analyze the acoustic blockage in unsteady turbulent transonic flows using the same nozzle test case. Different flow conditions from a subsonic case to a completely choked nozzle, using a simple mixing length turbulence model were computed to investigate this phenomenon. All results were in good agreement with the experiment conducted by Ott et al. (1993) regarding the harmonic shock motion. The results also confirmed the acoustic blockage theory, and showed that viscous simulations are much more complex because of two dimensional interactions between steady and unsteady flows. More especially, the SBLI was investigated carefully in its unsteady behaviour. It was found, from pressure signals probed in subsonic and supersonic regions, a strong non-linear behaviour associated with the emergence of higher modes. This production of higher harmonics was believed to be a result of the viscous effects since it was not found in the Euler simulations. It was also described how the lambda shock structure interacts with the BL and the separated zone to produce high time harmonics and phase lags. It was assumed that the outlet disturbances propagate upstream and interact in the the lambda foot region with the BL to create higher harmonics (which means a non-linear interaction between steady and unsteady flows), and then reflect on the walls to interact again in the main flow. Moreover, a non-linear behaviour of the separated zone could be noticed as the separation point and the reattachment point were found to oscillate in a non-linear way including a phase lag.
with the shock wave motion. Finally, it was assumed that depending on the shock wave strength, the shock/boundary layer interaction or the acoustic blockage is prevailing. In any case, phase lags between the shock motion and the different phenomena were found to be of high importance and have to be further investigated.

2.4 Shock/boundary layer interaction

2.4.1 General introduction

Shock wave/boundary layer interactions are ubiquitous in high speed flight, occurring in an almost limitless number of internal and external flow problems relevant not only to turbomachines but also aircraft, missile, rockets or projectiles. Maximum mean, fluctuating pressure levels and thermal loads that a structure is exposed to are generally found in region of shock/boundary layer and shock/shear layer interaction and can effect vehicle and component geometry, structural integrity, material selection, fatigue life, thermal protection systems, weight, and cost. Despite truly remarkable progress in computational and measurement capabilities there are still important quantities which cannot be predicted accurately (i.e. peak heating in strong interactions) or cannot be predicted at all (i.e. unsteady pressure loads), observations which cannot be satisfactorily explained, and physical processes which are not well understood.

2.4.2 Structure of turbulent boundary layers

At high Reynolds number, a fully developed turbulent flow features a three-dimensional (3D) and rotational velocity field, enhanced mixing capabilities, high dissipation rates, intermittent phenomena, and inherent chaotic characteristics. The turbulent mixing process can be seen as a coexistence of an infinite number of flow structures (eddies) with different length scales over a continuous spectrum. Assuming those scales statistically independent, it is shown that the motions of the small scales depends on the energy supply from larger scales and on the kinematic viscosity, and that the rate of this energy supply equals the rate of dissipation (Kolmogorov’s universal equilibrium theory (Kolmogorov, 1942)). Basically, large scales which originate the mean flow, transfer energy to the smaller scales (Kolmogorov’s scales) which dissipate it into heat. The dynamic of this interaction between large and small scales is non-linear and due to this turbulence features a high sensibility to initial and boundary conditions.

2.4.2.1 Equilibrium turbulent boundary layers

The flow velocity across a boundary layer decreases from high external value to zero at the wall where the no slip condition must be satisfied. The characteristic velocity scale through which the evolution of different flow quantities in the boundary layer can be described is the friction velocity $u_\tau = \sqrt{\tau_w/\rho}$ where $\tau_w = \mu (\partial u/\partial y)_{y=0}$ is the wall shear stress. The different flow quantities are then described using dimensionless variables (also
called viscous scaling): \( u^+ = u/u_r \) and \( y^+ = y.u_r/\nu \).

![Structure of an equilibrium boundary layer (Schlichting, 1979)]

Experimental observation (Cousteix, 1989) shows that a boundary layer under a zero pressure gradient (ZPG) has a composite nature and can be "decomposed" into different layers (figure 2.4):

- A viscous sub-layer very close to the wall \((0 < y^+ < 5)\), in which molecular viscosity is predominant compared to turbulence effects. In this region, the velocity profiles are self-similar, linear in the sense of viscous scaling and described by \( u^+ = y^+ \).

- A "buffer" zone \((5 < y^+ < 40 \sim 60)\) where the turbulent and viscous effects have the same order of magnitude.

- A "log" region \((40 \sim 60 < y^+ < 300 \sim 600)\), fully turbulent, where the viscous effects are negligible. In a region of constant total shear-stress \( \tau^+ \equiv \frac{\partial U}{\partial y} + \bar{u}'\bar{v}' = 1 \), the mean velocity profile is described by

\[
\frac{u^+}{U_\infty} = k^{-1} \ln(y^+) + C \quad \text{with} \quad k = 0.41 \quad \text{and} \quad C = 5 \tag{2.1}
\]

This is referred to as the logarithmic law of the wall. However, in Adverse Pressure Gradient (APG) flow, the pressure gradient is not zero and the equation for the inner region scaled with \( u_r \) has the form \( \tau^+ = 1 + \lambda y^+ \) with \( \lambda = \left( \frac{u_{ps}}{u_r} \right)^3 \) and \( u_{ps} = \left( \frac{\nu}{\rho} \frac{\partial P}{\partial x} \right)^{1/3} \). The influence of the pressure gradient on the total shear stress is reflected in \( \lambda \), the ratio between a viscous pressure gradient velocity scale \( u_{ps} \) and \( u_r \).

- An outer region where the flow is fully turbulent and where the outer pressure gradient plays an important role (see figure 2.5). The dynamic of the flow is then governed by the large turbulent eddy structures. This region is usually scaled in velocity defect form:

\[
\frac{U_\infty - U}{u_r} = F(\eta), \quad \frac{-\bar{u}'\bar{v}'}{u_r^2} = G(\eta), \quad \eta = \frac{y}{\Delta}, \quad \Delta = \frac{\delta^* U_\infty}{u_r} \tag{2.2}
\]
For APG boundary layer, assuming solutions on this form and neglecting the viscous terms \((Re \to \infty)\), the turbulent boundary layer equation leads to:

\[
-2\beta F - (1 + \beta) \frac{\partial F}{\partial \eta} = \frac{\partial G}{\partial \eta}
\]

with \(\beta = \frac{\delta^* \partial P}{\tau_w \partial x}\) \((2.3)\)

The mathematical criterion for similarity solutions to exist is that the parameter \(\beta\) is constant. \(\beta\) represents a ratio of the pressure gradient and the wall shear-stress. With increasing \(\beta\), the influence of the pressure gradient is increasing. \(\beta\) has a similar role as \(\lambda\) in equation 2.1 and the ratio between these two parameters are the ratio between an outer \((\delta^*)\) and the inner \((\nu/u_t)\) length scale. A turbulent boundary layer which is self similar in this manner is said to be in equilibrium and does not depend on local or upstream influences. It is said in such case that the turbulence has no history, no memory.

Further formulations and discussions about the velocity scaling within ZPG or APG boundary layers, with or without separation, can be found in (Angele, 2003; Cousteix, 1989; Schlichting, 1979; Simpson et al., 1981; Elsberry et al., 2000; Dengel and Fernholz, 1990).

The region in the vicinity of the wall, where strong normal velocity gradients are located, is subject to both high turbulent energy production and dissipation rates. The energetic balance shows that those terms are of the same order of magnitude in the largest part of the boundary layer. When the turbulent energy production rate equals the dissipation rate, the BL is said to be in energetic equilibrium.

### 2.4.2.2 Historical effects

As mentioned above, an equilibrium turbulent boundary layer has no history. It has no memory of local or upstream influences. However, a boundary layer developing towards separation is not in equilibrium and is continuously changing. Various attempts have been made to try to overcome this problem and establish the proper scaling which would predict self similar profiles within the boundary layer. A linear combination of the logarithmic law of the wall and an outer wake profile based on empirical evidence was for instance developed by Coles (1956). This scaling was proved to be successful in moderate pressure gradients, where the logarithmic region is still present, but it has been shown to be less successful as separation approaches (Dengel and Fernholz, 1990). Indeed, the problem when it comes to a developing turbulent boundary layer is that the flow usually "suffers" from historical effects. The profiles keep the "memory" of upstream and local flow gradients. The stronger the APG is, the more pronounced the historical effect becomes. A short literature survey about the different scaling and solutions in order to lower the historical effects of turbulent boundary layers is presented in (Angele, 2003; Skote, 2001).
2.4.2.3 APG boundary layer effect

The flow under an Adverse Pressure Gradient (\( \frac{\partial P}{\partial x} > 0 \)) is decelerated. The turbulent boundary layer thickens and the skin friction decreases. The standard logarithmic law remains valid close to the wall but is rapidly put into defect in the outer part of the BL as illustrated in figure 2.5. The size of the "log" region decreases as the APG increases (outer \( y^+ \) value rather close to 300 than 600).

![Figure 2.5: Illustration of APG effect, modification of the "log" and outer region](image)

The fact that the static pressure is constant throughout the BL in the direction normal to the wall gives rise to a larger deceleration close to the wall where the flow carries less momentum.

The skin-friction coefficient \( C_f = \tau_w / \frac{1}{2} \rho U_\infty^2 \) based on the wall shear-stress, \( \tau_w \), decreases as a consequence of this. This also implies that the shape of the profile is changed (see figure 2.6), best displayed in terms of the increase in the shape-factor, \( H_{12} = \delta^*/\theta \), based on

the displacement thickness

\[
\delta^* = \int_0^\infty \left( 1 - \frac{U(y)}{U_\infty} \right) dy, \tag{2.4}
\]

and the momentum-loss thickness

\[
\theta = \int_0^\infty \frac{U(y)}{U_\infty} \left( 1 - \frac{U(y)}{U_\infty} \right) dy. \tag{2.5}
\]

The largest gradient in the mean velocity profile moves out from the wall as the flow develops towards separation as illustrated in figure 2.6(a).
2.4. SHOCK/BOUNDARY LAYER INTERACTION

(a) BL velocity profiles
(b) BL $U_{\text{rms}}$ profiles

Figure 2.6: Illustration of APG effects on BL profiles (LDV measurements (Angele, 2003))

This completely changes the character of the flow. The near-wall turbulence generation is weakened and the spanwise spacing of the sub-layer streaks increases (Skote, 2001). The total shear-stress distribution, $\tau$, which is the sum of turbulent and viscous shear-stresses $\tau = -\rho u'v' + \mu \frac{\partial U}{\partial y}$ is also modified by the APG as illustrated in figure 2.7.

(a) Total shear-stress under ZPG
(b) Total shear-stress under APG

Figure 2.7: Experimental profiles of turbulent and total shear-stress on a flat plate BL (Cousteix, 1989)

Schubauer and Spangenberg (1960) investigated the effect of different pressure distributions on the boundary layer development, separation and pressure recovery. They observed that an initially steep and progressively relaxed APG gives the highest pressure recovery in the shortest distance. This implies that the boundary layer can withstand a stronger pressure gradient at an early stage when it is not yet affected but becomes less resistant as the profile has been changed. If the pressure gradient is strong and persistent the flow ultimately shows similarities to a mixing layer and separates. APG induced separation is a continuous process, with intermittent instantaneous back-flow upstream of the mean separation point (as further discussed in the forthcoming sections), in opposition to the case where the flow separates at a sharp corner. According to the extensive review by Simpson (1989), steady two-dimensional separation is defined by $C_f = 0$ and $\gamma_P = 50\%$ where $\gamma_P$ is the "back-flow coefficient" (or intermittency parameter) in the vicinity of the wall, which is defined as the amount of time (with respect to the total time) the flow spends in the upstream direction.
2.4.3 Shock/turbulence interaction

In order to better understand some of the basic phenomena in the interaction between a shock wave and a turbulent boundary layer, it is useful first to consider the ideal situation in which an isotropic turbulent flow field interacts with a plane shock wave.

From an analytical point of view, studies conducted in the 50s (Ribner, 1952, 1954; Moore, 1963) mainly concentrated on investigating the interaction of a pattern of vorticity and noise through a plane wave. Aerodynamic modes like acoustic, vortical, and entropic waves can experience considerable amplification on passage through a shock (Hardy and Atassi, 1997), while downstream of a shock there exists a critical angle for incident acoustic modes where the reflection coefficient is 1. The above cited analytical studies assumed that the incident, radiated and resulting perturbations, including the shock front distortion, are of small amplitude, allowing a linearized description. However, not addressed by these papers is the nonlinear response of the shock wave to the incident turbulence, nor the conversion of mean shock wave momentum and energy to the amplification of the incident turbulence. As shown by more recent studies (see following paragraph), upstream turbulence can actually affect the mean flow variables of a shock wave significantly.

Results from analytical studies performed in the late 50s are nowadays completed with experimental investigations and numerical simulations. Several measurements (Trollier, 1985; Honkan and Andreopoulos, 1990; Honkan et al., 1994; Barre et al., 1996) of the interaction between a shock wave and a decreasing homogeneous and isotropic turbulence generated from a grid are available for comparison with advanced Direct Numerical Simulations (DNS) performed in 2D (Rotman, 1991; Hannappel and Friedrich, 1993) and 3D (Lee et al., 1993, 1994, 1997) geometries. According to the different results, the interaction between a plane shock wave and an isotropic turbulence features the following characteristics:

- an oscillatory motion and a deformation of the shock wave, whose thickness is no longer constant but fluctuates with time (due to turbulence and compressibility effects).

- a global increase of the turbulent intensity downstream of the shock as illustrated in figure 2.8. A pronounced amplification of the turbulent kinetic energy, $k$, and the transversal components of vorticity (whereas the streamwise component remain nearly constant). Turbulence, if isotropic upstream, becomes axisymmetric downstream of the shock. Due to the shock motion the Reynolds shear stress, $\overline{u_i u_j}$, is more amplified than transversal shear stresses ($\overline{u_i u_j}$ with $i \neq j$) and clearly presents a peak at the shock location. It should be noted that, according to the kinetic energy transport equation, the redistribution term plays an major role in the downstream evolution of $k$.

- a modification of the turbulence characteristic length scales: small scales are more amplified than larger ones, which leads to a global reduction of the Taylor micro scale.
2.4. SHOCK/BOUNDARY LAYER INTERACTION

Figure 2.8: Distribution of normalized rms value of thermodynamic variables (left) and normal Reynolds shear stresses (right) through a shock wave (Lee et al., 1993)

- a mean flow globally in accord with the Rankine-Hugoniot (R-H) shock jump relations (although different than the laminar case).

Besides the above characteristics, Mahesh (Mahesh et al., 1997) found that the correlations between vorticity and entropic fluctuations play a major role in the interaction.

More recently, similar results were reproduced by Large Eddy Simulations (LES) (Ducros et al., 1999; Garnier et al., 1999) providing, however, a few corrections related to the numerical dissipation.

Also to be mentioned is the capability of second order closure models to predict the turbulence anisotropy through a weak shock wave as showed by Uhlmann (1997) whereas first order turbulent models (namely two-equations models) tend to overestimate the amplification of turbulent kinetic energy through the shock.

Back to an analytical point of view, previous studies assumed for simplicity that the shock front response to upstream disturbances is a small-amplitude sinusoid (Hardy and Atassi, 1997; Moore, 1963). As pointed out by Lele (1992), however, the classical R-H conditions, which relate the upstream and downstream states of a shock wave, are no longer exact for the mean flow. The mean R-H conditions are indeed modified by contributions to the mass, momentum, and energy fluxes from turbulent fluctuations. To properly understand the interaction of turbulence with shocks, Lele stated that the back reaction of the shock-turbulence interaction must as well be incorporated. Although already captured in numerical simulations, this approach appeared to be an entirely different perspective from that developed in previous analytic studies which only considered the interaction of linear modes with the shock. To address the closure problem, i.e., how certain turbulent fluxes change across the shock, Lele appealed to a form of Rapid Distortion Theory (RDT) across a thin, structured shock. Unfortunately, Lele’s initial assumption of a local 1D axis of symmetry aligned with the shock normal compromised his basic analysis. In a more recent study, Zank et al. (2002) developed a basic model which couples the turbulent energy density and correlation length of the flow with the mean flow field.

A summary of the different results found in the literature was addressed by Smits
and Dussauge (1996) who also discussed the appropriate use of both analytical linear approaches (i.e. the Linear Interaction Approach (LIA) and the RDT) to the different type of flow configurations.

2.4.4 Phenomenological description of transonic viscous interaction on airfoils

A phenomenological description of transonic viscous interaction on airfoils seems ideal to fully measure the importance of SBLI in unsteady flow.

For an airfoil, viscous effects are of special importance in two regions: in the vicinity of the shock foot and near the trailing edge, the situation at the trailing edge being obviously strongly dependent on the previous history of the boundary-layer, which includes its interaction with the shock.

The phenomenological description of the flow development past an airfoil at transonic speed was first given by Pearcey (1955, 1968). According to this well known and now classical work, interactions entailing flow separation (which are of special importance for practical purposes) are classified into Type A and Type B separation patterns:

- In **Type A flows**, a moderately strong shock induces a local thickening of the boundary-layer (see figure 2.9a). As the free stream (or incident) Mach number is increased, the shock becomes stronger and a limit is reached where Incipient Separation (IS) occurs at the shock foot. Thereafter, a separation bubble forms at the shock foot and any further increase in Mach number beyond that stage results in the growth of the separation bubble. The progressive growth of the bubble is thus a characteristic feature of this type of flow with the separation point fixed at the shock foot and the reattachment point moving progressively downstream towards the trailing edge as the overall strength of the shock increases. According to Pearcey, such a situation does not depend much on the boundary-layer thickness at the shock foot (provided it is fully turbulent). The reason is that Incipient Separation is weakly dependent on the Reynolds number as stated by Delery and Marvin (1986).

Furthermore, the growth of the separation bubble, in relation to an increase in the shock strength, is so rapid that the flow cannot be strongly influenced by scaling effects in the trailing edge region. In this situation, the rapid divergence of the trailing edge pressure and the correlative change in circulation occurs as a consequence of a rapid bubble growth triggered from the shock foot. This kind of interaction, designated as **Type A flow**, occurs with moderate APG downstream of the shock.

- In **Type B flows**, a very strong recompression takes place in the rear part of the profile. This situation corresponds to highly rear-loaded airfoils, as is the case with supercritical airfoils. The essential difference between this type and Type A is the inclusion of a second separation in the subsonic flow approaching the trailing-edge (figure 2.9b). This second separation is the classical subsonic, turbulent, rear-separation occurring in
2.4. SHOCK/BOUNDARY LAYER INTERACTION

an APG present on the rear part of an airfoil.

Additionally the separation of a boundary-layer primarily depends on the two following factors:

- the local APG imparted to the boundary-layer; or, more exactly, the pressure gradient scaled to the thickness $\delta$ of the boundary-layer,
- the velocity distribution across the boundary-layer. A destabilized boundary-layer having a high shape parameter is more likely to separate than one with a low $H_{12}$.

The shock effect serves both to thicken the boundary-layer (which increases the intensity of the APG) and to increase its shape parameter by decreasing the momentum in the near-wall region. Hence, it is clear that the shock interaction will "catalyze" the development of a rear separation that was already either incipient or actually present in the subsonic rear gradients before shock-waves appeared.

In such circumstances, one can expect several variants of the Type B flow as illustrated in figure 2.10 by Pearcey (1968):

- the shock interaction produces a bubble at the shock foot, thus a strong destabilisation of the boundary-layer which thereafter separates near the trailing edge,
- the perturbation produced in an interaction without separation is strong enough to promote rear separation.
• rear separation is already present in subsonic conditions, but the occurrence of a shock at higher Mach numbers worsens the situation by provoking a rapid extension of the rear separation.

Figure 2.10: Variants of type B flow in transonic interaction (Pearcey, 1968)

For all these circumstances, an increase in the upstream Mach number or in the angle of attack, results in the formation of a large separated zone extending from the shock foot. The formation of this zone can be the consequence of the merging of the bubble present at the shock foot and of the rear separated zone, which is usually a criteria for aerelastic phenomenon occurrence on airfoil profiles like buffeting.

2.4.5 Shock-boundary layer interaction without separation

A schematic representation of the SBLI without shock induced separation is shown in figure 2.11. The rise in pressure produced by the shock propagates upstream through the subsonic part of the boundary-layer. The subsequent deceleration entails a thickening of this subsonic layer. The corresponding bending of the sonic line generates compression waves which propagate in the supersonic part of the flow. Hence, in this region, the shock discontinuity is replaced by a gradual compression.

When the upstream Mach number is very close to unity, the "elliptic leakage" (presence of a subsonic region) beneath the shock produces a relatively slight thickening of the boundary-layer. Consequently, the compression waves induced in the adjacent supersonic part of the flow are very weak so that the shock is only slightly weakened as it propagates in the boundary-layer. It disappears only upon reaching the sonic line. On the other hand, when the incident shock becomes stronger, the thickening of the subsonic layer is more rapid, with subsequent higher local deflection angles. In these circumstances, the
induced compression waves are more intense. There results a greater (streamwise) weakening of the shock in the boundary-layer structure.

Laser measurements within the SBLI region over a flat plate were performed by Delery (1977) and streamwise Mach number distributions corresponding to increasing distance from the wall are presented in figure 2.12 as an illustration. A progressive and monotonic decrease in the Mach number for the region closest to the wall can be observed. Farther from the wall, the decrease in Mach number occurs through the shock discontinuity, the compression jump being immediately followed by an expansion. Further downstream, the Mach number increases anew before reaching a nearly constant level. The amplitude of the so-called "post shock expansion" increases with the upstream Mach number. This post shock expansion is a typical feature of the inviscid flow field associated with a transonic SBLI. It is also observed in airfoil flows. The phenomenon was described by Bohning and Zierep (1980) as an apparent wall curvature effect resulting from the rapid growth of the BL displacement thickness in the interaction region. As the streamlines are roughly parallel to the wall in the free stream, there results a slight contraction of the streamtubes and thus a slight acceleration of the flow downstream of the shock. Upstream of the shock, the influence of the boundary-layer thickening is transmitted along Mach waves and it is thus felt only near the wall.

Reynolds number influence

The SBLI has been investigated for different Reynolds number by Laurent (1977) in a "bump-on-the-wall" setup experiments. Pressure measurements were performed on the flat wall opposite to the bump in a wind tunnel allowing large variations of the Reynolds number by adjustment of the stagnation pressure. Isentropic Mach number distributions for different upstream Mach number values are presented in figure 2.13 for two different Reynolds number, \( Re_{\delta} \), based on the BL thickness.

Clearly the shock location is influenced by the Reynolds number value. At low
CHAPTER 2. BACKGROUND

(a) Streamwise Mach number distributions

(b) Interferogram of flow field

Figure 2.12: Mach number streamwise distributions in the outer flow (Delery, 1977)

Figure 2.13: Influence of Reynolds number on wall pressure distribution in transonic interaction (Laurent, 1977)

Reynolds number, the shock position spreads over a larger area for different upstream Mach number values than at higher Reynolds number.

Besides, it was observed that the lower the Reynolds number, the longer the pressure recovery downstream of the shock. Figure 2.14 presents different wall Mach number distributions relative to different Reynolds numbers, the initial Mach number $M_o$ and shape parameter $H_o$ being the same. An examination of these curves reveals that the spreading of the wall pressure distribution strongly depends on the local Reynolds number. The streamwise extent of this spreading significantly decreases when the Reynolds number increases. Basically the higher the Reynolds number the faster the pressure recovery.

Correlations on the interaction length

The interaction length, $L^*$, defined by Delery and Marvin (1986) as the domain of "supersonic interaction", corresponds to the distance between the origin of the interaction (i.e. the point where the pressure at the wall starts to rise) and the x-wise station where the local pressure is equal to the critical value corresponding to a "wall" Mach
2.4. SHOCK/BOUNDARY LAYER INTERACTION

Among the factors likely to influence the extend of the interaction zone, the most commonly investigated are the upstream Mach number, \( M_0 \), and Reynolds number, \( Re_{\delta^*} \), based on the displacement thickness. A thorough analysis of these effects was carried out by Delery (1980) on a "bump-on-the-wall" configuration.

The results obtained are presented in figure 2.15(a) in the form of a diagram giving the evolution of the normalized interaction length \( L^*/\delta_0^* \) as a function of the upstream Mach number \( M_0 \) (1.09 < \( M_0 \) < 1.30) for different \( Re_{\delta^*} \) values (from 0.15x10^4 to 1.08x10^5) and the same shape parameter value (\( H_0 \approx 1.2 \)) for the whole set of measurements.

The influence of the Reynolds number, very marked both on the physical extent \( L^* \) and on the thickness \( \delta_0^* \), disappears when these two variables are normalized one by the other. Thus, it can be concluded that, for a given value of the shape parameter, the displacement thickness of the incoming boundary-layer is a proper scale for the interaction length \( L^* \). Moreover, it appeared that the ratio \( L^*/\delta_0^* \) is not very sensitive to the effect of the upstream Mach number \( M_0 \).

Figure 2.14: Influence of Reynolds number on interaction zone extend (Laurent, 1977)

Figure 2.15: Influence of initial conditions on the supersonic interaction length (Delery, 1980)

Figure 2.16: Correlation of supersonic interaction length (Delery, 1980)
CHAPTER 2. BACKGROUND

When the normalization $L^*/\delta^*$ is applied to a more complete set of results obtained in various experimental facilities and corresponding to very different situations as regards to the state of the incoming boundary-layer (i.e. very different $H_0$) it appears that the data points are constant function of $H_0$ as shown in figure 2.15(b). This increase of $L^*/\delta^*$ when $H_0$ is higher can be explained by the fact that when $H_0$ is high, the boundary-layer has less momentum close to the wall and, consequently, its subsonic part is thicker. Hence, it seems natural that the distance for the propagation of upstream influence is longer.

Considering the above experimental evidence, a correlation of the results was presented in order to account the influence of $H_0$ on the normalization $L^*/\delta^*$ applied previously. The completely empirical law shown in figure 2.16 leads to a rather satisfactory grouping of the results and makes it possible to predict the streamwise extent of the supersonic part of a transonic interaction with reasonable accuracy within the domain of variation of the parameters involved (namely, $1.15 < H_0 < 1.50$ and $1.10 < M_0 < 1.30$).

2.4.6 Incipient shock induced separation in transonic flows

By definition, Incipient Separation (IS) is the situation in which the minimum of the wall shear stress $\tau_w$ in the shock interaction region is exactly equal to zero as illustrated in figure 2.17. A further increase of the shock strength beyond that point leads to a change in the sign of $\tau_w$, the region where $\tau_w$ is negative being called 'separated'.

![Figure 2.17: Definition of incipient separation from wall shear stress distributions](image)

As a matter of fact, the essential properties of a BL as well as the general flow structure are not greatly affected in an IS situation, the occurrence of separation being a rather progressive process. However, any slight change in outer conditions beyond the IS state may provoke a very rapid growth of an initially tiny separation bubble. Such increase of the size of the dissipative region considerably affects the whole flow field and generally leads to a catastrophic loss in terms of performance.

Due to turbulent fluctuations, the wall shear stress value at the IS location fluctuates around zero and an intermittent behaviour appears. Simpson et al. (1981) proposed a
2.4. SHOCK/BOUNDARY LAYER INTERACTION

Definition of separation based on the fraction of time, or intermittent parameter $\gamma_{P_u}$, during which the flow velocity at the wall follows the main flow direction. As a result, Simpson defined several positions downstream of the IS position in order to characterize the separation:

- $\gamma_{P_u}=1$: Attached flow
- $\gamma_{P_u}=0.99$: Incipient separation
- $\gamma_{P_u}=0.8$: Intermittent Transitory Separation (ITS)
- $\gamma_{P_u}=0.5$: Transitory Separation
- $\gamma_{P_u}=0$: Separated flow

For a probability under the Intermittent Transitory Separation (ITS) value, the velocity profiles within the BL do not follow the "log" law anymore and the turbulent models based on the similarity law hypothesis basically reach their limit of prediction.

Parametric influence on induced separation prediction

Various attempts have been made to determine criteria in order to predict induced separation. Among them, the results obtained during an extensive experimental investigation performed at ONERA (Delery, 1980) corroborated the fact that IS mainly depends on two parameters, namely:

- the Mach number $M_0$ on the upstream face of the shock,
- the shape factor parameter $H_0$ of the incoming BL (the often more commonly considered Reynolds number effect is for the greatest part included in the variation of $H_0$).

![Figure 2.18: Experimental shock induced incipient separation limit in transonic flow (Delery, 1980)](image-url)
A set of experiments in which the IS location was determined in various experimental facilities is presented in figure 2.18. Results have been plotted as a function of $H_0$ and $M_0$ and nearly collapse on a single curve defining a boundary between interactions with and without separation. A slow increase in the limit Mach number $M_0$ can be observed when $H_0$ is decreasing. This tendency could be anticipated since a lower value of $H_0$ means a higher momentum boundary-layer, hence a greater resistance of the BL to separation.

### 2.4.7 Transonic interaction with boundary-layer separation

When the upstream Mach number $M_0$ becomes noticeably greater than 1.3, a sizeable separation bubble forms at the shock foot. This bubble is extremely sensitive to external factors and its streamwise extent can increase dramatically as a consequence of a further rise in $M_0$ or the action of a downstream APG, such as the one existing on a highly rear-loaded airfoil or on a strongly curved surface.

As the size of the separated region increases, the outer flow develops a well-defined structure characterized by a *lambda shock pattern* which progressively emerges from the flow pattern observed in the unseparated configuration (see figure 2.19(a)). LDA measurements performed by Delery (1978) allowed to plot the iso-Mach number contours presented in figure 2.20. The analysis of the flow pattern revealed that the shock system is composed of:

- an oblique shock $C_1$ produced by the coalescence of compression waves resulting from the strong adverse pressure gradient at separation,
- a quasi-normal shock $C_2$ which meets $C_1$ at point I (sometimes called a bifurcation point),
- a third shock $C_3$, emanating from the triple point I.

The necessity of this lambda shock pattern comes from the fact that $C_1$ is a "weak" oblique shock (in the sense of the "weak solution" of the oblique shock theory) whose strength is only a function of the upstream Mach number $M_0$ and of the incoming BL properties. Thus, when this shock $C_1$ meets the "strong" quasi-normal shock $C_3$ present in the far outer field there exists behind $C_1$ and $C_3$ two states 1 and 3 with different pressures and velocity inclinations. At the meeting point I of the two shocks, these states are not compatible, as can be seen on the shock polar diagram shown in figure 2.21(a). In order to fulfill the conditions for two adjacent flows to be compatible (i.e. having same pressure and same velocity inclination), a third State 2, having the pressure and the velocity inclination of State 3, must be introduced. This state is reached through a shock-wave $C_2$, as shown on the shock polar diagram of figure 2.21(a).

Figure 2.21(b) presents more information on the nature and the strength of the shocks constituting the lambda pattern visible on the interferogram of figure 2.19(a). The various Mach numbers $M_0$, $M_1$, $M_2$ and $M_3$ were directly deduced from field measurements whereas the deflections $\Delta \phi$ have been computed from oblique shock theory. Due to the
2.4. SHOCK/BOUNDARY LAYER INTERACTION

(a) Experimental measurements
(b) Recomposed flow field structure

Figure 2.19: Streamwise Mach number distribution in a transonic interaction with a shock induced separation (Delery, 1978)

Figure 2.20: Lambda shock system above an extended separation in a transonic channel flow (Delery, 1978)

non uniformity of the supersonic incoming stream, conditions on the front face of shocks $C_1$ and $C_2$ are not rigorously constant.

It is observed that downstream of $C_1$ the Mach number of the outer flow remains everywhere supersonic with a general increasing trend from the boundary-layer edge to the vicinity of the triple point I. Shock $C_1$ has therefore the structure of a "weak" oblique shock-wave. A contrary, the Mach number distribution on the downstream face of $C_2$ features the opposite trend and decreases from the edge of the dissipative layer to point I. This result is not really surprising considering the normal R-H shock relations (the higher the upstream Mach number the stronger the jump and thus the lower the downstream Mach number) but leads to a situation where a high speed flow region is located close to
CHAPTER 2. BACKGROUND

(a) Flow situation at triple point I

Figure 2.21: Description of the lambda shock system pattern (Delery, 1978)

the BL edge whereas a (comparatively) "lower" flow region is located in the vicinity of the triple point I.

The local conditions at the triple point I are given in figure 2.21(a), the triple shock solution being represented on the shock polar diagram shown in the same figure. As a general property of this kind of solution, the flow Mach number after the bifurcated shocks $C_1$ and $C_2$ is always greater than the Mach number $M_3$ downstream of the unique shock $C_3$ (the total rise in entropy through successive shocks is always less than the rise through a unique shock leading to the same final static pressure). Consequently, the velocity in region 2 is greater than the velocity in region 3, since the stagnation enthalpy does not vary through a shock. This discontinuity in velocity leads to the existence of a slip line originating from the triple point I and which separates flow regions 2 and 3. The slip line is actually a shear-layer and corresponds to thin layer across which the flow properties vary in a continuous manner. The term "slip line" is commonly used in allusion to the outer flow inviscid behaviour. The term "vortex sheet" is sometimes employed. However in the present case, this slip line is barely visible on the interferogram of figure 2.19(a).

According to the LDA measurements, the deflection $\Delta \varphi$ across $C_2$ is always other than zero, which shows that $C_2$ is actually a "strong" solution of an oblique shock-wave (in comparison with $C_1$ which was stated as the "weak" solution of an oblique shock-wave). The strength of $C_2$ is seen to decrease while approaching the wall. According to Delery (1978) this weakening of $C_2$ is partly due to the varying upstream conditions (varying upstream Mach number in this case) and partly due to the effect of the compression waves generated by the growth of the BL displacement effect.

Similarly, the velocity measurements revealed that the shock $C_3$ is also a "strong" oblique shock which induces a quite important deflection at the triple point I. In the far field above I, the flow deflection decreases steadily as the shock becomes progressively
normal, and accordingly, its strength increases. Besides, the streamwise Mach number distributions obtained from both interferometry and LDA measurements are plotted in figure 2.19(b) and show that $C_3$ is immediately followed by a post-shock expansion whose amplitude, nearly non-existent at the triple point I, increases in the far field above I. This phenomenon is met within every transonic flow involving strong viscous-inviscid interaction and results from an apparent wall curvature effect resulting from the rapid growth of the BL displacement thickness in the interaction region as stated in the previous section on SBLI without separation.

In the present configuration, the rear shock $C_2$ is weakened to such an extent that it causes no disturbance to the wall static pressure distribution as seen in figure 2.19(a). Besides, the flow downstream of $C_2$ is everywhere subsonic. This situation is not a general property, but it is here due to the fact that the upstream Mach number is not very high ($M_0=1.37$ on the BL edge). For higher values of $M_0$ ($M_0 > 1.4$), a locally supersonic zone may exist downstream of $C_2$. The extent of this zone (introduced by Seddon (1976) as the supersonic tongue and illustrated in figure 2.22) depends on the particular conditions for the strong coupling process associated with the deviation towards the wall of the reattaching dissipative layer. Available experimental data (Kooi, 1978; East, 1976; Abiss et al., 1976; Seddon, 1976) show that the supersonic tongue appears from an upstream Mach number about 1.4. However, its streamwise extension and shape are extremely variable since, for a given value of $M_0$, the structure of the downstream part of the interaction strongly depends on the viscous-inviscid coupling process and on the conditions prescribed on the boundaries of the subsonic part of the flow field. A tedious flow investigation was performed by East (1976) in a wind tunnel channel at the same Reynolds number but different upstream Mach number values. The results illustrated the evolution of the supersonic tongue downstream of the shock $C_2$ as a function of the upstream Mach number $M_0$. However, Kooi (1978) obtained, for the same Mach number values, different results than Abiss and East. The observed differences were thought to originate either far-field effects or measurements uncertainties during a comparison performed by Delery and Marvin (1986).

![Figure 2.22: Illustration of flow field structure in a transonic interaction with extended separation (Seddon, 1976)](image)

Additionally, an interesting study concerning the angle influence between the shock wave and the main flow direction, and the effect on the shock induced separation bubble
CHAPTER 2. BACKGROUND

was performed by Green (1970) on airfoil profiles. The investigation of the flow structure revealed a strong three-dimensionality of the interaction when the shock was not perpendicular to the main flow direction. In the ideal case of an infinite geometry with a shock perpendicular to the origin of the BL, Green showed that the interaction could be considered as quasi 2D, i.e. no transversal flow, and featured a closed separation bubble as illustrated in figure 2.23a. However, in the case of an angle between the origin of the BL (namely the leading edge of an inclined airfoil) and the shock wave, Green diagnosed two possible behaviours:

- when the shock has a higher inclination than the leading edge, the transversal flow within the separation bubble increases and more fluid is directed towards the extremity of the airfoil (figure 2.23b),

- a contrary, if the shock has a lower inclination compared to the origin of the BL, the opposite configuration occurs and some flow leaks from the recirculation zone as illustrated in figure 2.23c.

![Figure 2.23: Illustration of angle influence on shock induced separation bubble shape (Green, 1970)](image)

Finally, an investigation of the SBLI over a 2D compression ramp flow performed by Andreopoulos and Muck (1987) revealed that the natural frequency of the shock-system unsteadiness scales on the bursting frequency of the incoming BL, which strongly suggests that the turbulence within the incoming BL plays a dominant role in triggering the shock wave natural unsteadiness. Besides, the study enlightened the fact that there actually is a wide band of timescales associated with the burst-sweep mechanism in a BL and that there therefore exists a wide band of frequency of oscillations the shock system. Measurements showed that the turbulence as inferred from wall pressure fluctuations might significantly amplified approaching the shock. Andréopoulos stated that a possible mechanism responsible for this effect might be the upstream propagation of the shock influence through the subsonic layer and/or the unsteady motion of the shock wave. It was also found that the time-averaged picture of the interaction region is three-dimensional, most probably due to the existence of Taylor-Görtler vortices which are formed in the concavely
2.4. SHOCK/BOUNDARY LAYER INTERACTION

curved surface.

2.4.8 Short review on unsteady SBLI in nozzle flow

Experimental studies on shock-boundary layer interaction in nozzles were performed by Edwards and Squire (1986). A rectangular flow channel with flat parallel walls downstream of a De Laval nozzle was used to investigate the normal shock interaction with a natural turbulent boundary layer at \( M_0 = 1.3 \) to 1.5. A downstream rotating, triangular shaped rod provided a time dependent back pressure, leading to an oscillating normal shock with a frequency up to 240 Hz. The reduced frequency based on equivalent chord was 0.2 to 2. Unsteady surface pressure measurements in the shock-boundary layer interaction region showed a negligible phase change between the shock motion and the surface pressure. It was concluded that quasi-steady assumptions can be used to calculate surface pressures at low frequency conditions. Boundary layer investigations were only made using spark shadowgraph photographs. From those it was not possible to decide if the leading foot of the lambda shock caused time-dependent separation. In this case close similarities to stall flutter would be obvious.

Another experimental work on a 2D nozzle with contoured upper and lower walls was performed by Ott (1991). Again, the shock oscillated due to a downstream rotating ellipse, causing time-dependent back pressure. However, the maximum shock oscillation frequency was limited to 180 Hz. Unsteady pressure measurements were performed on the side wall together with Schlieren visualization using a fast line scan camera. There again the pressure signals and shock oscillations were found to be in phase. Pressure excitations propagated upstream in front of the shock through the thick side wall BLs and the fast response surface pressure transducers sensed the arriving, upstream propagating shock before the shock had actually reached the transducer positions. The pressure amplitude seemed to depend strongly on the perturbation frequency and boundary layer thickness.

Ott et al. (1993) compared these experimental data with a numerical study. An unsteady Euler code in conservative form was used. The code was extended from a previous work, presented more in detail by Bölcs et al. (1988). Although the viscous effects were important in the experiments, the inviscid computational approach could approximately reproduce the shock position and amplitude. No significant phase shift between shock movement and pressure response was found at these low reduced frequencies, reproducing in that sense the experimental results. Small differences however appeared in the pressure signals due to viscous effects.

Another combined experimental and numerical work was performed by Cahen et al. (1995), based on experiments carried out by Delery and Marvin (1986) and Benay et al. (1986) on the same experimental setup. A swept three-dimensional bump was mounted on the lower wall of a transonic channel, with a maximum Mach number over the bump of 1.75. Although the back pressure was kept steady, some unsteady flow behaviour could be observed. Steady surface pressure measurements, flow visualizations and 3D Laser Doppler Anemometry (LDA) measurements were performed. Complex 3D flow structures were observed. The boundary layer separated due to the shock influence on the
downstream side of the bump. Separation lines were found to be dependent on local surface curvatures. They induced single and double vortex structures. The vortices caused strong viscous-inviscid interactions on the affected surfaces on all four side walls of the channel. The experimental data were compared with a 3D unsteady Reynolds Averaged Navier-Stokes code with two alternative turbulence models. Both models gave a faithful prediction of the essential features of the flow, in particular the shock pattern. The Jones-Launder $k-\epsilon$ model showed a clear superiority in the prediction of the behaviour of the interacting dissipative layers, in comparison to the algebraic turbulence model. The computations had some problems in predicting the complex separation line but especially the induced three-dimensional single and double vortex structures. The prediction of the turbulent quantities variation were far from being correct, i.e. a measured slight oscillation of the shock in the overall steady flow was not captured. The cause for this measured oscillation was not investigated.

The experimental data from Ott (1991); Ott et al. (1993) were compared with a new numerical approach developed by Gerolymos and Breus (1994). The numerical method was based on the integration of the unsteady two-dimensional Euler equations with a third-order upwind-biased scheme. Satisfactory agreement was obtained although the inviscid approach was used and the large experimental influence towards side-wall boundary layers. Strong non-linear effects in the pressure signals were found for moderate back-pressure fluctuation amplitudes.

Later on, Gerolymos et al. (1996) used a three-dimensional compressible Navier-Stokes solver with the Launder-Sharma near wall $k-\epsilon$ turbulence model to compare results with the same experimental data. The unsteady nozzle flow with an oscillating shock due to back pressure fluctuations was computed. Although the nozzle geometry was 2D, large 3D recirculating zones appeared in the corners of the nozzle due to the interaction between the BLs and the oscillating shock. The computed and measured unsteady pressure distributions agreed fairly well. Less good agreement was found for the velocity profiles and wall shear stresses.

Gerolymos and Vallet (1997) thereafter presented a computational method for the Favre-Reynolds-averaged 3D compressible Navier-Stokes equations using near-wall Reynolds-stress turbulence closure. This approach combined the Launder-Shima formulation for the Reynolds-stresses (Launder and Shima, 1989) or the Jones-Launder-Sharma modified dissipation equation (Launder and Sharma, 1974). The code was compared with the results from Cuhlen et al. (1995) and Delery and Marvin (1986) over the 3D swept transonic nozzle. The Mach number predictions showed an overall good agreement. The lambda-shock structure was well predicted whereas the $k-\epsilon$ turbulence model under-predicted the flow deceleration after the shock wave. Both models failed to reproduce the three-dimensional structure of the large recirculation zone at the near wall.

A pure computational approach on unsteady transonic nozzle flow over the experimental configuration of Ott was undertaken by Ferrand et al. (1997). Four parts of the complex flow structure were identified to interact: The outlet disturbance, the shock motion, the boundary layer separation and the SBLI area. The shock motion was found to be mainly harmonic. Static pressures at the shock-boundary layer interaction region and
further downstream showed strong non-linear behaviour with the appearance of higher modes. Deeper studies over weaker shock configurations showed a reduced effect of the dominating shock-boundary layer-stall interaction and an increase of static surface pressures. For very weak shocks acoustic propagation effects seemed to dominate and very large pressure amplifications were observed. It was concluded that weaker shock configuration produce larger instabilities of the unsteady flow.

All these experimental and numerical investigations have shown that although there have been significant progress in the field of compressible nozzle flow studies over the last few years, still many fundamental problems remain to be solved. These range from studies of transition on a "simple" stationary 2D flow to the complete unsteady 3D SBLI. Some of the numerical approaches discussed above seem to give good agreement with experimental data for isolated flow cases, but general agreement is poor.

As a conclusion a need for high frequency unsteady 3D experimental data having realistic three-dimensionally curved nozzle surfaces exists. Some approaches to this have been undertaken, but limited to either two-dimensional flow, steady or low frequency unsteady flow. Numerical approaches showed problems in predicting complex turbulent effects such as separation location and size, and viscous effects in general like shear stresses and 3D vortex structures. Only performed at low reduced frequencies, the modelling of quasi-steady behaviour between shock and surface pressure was not a problem.

2.5 Conclusion

This chapter was dedicated to the description of unsteady flow phenomena and aerelasticity related problems in axial turbomachines. The emphasis was put on a particular type of problems involving the coupling between a shock wave occurring in transonic flows and the turbulent boundary layer developing on the surface on an airfoil or a blade profile, namely the Shock/Turbulent Boundary Layer Interaction. Consequently, a detailed description of turbulent boundary layer flows, with and without pressure gradient influence, was presented, followed by a phenomenological description of the complex SBLI on airfoils and wind tunnel applications. A non exhaustive literature survey on transonic viscous interactions, with and without separations, was also presented, with the emphasis on the description of main flow characteristics within the interaction region and the assessment of influence parameters. Finally, a short literature review on the unsteady SBLI in transonic nozzle flow was presented as an introduction and state-of-the-art regarding the present thesis work.
Chapter 3

Objectives and method of attack

3.1 Objectives

The overall objective of the present work is, on one hand, to enhance the fundamental understanding of unsteady flow phenomena and their interaction with steady flow in axial turbomachines, and, on the other hand, to be able to better predict the unsteady pressure load on an airfoil or similar configuration.

According the discussion proposed in the previous chapter, the understanding of the SBLI in transonic unsteady flow is of particular importance in various kind of application. The present study should therefore focus on the interaction between upstream propagating pressure waves and a fully three-dimensional transonic flow structure, and more especially on the following points:

- the unsteady pressure distribution along the wall,
- the unsteady shock motion and eventual time lags with upstream propagating pressure perturbations,
- the unsteady shock-boundary layer interaction,
- the boundary layer separation motion,
- the propagation, amplification (or dissipation), and interaction of upstream propagating pressure waves in a two and three-dimensional compressible transonic flow.

3.2 Method of attack

In order to focus the analysis on the essential flow features described in the previous paragraph, the analysis is performed on simple models such as convergent-divergent nozzles. This approach has the advantage to avoid leading and trailing edges influences as well as inter-rows region interactions.
CHAPTER 3. OBJECTIVES AND METHOD OF ATTACK

With the aim at simulating realistic configurations, the test section of a wind tunnel was instrumented with various test objects, so called "bumps". In order to best simulate three-dimensional flow structures encountered in axial transonic turbomachines, one of the test object was numerically designed to reproduce varying local mean flow gradients through the width of the wind tunnel channel. In such configuration, at a particular operating point, a complex three-dimensional shock wave impacts on the turbulent boundary layer and spreads itself in the channel. Considering the natural characteristics of boundary layers, the interaction is non-linear and leads to low amplitude shock oscillations, which also corresponds to fluctuations of the aerodynamical resultant force on the surface. However, in order to first validate the experiments on a simple test case, a simpler 2D test object was also designed to slide through the width of the test section and therefore investigate the two dimensionality of the flow as well as side walls influence and corner effects.

Furthermore, similarly to potential interactions, periodic back pressure fluctuations were imposed to force the shock into larger amplitude oscillating motions with possible time lags between the shock motion and the unsteady pressure distribution on the surface.

In order to obtain a coherent validation of the results, the present research work was conducted both on an experimental and numerical levels. Numerical simulations were performed using the CFD tool, so called PROUST, developed at Ecole Centrale de Lyon, France, by Aubert (1993), and Smati (1997). This code solves the fully unsteady three-dimensional (3D) compressible Reynolds Averaged Navier-Stokes (RANS) equations, using a finite volume formulation and a linear two-equations turbulent model originally developed by Wilcox (1976). Experimentally, the investigation was performed in the VM100 wind tunnel at the Royal Institute of Technology, Sweden. In order to comply to the needs of the present study, the test section was entirely redesigned in a modular way and now features easy accesses for instrumentation, flow visualizations, and test object exchange. Several measuring techniques were used to investigate the steady and unsteady pressure distribution over the surface of test objects, the extend of the separated region, the heat transfer rate and flow structures within the boundary layer, as well as the periodic motion of the shock wave in the test section.
Chapter 4

Numerical Model

4.1 Introduction

The aim of this chapter is to give the reader an overview of the numerical model used to simulate the shock boundary layer interaction in two and three-dimensional nozzles. The 3D, unsteady, turbulent, compressible and non dimensional Reynolds averaged Navier-Stokes (RANS) equations are briefly introduced in the first part of this chapter, together with the two equations turbulence model developed by Wilcox and Traci (1988). The second part of this chapter presents a short summary of the numerical methods used to discretize the governing equations.

With the aim to keep this chapter as an -easy to read- introduction towards numerical methods, the contents of the present was kept as brief as possible but a complete description of the computational tool, including the main formulation as well as the different techniques applied in fluxes and numerical boundary discretizations, is presented in Appendix A.

4.2 Governing equations

The motion of a compressible viscous Newtonian fluid is governed by the Navier-Stokes equations, which originate the fundamental principle of mechanic and thermodynamic. Those equations are determined from the conservation laws for mass, momentum, and energy, and the thermodynamic relations for a perfect gas.

In order to account for natural turbulent characteristics of the flow, statistical methods are usually applied to the instantaneous governing equations. The equations are then linearized, which reveals double correlation terms and leads to an unclosed mathematical system. The evolution equations of those new terms would also reveal higher order correlation terms, and so on, \textit{ad infinitum}. The modelisation of all high order correlation terms is thus required in order to close the problem.

Using Boussinesq’s hypothesis (Boussinesq, 1897), the Reynolds stress tensor is expressed as a linear relation between the mean strain-rate tensor and a \textit{turbulent viscosity}. 

41
CHAPTER 4. NUMERICAL MODEL

The turbulence closure problem therefore consists in best evaluating this turbulent viscosity term in order to correctly account for the various time and length scales present in the different region of the flow. A basic dimensional analysis would show that the turbulent viscosity can be obtain by the determination of two characteristics scales. Among the different available modelizations, the two equations turbulent model proposed by Wilcox and Traci (1988); Wilcox (1976) basically consists of solving the transport equations for the turbulent kinetic energy and the specific dissipation, which represent respectively velocity (fluctuations) and time (decay of large structures) scales.

Finally, the RANS equations are made dimensionless in order to bring all variables to an order of magnitude close to unity and simplify numerical treatment of the equations. The dimensionless RANS equations used to simulate the motion of a compressible turbulent flow are expressed below:

**Mass conservation**

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot \left[ \rho \mathbf{V} \right] = 0 \]  

**Momentum conservation**

\[ \frac{\partial \rho \mathbf{V}}{\partial t} + \nabla \cdot \left[ \rho \mathbf{V} \otimes \mathbf{V} + p \mathbf{I} - \frac{1}{Re} \tau_{t+} \right] = 0 \]

**Energy conservation**

\[ \frac{\partial \rho E}{\partial t} + \nabla \cdot \left[ (\rho E + p) V - \frac{1}{Re} \left\{ \gamma \left( \mu \frac{P_v}{P_t} + \frac{\mu_t}{\sigma} \right) \mathbf{V} \cdot \nabla \mathbf{V} + \left( \mu + \frac{\mu_t}{\sigma} \right) \nabla \rho \right\} \right] = 0 \]

with: \( p = \rho (\gamma - 1) \left[ E - \frac{1}{2} \mathbf{V}^2 - k \right] \)

\( \tau_{t+} = (\lambda + \lambda_t) \nabla \mathbf{V} \mathbf{V} + (\mu + \mu_t) (\nabla \mathbf{V} + \nabla \mathbf{V}^T) - \frac{2}{3} Re \rho k \mathbf{I} \)

**Transport equation on** \( k \)

\[ \frac{\partial \rho k}{\partial t} + \nabla \cdot \left[ \rho \mathbf{V} \left( \mathbf{V} - \left( \rho + \frac{\mu_t}{\sigma} \right) \nabla \rho \right) \right] = \frac{1}{Re} P_k - C_k \rho \omega k \]

**Transport equation on** \( \omega \)

\[ \frac{\partial \rho \omega}{\partial t} + \nabla \cdot \left[ \rho \mathbf{V} \left( \mathbf{V} - \left( \rho + \frac{\mu_t}{\sigma} \right) \nabla \rho \right) \right] = C_{\omega_1} \frac{\omega}{k} P_k - C_{\omega_2} \rho \omega^2 \]

with: \( P_k = \tau_{t+} \mathbf{V} \mathbf{V} \)

\( C_k = 0.09 \quad C_{\omega_1} = \left( C_{\omega_2} - \frac{\mu^2}{\sigma} \right) \approx \frac{5}{4} \)

\( \sigma_k = 2.0 \quad \sigma_\omega = 2.0 \quad C_{\omega_2} = \frac{1}{4} \)
4.3 Numerical methods

The unsteady motion of a three-dimensional compressible turbulent flow has been modelled in the previous section by a partial differential equations system defined in a bounded continuous domain. In many cases, however, the resolution of such system does not admit any analytical solution and has to be numerically resolved. To do so, both the domain and the mathematical system have to be discretised. That is to say, the partial differential equation system is being replaced by an algebraic equation system for variables calculated only at a finite number of discrete positions in time and space. The discrete solution is thus an approximation of the initial problem, as accurate as the discretization is fine and close to a continuity.

There exists many different techniques to discretize the differential operators. The choice regarding a numerical method mostly depends on the physic of the phenomenon, but also on the complexity of the geometry, and on the development and exploitation costs. The spatial discretization of the present computational model is based on a finite volume formulation with vertex variable storage, developed on a structured mesh. The convective terms are evaluated using a upwind flux vector splitting scheme coupled to a MUSCL approach, whereas the viscous and turbulent terms are computed using a second order finite difference scheme. The time discretization is performed using an explicit five-steps Runge-Kutta time marching algorithm.

Finite volume formulation

The basic idea of a finite volume formulation is to fragment the domain into elementary contiguous volumes, on which are integrated the differential conservative equations. The cells have to be small enough so that the flow properties slowly vary within each integrating volumes.

This approach presents several advantages. First of all, it is conservative, which makes the discretization of the differential equations more natural. Moreover, discontinuities like shock waves and contact surfaces are naturally treated as the are weak solutions of the problem. Indeed, although the flow field might be discontinuous, the fluxes through the surfaces of the integrating cells remain continuous. At last, there is no real strain concerning the size and shape of the cells as far as the mesh is fine enough to consider the flow properties constant inside each cell. In practice however, the accuracy of the solution is degraded by grid distortion and an orthogonal or near-orthogonal grid should be used.

Looking at the partial differential equations governing the motion of a fluid, one can note the following general form:

\[
\frac{\partial A}{\partial t} + \nabla \cdot \bar{B} = S
\]

(4.6)

where: \( A \) and \( S \) are scalar variables (respectively vector) \( \bar{B} \) is a vector variable (respectively tensor)
In a finite volume formulation with structured mesh, the above equation is integrated in a curvilinear system, on each single contiguous elementary volume. Using Green’s theorem, it comes:

\[
\frac{d}{dt} \int_{V(t)} A \, dV + \oint_{S(t)} \vec{B} \cdot \vec{n} \, dS = \int_{V(t)} S \, dV
\]  

where:

- \( S(t) \) is the surface of the integrating cell.
- \( V(t) \) is the volume of the integrating cell.
- \( \vec{n} \) is a unit vector normal to \( S(t) \), pointing outside \( V(t) \).

The evaluation of those integrals is performed by introducing a transformation from physical \((x, y, z, t)\) space to a generic curvilinear coordinate \((\xi^1, \xi^2, \xi^3, t)\) space, which is well adapted for the description of a structured mesh (figure 4.1).

The governing equations are then expressed in terms of the generic coordinates as independent variables and the discretization is undertaken in the generic coordinates space. As a result, the governing equations can then be rewritten, in the general form, as follow:

\[
\frac{\partial}{\partial t} \left( \sqrt{g} A \right) \mid_{\xi^1,\xi^2,\xi^3} + \sum_{i=1}^{3} \left\{ \pm F^i \bigg|_{\xi^i \pm \frac{1}{2}} \right\} = \sqrt{g} S \mid_{\xi^1,\xi^2,\xi^3}
\]  

where \( F^i \) is the flux defined by: \( F^i = \sqrt{g} \vec{B} \cdot \vec{a}^i \).

The expression of the fluxes are developed for the conservative equations in Appendix A.

Time discretization

It comes from the finite volume formulation that the numerical evolution of the aerodynamic flow field is described by the following semi-discrete formulation:
4.3. NUMERICAL METHODS

\[
\frac{\partial}{\partial t} (\sqrt{g}q) \bigg|_{\xi_1, \xi_2, \xi_3} = -R|_{\xi_1, \xi_2, \xi_3, t} \tag{4.9}
\]

where, \( R \), so called residual, is defined by:

\[
R = \sum_{i=1}^{3} \left[ F^i(\xi^i + \frac{1}{2}) - F^i(\xi^i - \frac{1}{2}) \right] - \sqrt{g}S \tag{4.10}
\]

The time integration of the above equation then allows to calculate the solution \( q^{(n+1)} \) at the next time step \( (n+1) \), from the solution \( q \) at one or several previous time steps \( q^{(n)}, q^{(n-1)}, q^{(n-2)}, ... \)

Considering the objective of the present work, i.e. the analysis of interactions in unsteady transonic flows, the use of an explicit time marching method was adopted in regards to the numerical perturbations inherently introduced in implicit schemes. As a result, the Runge-Kutta scheme was used since it offers the advantage to be a second order precision technique for a simple and straightforward application. The scheme as well as time stepping calculation and smoothing techniques are further developed in Appendix A.

Spatial discretization

From the finite volume formulation, we now have to evaluate the fluxes through the boundaries of the integration cell. The dimensionless RANS equations describing the unsteady motion of a compressible turbulent flow, constitutes an hybrid partial differential equation system. That is to say, the diffusive and thermal conductivity terms are described by elliptic equations whereas the convective terms are hyperbolic by nature. Those two types of equation systems corresponds to different physical phenomena and thus require different numerical discretization. As a result, the flux evaluation will be dissociated into two parts, the \textit{convective fluxes} and the \textit{diffusive fluxes}.

The hyperbolic character of certain equations suggests a wave propagation system, with favored velocity and directions of propagation in the flow, as well as possible discontinuities. It appears that upwind schemes are especially suited to represent such physical phenomena. They are based on the separation of both the upstream and downstream influences of physical perturbations which propagate following the characteristic waves of a Riemann problem. This kind of schemes allows a good capture of the discontinuities and naturally possess an artificial dissipation, which makes them unconditionally stable. Moreover they nicely behave in presence of undulatory phenomena and thus allow a good representation of unsteady flows.

On the other hand, elliptic equation systems, basically describing diffusive phenomena, does not feature preponderant directions in the flow field, and will be discretised using a centered scheme.

Once again, all details of the numerical techniques concerning spatial discretization is presented in Appendix A. Especially, the expression of convective and diffusive fluxes, for both the RANS and the turbulent equations, as well as the spatial accuracy limiters.
or the use of numerical filters are fully described in the Appendix.

Boundary discretization

The description of the numerical method has until now only considered nodes inside the computational domain. In order to account for external influences such as guiding adiabatic walls, steady subsonic inflow or unsteady outflow properties, a special treatment has to be realized on the nodes contiguous to a boundary. The method used is based on the creation of external supplementary nodes, and the expression of compatibility relations. The nodes located on the boundaries are then treated exactly like the nodes inside the computational domain. This treatment ensures the coherence between the spatial-temporal schemes used both inside and on the boundaries of the computational domain. The scheme accuracy order is then preserved, avoiding at the same time any eventual parasite time phase lags.

Although many types of boundary conditions are treated in the CFD tool, only five different types are considered in the present work:

- "Free" boundaries like inlet/outlet boundaries,
- adherent and sliding solid walls,
- symmetry planes,
- and inter-domain boundaries.

In the general case, boundary conditions can be separated into two kinds: geometrical and physical conditions. In the first category, a geometrical relation is established between the boundary nodes and the internal nodes, whereas in the second category, a physical condition is introduced in the system as a new equation to resolve.

Those two types of conditions are applied in an unsteady local mono-dimensional way to a boundary. Therefore, in the case of an edge or a corner, there is a excess of boundary conditions compared to the set of equations and mixing those conditions sometimes ends up in numerical interferences. Whereas geometrical conditions can easily be combined, physical conditions are more delicate to handle because of the physical balance that has to be respected. Which relation should be preponderant in case, for instance, of a corner between a outlet boundary, an adiabatic adherent wall, and a symmetrical plane boundary? The choice concerning the boundary condition to apply thus depends on the physical one want to introduce into the computation.

The numerical treatment of each type of boundary condition, physical or geometrical, as well as the general method based on the creation on supplementary planes and compatibility relations are further developed in Appendix A.
4.4 Conclusion

This chapter was dedicated to the overall presentation of the numerical methods used to resolve the set of equations modelling an unsteady compressible turbulent flow. A more detailed description of the techniques being presented in Appendix A.

First of all, the instantaneous Navier-Stokes equations, statistically averaged to account for turbulence effects, and made dimensionless to facilitate the numerical treatment were presented.

Secondly, the finite volume discretization method in curvilinear structured mesh was introduced. The time and space were thereafter briefly presented with reference to a more complete description presented in Appendix A. Finally, the numerical treatment of different geometrical and physical boundary conditions, based on compatibility relations and the extension of the computational domain, was exposed.

The numerical model being globally described, a more detailed description of the different simulations, test objects geometries, grid meshes and boundary conditions will be presented in chapter seven. The next chapter will attempt to present the experimental model, that is to say the facility as well as the different measuring techniques and respective accuracy.
Chapter 5
Experimental Model

5.1 Introduction
The present experimental study aims at further understand basic flow interactions that occur in turbomachines. In order to focus the analysis on essential features, the present investigation is being carried out on simple geometries such as convergent divergent nozzles. Special influences of leading and trailing edges, or interblades row region interactions are therefore not considered.

The test section of an already existing supersonic wind tunnel was entirely rebuilt for this purpose, and now allows the investigation of subsonic and transonic unsteady flows. The new test section is now modular and can be equipped with different test objects like 2D or 3D bumps in order to create a contraction of the channel. A shock wave is then expected to occur and interact with the incoming boundary layer. A perturbation generator was also designed to create back pressure fluctuations, and put the shock wave into an oscillating motion.

In the first section is described the overall facility, including the air supply wind tunnel, the newly designed test section and the test object design objectives. The second section presents the instrumentation of all experimental devices used. Finally, the different measuring techniques are presented together with their respective data evaluation method and accuracy.

5.2 Experimental apparatus
In this section the test facility apparatus is described, including the air supply wind tunnel, the redesigned test section, the test objects, and the perturbation generator.

5.2.1 Air supply wind tunnel facility
The air supply apparatus consists of an opened wind tunnel facility in which the air is sucked by a screw compressor, driven by a 1MW electrical motor, from the atmosphere
into a dryer, and compressed at a fixed operating point. The maximum mass flow available is about 4.7 Kg/s at 4 bars and 30°C (measuring conditions). The air exhaust temperature from the compressor is approximately 180°C and can be adjusted down to 30°C by an air cooling system immediately followed by the condenser. At this location, the standard orifice plate allows the measurement of the effective mass flow to be performed by simple pressure taping. Further downstream, a set of valves redirects the flow either in a test turbine or in an exchangeable test section as sketched in Figure 5.2.1.

Figure 5.1: Overall air supply facility at HPT

Another set of three valves allows to control both the mass flow and the pressure level in the test section as illustrated in Figure 5.2.

Figure 5.2: Operating scheme of the overall facility

By opening the inlet valve or closing the bypass valve, the mass flow can be increased inside the test section. Closing the outlet valve has the effect of increasing the pressure level and decreasing the mass flow. A sensitive set up of the different valves is necessary to adjust the mass flow and pressure level in the test section, i.e the inlet Mach number and
5.2. EXPERIMENTAL APPARATUS

the Reynolds number in the respective range of $M_i = 0.1 - 0.8$ and $Re = \frac{\rho_{air} U_{\infty} d}{\nu_{air}} = 1.87 \times 10^4 - 1.57 \times 10^6$ with $d = 0.26 \text{m}$, $\rho_{air} = 0.54 - 4.48 \text{ kg/m}^3$ and $\nu_{air} = 1.5 \times 10^{-5} \text{ m}^2/\text{s}$. In order to compensate for the pressure losses in the different pipes between the compressor and the exhaust pipe, a fan sucks the air downstream of the test section out to the atmosphere. As a result, the pressure level can be set under atmosphere pressure when the mass flow is low.

5.2.2 Modular test section

One of the first requirements to carry on the present study of shock boundary layer interaction in unsteady transonic flows was to build a facility that would, first, allow such an investigation, and secondly, still be adaptable for future experimental research on the subject. As a result, an already existing facility was taken apart and a brand new test section was built as modular as possible to suit the present investigation, and allow future developments. The complete redesign procedure is further described in Appendix B.

The exchangeable test section of the overall facility is presented in Figure 5.3. It features a $250 \text{mm}^2$ square settling chamber equipped with screens and honey combs, followed by two convergent parts which accelerate the flow into the $120 \text{mm}$ high and $100 \text{mm}$ wide channel. The facility features a modular test section which can be equipped with different test objects such as bumps or flat plate. Besides, both the upper and lower walls, as well as the side walls feature large openings for optical access or instrumentation.

![Detailed drawing of the VM100 test section](image.png)

Figure 5.3: Detailed drawing of the VM100 test section
CHAPTER 5. EXPERIMENTAL MODEL

Visualizations, laser measurements, or many other types of measurements are thus possible and easy to perform. Naturally, a number of pressure taps and probe inserts have been located in the test section in order to monitor the flow conditions.

5.2.3 Convergent-divergent nozzles

This subsection is dedicated to the description of the test objects. First, the design objectives will be introduced and discussed, and a full description of the geometry will finally be presented.

5.2.3.1 Three-dimensional nozzle

The main objective of the present study is a better understanding of basic unsteady interactions in turbomachinery flows. More especially, the pressure amplification mechanisms on blades surface and the phase lags between the shock motion and the pressure distribution are of particular interest concerning aeroelasticity problems. In order to carry on such study, a three-dimensional test object was theoretically designed and numerically validated by Bron (1997). A brief description of the background and design objectives is given below, as well as the complete geometrical description of the 3D bump:

Introduction

Originally, the design of the 3D bump is based on remarks and observations issued from different publications on unsteady flows and acoustic flow interactions. The main contributions however were the State Doctoral thesis by Ferrand (1986) on choked flutter in turbomachines, and the more recent paper by Ferrand et al. (1996) on unsteady flow amplifications by downstream pressure perturbations.

The first reference concerns a simulation of the experience on choking flutter conducted by Tanida and Saito (1977). The authors numerically reproduced the unsteady flow field characteristics around an oscillating blade profile in a wind tunnel. The strong point of the analysis was the phenomenological and parametrical study assessed to establish influences from different flow phenomena like the strength of the shock wave, the mean flow gradient, or the reduced frequency.

In the second publication (Tanida and Saito, 1977), the authors performed an analytical and numerical study on unsteady pressure fluctuations propagating in a convergent-divergent nozzle. Basically, the analytical solution of the unsteady mono-dimensional Euler equations, originally expressed by Carrière (1976), was first rewritten as a relation between static pressure and velocity fluctuations. Then, from this general formulation, the authors could express the behavior of upstream and downstream propagating waves inside the nozzle for different flow configurations. They could therefore isolate the influences of local flow speed, mean flow gradients, shock strength, or reduce frequency. Especially, it
was stated that, as soon as the flow becomes sonic at the throat\(^1\), the system can be expressed as an algebraic equation between the variables. From this point, the role played by the mean flow gradients was found essential. Typically, it was postulated that the value of the mean pressure gradient at the throat would condition the behavior of the static pressure fluctuations compared to the velocity fluctuations, establishing a sort of filter which would either block, magnify or introduce a phase lag between the two kind of fluctuations.

From the analysis made in the above cited references, the following points could be highlighted:

- Outlet pressure perturbations are magnified when propagating into a near sonic flow region. This effect is called the *acoustic blockage* (Atassi et al., 1994b).
- The values of the mean flow gradients both upstream and downstream of the sonic line location seem to have an influence on the amplification factor of the forward and backward propagating perturbations.
- The reduce frequency has an influence on the phase lag between static pressure and velocity fluctuations when reaching the shock location. High frequencies tend to increase this phase lag.

### Design objectives

The above key points were extrapolated into the following new points for a two and three-dimensional flow fields:

- The area, where the sonic line and the shock collapse together, is characterized by a zero mean flow gradient and should thus be unstable regarding the amplification factor of the upstream propagating pressure perturbation and the shock motion.
- The high mean flow gradient region, located where the local curvatures are the strongest, should on the contrary, be attenuating and introduce a filter between static pressure and velocity fluctuations.
- All locations in between the two situations described above should then present a continuous evolution of the phase shift.

According to these new statements, the three-dimensional nozzle was globally designed to check the above key-points. The design objectives were to create a flow structure which would be both exciting and attenuating (at another location) regarding the shock motion and the amplification of back pressure fluctuations. As a result, it was decided to modify the local curvatures in such way to create a near sonic flow region with low mean flow

\(^1\)It should be noted that the analytical study performed in the above cited references was performed using a mono-dimensional formulation, and included several assumptions regarding the flow gradient and the evolution law of the section. The design objectives listed below tries to incorporate the knowledge gained over many test cases and the reader is advised to refer to the references cited in the state-of-the-art chapter on acoustic blockage for further and detailed information about the background of this research.
gradients slightly downstream of a shock wave.

A sketch of the desired flow configuration over the 3D bump is presented in Figure 5.4. Strong curvatures on one side of the bump create a local flow acceleration whereas smooth curvatures on the other side keep the mean flow gradient low. As a result, a sonic pocket appears and develops itself on one side of the bump. The shock wave should however not extend through the width of the channel so that the region where both the sonic line and the shock wave meet should be located in the low mean flow gradients. Besides, the evolution law of the throat line was determined in order to slightly "bend" the shock and locate the unstable region further downstream.

Finally, the design objectives were achieved by varying the length and thickness of the convergent-divergent parts through the width of the channel. The important parameters were the length and thickness of the bump both in the converging and diverging part of the nozzle, as well as the throat line evolution law.

**Bump description**

Practically, the three-dimensional shape of the bump was determined in the following way: First, a sixth-order polynomial function was determined in the middle plane of the channel with respect to the continuity of the first and second derivatives. Then a transversal evolution law was established for the position of the throat and length of the bump. The leading edge being perpendicular to the main flow direction. The polynomial function’s design parameters were $X_{LE}^*$ and $X_{TE}^*$ the start and end location of the bump, $d_t^*$ the relative position of the throat, and $\epsilon^*$ the thickness of the bump. Furthermore, the parameters describing the bump in the transversal direction were $\alpha$, $\gamma$, and $\beta$, the three angles which determine respectively the slope of the throat line, the "trailing edge" line of the bump, and the transversal distribution of the thickness. The exact coordinates of the three-dimensional shape are presented in Table 5.1 and illustrated in Figure 5.5.
5.2. EXPERIMENTAL APPARATUS

Mid-channel polynomial function

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Transversal evolution parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>$X_{le}^* = 0.00m$</td>
<td>$Y_D \tan \alpha = 0.6 \Rightarrow \alpha \simeq 15.1^\circ$</td>
</tr>
<tr>
<td>$X_{te}^* = 0.15m$</td>
<td>$\tan \beta = 0.6 \Rightarrow \beta \simeq 3.09^\circ$</td>
</tr>
<tr>
<td>$Y_D = 0.1m$</td>
<td>$\gamma = 40^\circ$</td>
</tr>
<tr>
<td>$Z_D = 0.12m$</td>
<td>$\epsilon^* = 9% = 0.09$</td>
</tr>
<tr>
<td>$d_t^* = 30% = 0.3$</td>
<td>$\epsilon_y = \epsilon^* + \tan(\gamma) \left( \frac{y}{y_D} - \frac{1}{2} \right)$</td>
</tr>
</tbody>
</table>

Table 5.1: Design parameters for the 3D bump

\[
x_t = X_{le}^* + \left( X_{te}^* - X_{le}^* \right) d_t^* + \tan(\alpha) \frac{y_D}{D} \cos \left( \frac{\pi}{y_D} x \right)
\]

\[
\epsilon_y = \epsilon^* + \tan(\beta) \left( \frac{y}{y_D} - \frac{1}{2} \right)
\]

\[
X_{te}^* = X_{te}^* - \tan(\gamma) \left( \frac{z}{y_D} - \frac{1}{2} \right)
\]

\[
Z_{bump} = \epsilon_y \left( 6X^5 - 15X^4 + 10X^3 \right) Z_D
\]

Figure 5.5: Description of the three dimensional bump shape

Finally, Table 5.2 presents the location and thickness of the throat as well as the total length of the bump for three different transversal positions: the "strong curvature" plane, the middle plane and the "smooth curvature" plane.
CHAPTER 5. EXPERIMENTAL MODEL

Smooth Curvatures
\[ y = 0.0m \]

Middle Plane
\[ y = 0.05m \]

Strong Curvatures
\[ y = 0.1m \]

<table>
<thead>
<tr>
<th>Location in the test section</th>
</tr>
</thead>
<tbody>
<tr>
<td>Smooth Curvatures</td>
</tr>
<tr>
<td>( x_{te} = 0.0m )</td>
</tr>
<tr>
<td>( x_{throat} = 0.0585m )</td>
</tr>
<tr>
<td>( \epsilon_{throat} = 7.56mm )</td>
</tr>
<tr>
<td>( x_{te} = 0.192m )</td>
</tr>
<tr>
<td>( l_{bump} = 192mm )</td>
</tr>
</tbody>
</table>

Table 5.2: 3D bump profile description at different transversal locations in the channel

Location in the test section

Once the shape of the 3D bump was established, the test object was manufactured by a modeler with an accuracy within \( \pm 0.1mm \). Basically, the three-dimensional shape was milled out of block of resin injected with aluminium particles. A 25mm thick and a 100mm large rectangular base (see Figure 5.6) was left over in order to insert the 3D bump into the 290mm long hole located on the support part as seen in Figure B.3. The start of the bump was located exactly 70mm downstream of the beginning of the insert hole, as illustrated in Figure 5.7.

Figure 5.6: Three dimensional bump on its rectangular base

5.2.3.2 Two dimensional nozzle

As already mentioned, the pressure amplification mechanisms on blades surface as well as phase lags between an oscillating shock and the unsteady pressure distribution are of particular interest in the present study. However, the three-dimensional configuration as described in the previous paragraph will surely feature complex flow structures, making the analysis rather difficult alone. As a result, in order to simplify the analysis, and dissociate or even compare the three-dimensional flow effects from the two-dimensional ones, a simpler configuration was created to start the investigation with.
Two-dimensional bump profile:

The simplest geometry to investigate is of course a two-dimensional nozzle, but this new test object should have some similarities with the 3D bump created previously. For instance the mean flow gradients, that is to say the main curvature characteristics should be analogous. As a result, it was decided to preserve the same cross section area evolution, that is to say to have the exact same area as a function of $\vec{x}$, the main flow direction. The area function $A = f(x)$ of the 3D nozzle was therefore determined from the analytical expression given in the set of equations 5.1, and a two-dimensional profile was determined by simply dividing the area function by the width of the channel. A coarse set of coordinates is given in Table 5.3 and illustrated in Figure 5.8.

Finally, Table 5.4 presents the location and thickness of the throat as well as the total length of the bump.

Location in the test section

The two-dimensional test object was manufactured in aluminium with the same accuracy as the 3D bump. The beginning of the bump was located exactly as for the 3D bump, that is to say 70mm downstream of the insert hole’s edge, as illustrated in Figure 5.9. Besides, in addition to a first 100mm large 2D bump, a second but 350mm large test object was manufactured for a sliding purpose which will be further described in the instrumentation section. Consequently, the thickness of the rectangular base was set to 12mm so that the test object could slide through the side wall’s openings.
CHAPTER 5. EXPERIMENTAL MODEL

<table>
<thead>
<tr>
<th>X</th>
<th>z_{bump}</th>
<th>X</th>
<th>z_{bump}</th>
<th>X</th>
<th>z_{bump}</th>
<th>X</th>
<th>z_{bump}</th>
</tr>
</thead>
<tbody>
<tr>
<td>-70.00</td>
<td>0.00</td>
<td>26.29</td>
<td>7.50</td>
<td>61.11</td>
<td>9.82</td>
<td>95.94</td>
<td>4.92</td>
</tr>
<tr>
<td>-56.51</td>
<td>0.00</td>
<td>29.19</td>
<td>8.39</td>
<td>64.01</td>
<td>9.56</td>
<td>98.84</td>
<td>4.47</td>
</tr>
<tr>
<td>-43.11</td>
<td>0.00</td>
<td>32.09</td>
<td>9.07</td>
<td>66.92</td>
<td>9.25</td>
<td>101.96</td>
<td>4.01</td>
</tr>
<tr>
<td>-29.72</td>
<td>0.00</td>
<td>34.99</td>
<td>9.59</td>
<td>69.82</td>
<td>8.91</td>
<td>107.25</td>
<td>3.29</td>
</tr>
<tr>
<td>-16.39</td>
<td>0.00</td>
<td>37.90</td>
<td>9.97</td>
<td>72.72</td>
<td>8.54</td>
<td>117.22</td>
<td>2.16</td>
</tr>
<tr>
<td>-5.13</td>
<td>0.00</td>
<td>40.80</td>
<td>10.24</td>
<td>75.62</td>
<td>8.13</td>
<td>131.71</td>
<td>1.02</td>
</tr>
<tr>
<td>3.45</td>
<td>0.06</td>
<td>43.70</td>
<td>10.40</td>
<td>78.52</td>
<td>7.70</td>
<td>146.34</td>
<td>0.37</td>
</tr>
<tr>
<td>9.78</td>
<td>1.01</td>
<td>46.60</td>
<td>10.47</td>
<td>81.43</td>
<td>7.25</td>
<td>160.98</td>
<td>0.09</td>
</tr>
<tr>
<td>14.33</td>
<td>2.53</td>
<td>49.50</td>
<td>10.46</td>
<td>84.33</td>
<td>6.79</td>
<td>175.61</td>
<td>0.01</td>
</tr>
<tr>
<td>17.58</td>
<td>3.89</td>
<td>52.41</td>
<td>10.38</td>
<td>87.23</td>
<td>6.32</td>
<td>190.24</td>
<td>0.00</td>
</tr>
<tr>
<td>20.48</td>
<td>5.18</td>
<td>55.31</td>
<td>10.24</td>
<td>90.13</td>
<td>5.85</td>
<td>204.88</td>
<td>0.00</td>
</tr>
<tr>
<td>23.39</td>
<td>6.41</td>
<td>58.21</td>
<td>10.05</td>
<td>93.04</td>
<td>5.38</td>
<td>220.00</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Table 5.3: Two-dimensional test object profile coordinates

<table>
<thead>
<tr>
<th>2D bump profile</th>
</tr>
</thead>
<tbody>
<tr>
<td>y ∈ [0;0.1]m</td>
</tr>
<tr>
<td>x_{le} = 0.0m</td>
</tr>
<tr>
<td>x_{throat} = 0.0476m</td>
</tr>
<tr>
<td>\epsilon_{throat} = 10.48mm</td>
</tr>
<tr>
<td>x_{te} = 0.184m</td>
</tr>
<tr>
<td>l_{bump} = 184mm</td>
</tr>
</tbody>
</table>

Table 5.4: Profile description of the 2D bump geometry

5.2.4 Perturbation generator

With the aim at studying unsteady transonic flow interactions, the shock wave appearing in the divergent part of the nozzle was put into an oscillating motion by simply rotating an elliptical cam downstream of the test section. As the cam rotates, the amount of loss created by the wake periodically varies, which induces a fluctuating back pressure. In a subsonic flow, the pressure perturbations propagate both upstream towards the test section with the relative velocity, \(u - c\), and downstream towards the diffusor at \(u + c\), as illustrated in Figure 5.10. The amplitude and frequency of those pressure waves can be adjusted depending on the ratio between the semi-major and semi-minor axis of the elliptical rods and the rotating speed of the cam. As these acoustic waves have a relative velocity of \(u - c\) and \(u + c\), their wave length also depends on the operating conditions, i.e on the flow velocity \(\lambda = \frac{u + c}{u - c}\). Finally, as the back-pressure fluctuations reach the test section, the shock wave starts to oscillate at twice the rotating speed of the rod.

Basically, the design objectives were to generate back pressure perturbations with controlled amplitude and frequency up to 500Hz. As a result, different cams were manufactured and are presented in Table 5.5. The exact geometry of the different elliptical rods was thought to be of secondary importance and will not be discussed here. It is indeed much more interesting to directly look at the frequency content and phase angle of the pressure fluctuations propagating upstream in the test section.
5.2. EXPERIMENTAL APPARATUS

A high speed miniature DC motor was used in order to reach the high frequency requirement. The motor is controlled by a servo amplifier and features an optical encoder which is directly coupled to the power supply via a servo controlled feedback loop in order to stabilize and keep the rotating speed constant. Furthermore, as the device should be as stationary as possible, both during a single revolution and during a long term period, a metallic disk was added on the shaft in order to increase the inertia of the elliptical rod and counter act the aerodynamic forces that will induce a torque to the rotating rod. Finally, a set of plugs were manufacture with special bearings able to withstand such high rotating speeds.
5.3 Instrumentation

5.3.1 Atmospheric conditions

Barometer

Atmospheric pressure and temperature were measured using the 7885-1A Smart Digital Pressure (SDP) module. This high accuracy barometer uses as its main sensing element the 7881 vibrating cylinder pressure sensor with range 60-115kPa. On-board frequency and temperature measurement circuitry, based on a Motorola micro-controller, provide a fully controlled pressure output. The sensor and its electronic printed circuit boards are enclosed in a strong customized metal enclosure with one pressure port and two electrical ports for power supply and data transfer. Pressure data output is delivered via a RS232 serial port interface directly to the host computer.

It should finally be noted that this digital barometer is sent regularly (every second year) to the manufacturer to be calibrated both in pressure and temperature in order to account for temperature influence on the pressure sensor.

Atmospheric pressure accuracy

The barometer possesses the following characteristics:

- Overall accuracy (given by manufacturer): $<0.005\%FS$
5.3. INSTRUMENTATION

- Resolution/repeatability (given by manufacturer): < 0.002%FS
- Long term stability (given by manufacturer): < 0.01%FS/annum
- Over pressure capability: 3 x FS
- Full range: 15psia (105kPa)

An estimation of the accuracy can therefore be establish as follow:

\[ U = \pm \sqrt{0.005^2 + 0.002^2 + 0.01^2} = \pm 0.011\%FS = \pm 11.5Pa \]  (5.2)

5.3.2 Wind tunnel instrumentation

5.3.2.1 Settling chamber

Stagnation conditions, i.e pressure and temperature, should be measured in the settling chamber in which the flow velocity should ideally be zero. However, it appeared that the static pressure measured in the settling chamber with a simple pressure tapping did not exactly match the total pressure value, revealing the existence of a dynamic pressure component. As a result, it was decided to use a total pressure probe and a total temperature probe to measure the stagnation conditions. Their location in the settling chamber is illustrated in Figure 5.11. Furthermore, a static pressure tapping was also used to evaluate the dynamic pressure component and get an estimation of the velocity component within the settling chamber. Results are presented and discussed in the "Air flow quality" section further below.

![Figure 5.11: Stagnation probes location in the settling chamber](image)

A detailed description of the total pressure and total temperature probes is presented in Appendix C. In particular, the accuracy on stagnation measurements were evaluated to be ±42Pa and ±0.7K.
5.3.2.2 Test section

The instrumentation in the test section mainly consists of pressure tappings used either to set a given operating point, or gather information about the current operating conditions. A complete overview of the test section instrumentation is presented below and illustrated in Figure 5.3.

Inlet instrumentation

A set of three pressure tappings have therefore been located on the lower wall of the test section, exactly 115mm upstream of the edge of test object’s insert hole, that is to say at 185mm upstream of the beginning of the bump (x=-0.185m). Although these pressure taps were originally designed to be used to calculate the inlet Mach number, it appeared that they were situated too far from the test objects and that the Mach number was still increasing until the test objects. They were consequently not used to determine the inlet Mach number but rather as a source of information to check the pressure gradient at the inlet for instance.

Side window instrumentation

Two types of side windows were basically used during experiments. A set of optical glass windows (Figure 5.12 left) with fine tolerances on the thickness and planarity of the surfaces were used during Schlieren and oil visualizations (see "measuring techniques” section), whereas a set of plexiglas windows (Figure 5.12 right), partly described in the "Test object instrumentation” section, were used during pressure measurements.

![Figure 5.12: Optical glass (left) and plexiglas (right) side windows](image)

Whereas the optical glass side windows do not contain any instrumentation, the plexiglas windows were equipped with pressure taps and removable plugs, which, themselves, can contain steady state or unsteady pressure taps, probe access holes as shown in Figure 5.13.

In particular, two pressure taps were placed on each of the two side windows (left and right), at mid-height of the channel (z=0.06m) and at the station x=-0.09m, that is to
5.3. INSTRUMENTATION

Figure 5.13: Set of removable plugs to be inserted in the plexiglas side windows

say 90mm upstream of the beginning of the bump in order to match the inlet boundary of the numerical simulations. The inlet Mach number was thereafter calculated using the average between the two pressure values measured on each separate side window. As a result, the experimental inlet Mach number value could thereafter directly be compared with the values obtained with numerical simulations.

Upper wall instrumentation

As mentioned in the "experimental apparatus" section, the upper wall also feature a support part completely symmetrical to the one located on the lower wall, in which can inserted either a test object or a simple blinder to obtain a flat wall.

Consequently, two different plexiglas flat plates were used as blinders. The first was free of instrumentation holes and was used for visualizations such as oil-paint visualization. The second one featured pressure tapings and threaded holes for unsteady transducers glued in pipe for protection.

Outlet instrumentation

The outlet condition was monitored using a pressure tap located on the upper wall in the center of the channel (y=0.05m), exactly 290mm downstream of the beginning of the bump, at the abscissa x=0.29m, as seen in Figure 5.9.

5.3.2.3 Air flow quality

Once the new test facility had been designed and manufactured, an experimental investigation of the air flow quality was performed with the aim to characterize the flow field in the test section. This investigation mainly addresses the following issues. First, the acoustic signature of the wind tunnel had to be characterized in order to validate the
use of this facility for SBLI studies. With the aim at establishing the possible sources of noise among the different components of the wind tunnel, both the amplitude level and frequency content of the acoustic noise, inherent to the facility, had to be determined. Secondly, a comparison between the imposed back pressure perturbations and the noise level had to be performed in order to determine a possible interaction while "artificially" perturbing the shock. Finally, the repeatability and reproducibility of the measurements have to be either confirmed in order to proceed with further measurements.

The entire air flow quality investigation is fully addressed in Appendix D whereas the main have been summarized below:

Flow quality in settling chamber

A flow velocity component was found in the settling chamber. Its value linearly increases with the inlet Mach number until a maximum of 33m/s at approximately $P_{out} \approx 108kPa$. For lower outlet static pressure values, the density increases but the velocity component within the settling chamber remains constant as the flow if choked in the test section.

Flow quality in test section

High amplitude time varying pressure fluctuations were found in both the settling chamber and test section of the wind tunnel. After investigation, it appeared that those perturbations were actually characterized by low amplitude frequency components with random phase angles. The origin of the different inherent perturbations could be correlated upstream and downstream components of the facility. Furthermore, it was clearly established that the imposed downstream perturbation featured a much high amplitude level at a distinct frequency, which could therefore be analyzed by Fourier decomposition.

Measurements repeatability and reproducibility

Considered as the most sensitive and most important parameter to establish both the repeatability and reproducibility of the measurements in the present facility, the shock position was monitored using the conventional Schlieren technique. Considering both the experimental techniques accuracy and the slight differences between each operating conditions reached during the experiments, the measurements were considered to have a fairly good repeatability. On the other hand, the reproducibility was not found so good, mainly due to the high sensitivity of the shock towards the operating parameters. Especially a high sensitivity was observed towards the outlet static pressure.

Start up and transient regime of the compressor

It was observed, directly after the start-up of the compressor, a transient regime of approximately 40 minutes in order to obtain stable operating conditions. This observation mainly resulted from the low iteration between the cold water mass flow and the air
5.3. INSTRUMENTATION

flow temperature measurement within the cooler. The transient time therefore directly depends on the operating conditions and could be reduced at low mass flow and high temperature flow experiments.

5.3.3 Orifice plate

The mass flow within the test section was measured for each OP using an orifice plate device located in the pipeline upstream of the settling chamber (Figure 5.2.1). This primary device is a simple metal plate in which a circular aperture has been machined. The rate of flow can be determined from previous calibration sheets and measurements of the upstream static pressure and pressure drop through the perforated plate. A detailed description of the working principle, calculation procedure and error estimation is presented in Appendix E. As a result, the mass flow value was estimated to lie within a 95% Confidence Interval equal to ±0.032kg/s.

5.3.4 Test objects instrumentation

The two test objects were instrumented towards different measuring techniques, explicitly steady state and high speed pressure measurements as well as steady state and unsteady hot film measurements. However, in order to perform the foreseen experiments in a more modular and convenient way, part of the instrumentation was designed to be common to both test objects. Consequently, the same transducers, data acquisition method, and post treatment tools could be used either on the 2D or 3D bumps.

The following paragraphs intend to present both common and specific instrumentation relative to the different test objects, as well as the external instrumentation used together with the test objects, that is to say the instrumentation between the sensors and the measuring equipment.

5.3.4.1 Common pressure instrumentation

The main idea about the pressure measurement instrumentation was to be able to use both steady state and unsteady instrumentation not only on the same test object, but either on the 2D or 3D test objects. Consequently, the same pressure tapping design, foreseen to be used either during steady state and unsteady pressure measurements was machined on both test objects and is presented in Figure 5.14.

Basically, the pressure measurement tapping design consists of having, within a large cavity machined on the base of each test object, a certain number of 5mm deep small holes in which the transducer's instrumentation will be inserted. The deepness of the cavity was chosen to be 5mm due to thickness constrains on the 2D test object, and the diameter of the small holes was set to 3.2mm to have enough room for the transducer and still to obtain a fine pressure tap layout. On the bump surface, all pressure taps were drilled perpendicular to the surface with a length and diameter equal to 2mm.
Figure 5.14: Pressure taps design

and 0.6mm respectively. Finally, a 1.4mm diameter hole was drilled in order to connect both sides on each test object. Consequently, the length \(d\) of the 1.4mm diameter hole depends on the location of the pressure tap, i.e. the thickness of the bump at this position.

The pressure taps layout as well as the different instrumentation used either in steady state or unsteady pressure measurements is presented in the following paragraphs.

5.3.4.2 3D bump instrumentation

As described in a previous section, the 3D test object consists of a 25mm thick base on which was milled a three-dimensional shape, typically, the 3D bump. Subsequently, a 7mm deep cavity was milled out in this base and a 2mm thick plate was used to close the cavity and contain the instrumentation. A number of 350 pressure taps were machined according to the common design presented in the previous paragraph, and following the precise layout, illustrated in Figure 5.15 and designed to fit the shock structure both in its quasi-steady state and unsteady motion.

5.3.4.3 2D sliding bump specific instrumentation

The two-dimensional test object was designed and instrumented with the idea to map the entire channel in order to check the three dimensionality of the flow over a 2D geometry, or, in other words, check the influence of the side walls. As a result, it was decided to have a larger test object instrumented for both pressure and hotfilm measurements, and slide it through the width of the channel. However, this solution has raised a few problematic issues which are going to be further discussed in the following paragraphs.

5.3.4.3.1 Pressure instrumentation

First of all, in order to be able to slide the long 2D bump through the side walls without changing the dimensions of the large openings foreseen for the side windows, it
5.3. INSTRUMENTATION

Thereafter, following the common pressure tap design, a 6mm deep cavity was milled out on approximately half of the base, and a 1mm thick plate was then used to close the cavity and contain the instrumentation. The pressure tap layout was organized in three staggered rows distant of 5mm from each others and located in the middle of the test object as illustrated in Figure 5.16. Each row itself contains exactly 52 pressure taps distant of 4.5mm from each other. Besides, each row was staggered from the adjacent one with a 1.5mm step so that, by sliding each respective row to the same location in the channel, the final resolution of the pressure measurements would be 1.5mm.

5.3.4.3.2 Hotfilm instrumentation

A hotfilm gage basically consists of a 20\(\mu\)m thick, 0.1mm wide and 1.4mm long nickel element deposited onto a 120\(\mu\)m thick polyamide substrate foil. Each nickel gage is subsequently attached to a 50\(\mu\)m thick, 0.8mm large and 10cm long copper leads also deposited on the polyamide foil. A typical hotfilm sensor element is presented in Figure 5.17.

The hotfilm sheet used in these experiments contained exactly hundred of the above described gages, distant of 5.4mm from each others. The measuring length over the bump thus extends from the abscissa \(x \simeq -0.0585m\) to \(x \simeq 0.193m\). The polyamide foil was directly glued onto the 2D bump surface using a special double sided tape. As a result, the total thickness of the hotfilm instrumentation was measured to be about 0.25mm and could therefore disturb the flow the same way as a forward step would do. Consequently,
a 0.25mm groove was directly etched onto the 2D bump surface, at the location where the hotfilm were to be glued, as illustrated in Figure 5.16. The etched groove was furthermore extended around the corners and underneath the test object so that the hotfilm sheet could be folded around the 2D bump to better stick to the surface and present no discontinuity to the flow. It should also be noted that a particular attention was paid when applying both the double sided tape and thereafter the polyamide foil so that no air bubble would be trapped under the hotfilm sheet and modify the 2D bump profile. Finally, the hotfilm sensors were divided into 6 groups, each of them containing 16 sensors connected to the respective channels of the CTA via the multiplexer. For a given group, the distance between two consecutive running hotfilm sensor was therefore 32mm to minimize interaction between the different heating elements.
5.3.4.3.3 Traverse mechanism

The traverse mechanism designed to slide the 2D test object simply consists of a freely rotating threaded rod on which is screwed another part itself fixed to the sliding test object as illustrated in Figure 5.18. By rotating the threaded rod using a manual wheel, the intermediate part moves forward or backward, pushing or pulling the 2D bump through the channel’s width. A digital display Vernier calliper is also fixed to the test object and measures the relative position of the bump. The reference position is usually set up when one of the pressure taps rows is located in the channel’s corner, underneath a side wall. Since the Vernier calliper has a fairly good accuracy (less than a hundreds of a millimeter), the spatial accuracy therefore mainly depends on the reference position, which is roughly about ±0.1\,mm.

![Figure 5.18: Two dimensional bump sliding mechanism](image)

5.3.4.3.4 Side wall sealing method

One of the most problematic issues was to be able to slide the 2D test object without having any leakage on the side walls that would perturbate the flow field by injecting or sucking out a part of the mass flow. The gap between the plexiglas side windows and the 2D bump had thus to be really small, but on the other hand could not be too tight around the 2D bump in order not to scratch the surface when sliding the test object. Besides, the same set of plexiglas side windows had to be used whether using the sliding or a fixed test object.

Consequently, it was decided to first have an exchangeable cassette directly within the plexiglas side windows as seen in Figure 5.19, which would allow us to keep the same plexiglas side windows already instrumented when switching between the sliding or a
fixed test object. Secondly, a groove was milled in the large cassette and an air inflated O-ring was used to seal the small gap between the plexiglas material and the 2D bump. A complete drawing is presented in Figure 5.20 whereas Figure 5.21 illustrates the design and the working principle.

![Figure 5.19: Large cassettes located inside the plexiglas side window](image)

![Figure 5.20: Large cassette with inflated O-ring sealing mechanism](image)

Basically, the O-ring located in the groove which follows the 2D bump profile is inflated prior to each measurement, and deflated when sliding the 2D test object.

5.3.4.4 Steady state pressure instrumentation

The steady state pressure instrumentation basically consists of connecting the pressure taps to the scanners using plastic tubes. This is done by plugging stainless steel tubes into the 3.2mm holes located in the test object’s cavity and then connecting them to the plastic tubes. Both in order to hold the tubes into the holes and to avoid any leakage, a couple of tight O-rings were placed around the stainless steel tube. On the other side,
the plastic tubes were connected to a "quick connector" as seen in Figure 5.22, itself connected to the scanners of the steady state pressure measurement system.

2D test object

The two-dimensional test object was thus instrumented with 156 pressure taps organized in three staggered rows connected to three different quick connectors respectively. A picture of the steady state instrumentation on the 2D sliding bump is presented in Figure 5.23.

3D test object

The three-dimensional test object was similarly instrumented with 350 pressure taps also connected to plastic tubes and thereafter to 7 quick connectors following a precise connection table. The steady state pressure instrumentation on the 3D test object is presented in Figure 5.24.
5.3.4.5 Unsteady flow instrumentation

5.3.4.5.1 High frequency response transducers

Unsteady pressure measurements have been performed using ultraminiature (1.6x9.5mm) XCQ-062 Kulite transducers, which feature a unique integrated circuit sensor. The latter consists of a miniature silicon diaphragm onto which a dielectrically isolated piezoresistive Wheatstone bridge is atomically bonded. Basically, the bridge is unbalanced as the membrane vibrates due to pressure fluctuations, and one can measure the tension corresponding to those fluctuations. The miniaturization process yields a marked increase in the natural frequencies of the transducers making them especially suitable for shock pressure measurements in addition to static pressure measurements.

Moreover, integrated sensors are supplied with an external compensation module,
5.3. INSTRUMENTATION

which provides zero balance and temperature compensation. These modules contain only temperature insensitive trimming resistors and do not need to be at the same temperature as the transducer. Those latter are also provided with special screens to increase particle protection while having a minimal effect on frequency response and are presented in Figure 5.25.

Figure 5.25: XCQ-062 Kulite transducer

Whereas the pressure range of those sensors is 170kPa bar absolute, they can stand over pressures up to two times the rated pressure range with no change in calibration, which means that the transducers can actually measure a pressure from vacuum to 240kPa. The accuracy of such transducer depends on three parameters: the non-linearity, the hysteresis, and the repeatability. Kulite manufacturer provides a combined value for the non-linearity and the hysteresis, which in our case is about ±0.1% of the full pressure range. The effect of repeatability on the accuracy is also estimated to be about ±0.1% full scale. By summing the square values of all separate accuracies, the overall accuracy for the XCQ-062 Kulite transducers can be estimated to be around ±0.141%, that is to say ±240Pa for a pressure range from 0 to 170kPa.

5.3.4.5.2 Protection of transducers

Considering the number of pressure taps and the fragility of the fast response transducers, a protective hand held casing was designed to quickly exchange the pressure transducer from one location on the test object to another, avoid direct shocks on the sensitive transducer, and protect the four tiny electrical wires which are connected to the piezoresistive Wheatstone bridge bonded onto a vibrating membrane.

The protective casing consists of a 5mm outer diameter pipe in which a Kulite transducer has been non-permanently glued. The pipe was machined on one side down to 3.2mm to fit the pressure holes located on the test objects. Two O-rings were placed close
to the transducer to avoid leakage and limit the risk of acoustic resonance by minimizing the volume of air above the transducer. A special wax was used instead of permanent glue to fix the transducer into the pipe. The wax was chosen to melt down around 75°Celsius and allow a later removal of the transducer if needed. The tiny wires, which are especially fragile close to the transducer, were partly folded and “frozen” within the wax. The rest of the electrical wires were thereafter wrapped around a tiny Nylon cable which was glued between the protective casing and a 15 pin connector in order not to excessively pull and break the wire by mishandling. Finally, a spiral wrap plastic cable was wrapped around both the electrical wires and the Nylon cable to hold everything together and avoid any relative vibrations of the wires which would introduce a magnetic perturbation into the electric recorded signal. A complete illustration of the protective casing is presented in Figure 5.26(a).

![Illustration of the protective casing around a Kulite transducer](image)

**Figure 5.26**: Illustration of the locking mechanism for unsteady pressure instrumentation

The protective pipes were thereafter inserted into the 3.2mm pressure holes through a rectangular hole in the flat plate and remained in position due to a locking mechanism as illustrated in Figure 5.26.

### 5.3.4.5.3 Static calibration of transducers

The static calibration of the fast response transducers addresses several issues. First of all, one might want to check the linearity of the transducer, that is to say if the bridge output voltage is a linear function of the pressure applied onto the membrane. Secondly, the two coefficients which determine the transfer function between the transducer’s voltage and the pressure, have to be evaluated over the full pressure range. Finally, the last but not least of the check-up would be to determine whether or not there is a time drift in the transducer’s transfer function. This might not have a huge importance for a qualitative study, but the exact coefficients have to be established for a quantitative analysis of the pressure amplification and comparison with other similar transducers.
The experimental procedure and data reduction method are presented in detail in Appendix F. The main conclusions drawn out from the static calibration procedure have been summarized below:

- The linearity of the fast response transducers was measured within the specified (or even better) value given by the manufacturer.

- The sensitivity and offset coefficients of the linear transfer function was calculated for each transducer and stored in a database for easy access and future post treatment.

- An estimation of the time drift was evaluated for one transducer and it was found within the repeatability specification. A six point calibration was however suggested prior and after each unsteady measurements in order to limit the systematic error contribution.

### 5.3.4.5.4 Sound propagation related limitations

A few remarks should be introduced at this point regarding the propagation of acoustic waves in closed pipes and the influence on the unsteady pressure measurements. Basically, the air inside the pressure taps and capillarity pipes can be assimilated to an oscillating system with given dynamic properties. Among them is a phenomenon called resonance characterized by a critical frequency at which standing waves are sustained within the system. The implication in high speed measurements is a strong amplification of the measurand and a $180^\circ$ phase shift as the sampling frequency is close to this resonance frequency. This always results in non-exploitable perturbed results and can even sometimes damage the transducers. It is thus of high importance to characterize the dynamic behavior of the corresponding "pressure tap and capillarity pipe" system in order to set the sampling frequency below the critical frequency, even if transducers are flush mounted.

The other important property of the considered dynamic system is its transfer function, that is to say the damping and phase lag that will affect the pressure fluctuations when travelling through the system. Being beyond the scope of the present discussion, which is to evaluate the critical sampling frequency, a complete section will be dedicated to the evaluation of the transfer function of all capillarity pipes in the "Measuring techniques" section.

Two basic modelisations of the capillarity pipes are presented in Appendix G to evaluate the corresponding resonance frequency. Those models do not pretend to provide the very exact value of the critical frequency but provide an helpful estimation of the characteristic value. The reader is advised to consult the work performed by Bergh and Tijdemma (1965) and Bohn and Schnittfeld (1965) on the subject for more complex modelisations.

As a result, according to the instrumentation of the test objects, the resonance frequency for the 3D and 2D bumps approximately corresponds to 3.5kHz and 4.8kHz respectively. Comparatively with the organ pipe model, the effect of the container’s volume seems to be tempered as the length of the tubes increases.
To conclude, it should be noted that in the worst case, that is, on the longest pressure hole with an imposed perturbation of 500Hz, one can still obtain the first four harmonics by sampling at 4kHz with a low pass filter at 2kHz. However, this case only correspond to a few pressure holes on the 3D bump and most of the high speed pressure measurements will be performed at 8kHz with a 4kHz low pass filter thus allowing the calculation of many more harmonics, depending on the imposed frequency.

### 5.3.5 Perturbation generator instrumentation

The perturbation generator consists, as mentioned previously, of an elliptical cam placed downstream of the test section and rotated by a high speed motor. The related instrumentation is presented below.

#### 5.3.5.1 High speed motor instrumentation

The high speed motor used to fulfill the 500Hz frequency requirement was a Faulhaber\(^2\) brushless DC servo-controlled motor, capable of reaching speeds up to 20,000rpm in continuous run and 27,000rpm in short run. An internal optical encoder directly coupled to the servo amplifiers (Figure 5.27) initiates both a current feedback loop for the power regulation and a speed feedback loop to stabilize and maintain a constant rotating speed. Furthermore, in order to compensate for the aerodynamical forces acting on the rotating cam, a metallic disk was placed on the shaft to increase the inertia and smoothen the torque variations under a single revolution.

![Figure 5.27: Basic block-diagram of the servo amplifier for speed control with encoder](image)

The optical encoder basically consists of a rigid disk with small aperture holes rotating with the shaft and placed in between a set of emitting and receiving tiny diodes. An electronic signal is then emitted each time a hole is inline with both the diodes. The optical encoder was chosen to have two channels and to deliver two TTL output signals which could be used either to monitor the rotating speed or to trigger external measurements. The first TTL signal, which only delivers one pulse per revolution, was used to

\(^{2}\)Minimotors AB, 6980 Croglio, Switzerland
5.3. INSTRUMENTATION

determine the position of the cam under measurements, and basically sets a reference\textsuperscript{3} to coordinate all unsteady measurements. The second TTL signal which delivers, on the other hand, 360 pulses per revolution, was connected to a digital counter to monitor the rotating speed frequency with a good accuracy.

5.3.5.2 Electronic counter

As described above, the optical encoder delivers a one pulse per revolution TTL signal. However, most of the experimental measuring systems do not feature the high frequency counter required to "capture" the impulse signal in its "high" level voltage position and basically would not detect the TTL pulse at all. Indeed, as the pulse frequency increases the time duration of the high level voltage position of the pulse decreases down to a few micro seconds and thus requires a fairly high sampling frequency to be detected (up to a million Hertz).

Consequently, a 1bit digital counter was interfaced between the output TTL signal from the servo amplifier and the measuring equipment. Basically, a set of logical gates built in a ceramic ship (type 74ls90) initiates a change of state in the low-high level of the output TTL signal each time a high level state is input. As a result, for each pulse input in the digital counter the output level is switched from low to high or vice-versa as illustrated in Figure 5.28. The original frequency is thus divided by two, but most important, the TTL signal is now periodic and useable by other measuring equipments.

![Figure 5.28:](image)

5.3.5.3 Rotating speed accuracy

The digital frequency meter basically counts the pulses of the TTL signal output from the encoder and measures the sampling time. Naturally, to get the best accuracy possible, the TTL signal delivering 360 pulses per revolution was connected to the digital counter.

\textsuperscript{3}It should be noted that one revolution of the cam corresponds to two perturbation cycles as the cam is ideally symmetric
Table 5.6: Motor fluctuations at different rotating speeds

<table>
<thead>
<tr>
<th>Test at 9kHz (50Hz perturbation)</th>
<th>Average over 10 samples</th>
<th>Average over 100 samples</th>
<th>Average over 100 samples</th>
<th>Drift after 4min (averaged value)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean</td>
<td>9002Hz</td>
<td>9002Hz</td>
<td>8991Hz</td>
<td>( \Delta f = 15 \text{Hz} )</td>
</tr>
<tr>
<td>Max</td>
<td>9027Hz</td>
<td>9048Hz</td>
<td>9081Hz</td>
<td></td>
</tr>
<tr>
<td>min</td>
<td>8979Hz</td>
<td>8965Hz</td>
<td>8942Hz</td>
<td></td>
</tr>
<tr>
<td>( \Delta f )</td>
<td>48.5Hz</td>
<td>82Hz</td>
<td>140Hz</td>
<td></td>
</tr>
<tr>
<td>rms</td>
<td>18Hz</td>
<td>18Hz</td>
<td>21Hz</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test at 18kHz (100Hz perturbation)</th>
<th>Average over 10 samples</th>
<th>Average over 100 samples</th>
<th>Average over 100 samples</th>
<th>Drift after 4min (averaged value)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean</td>
<td>18003Hz</td>
<td>17999Hz</td>
<td>18003Hz</td>
<td>( \Delta f = 14 \text{Hz} )</td>
</tr>
<tr>
<td>Max</td>
<td>18075Hz</td>
<td>18127Hz</td>
<td>18139Hz</td>
<td></td>
</tr>
<tr>
<td>min</td>
<td>17924Hz</td>
<td>17899Hz</td>
<td>17898Hz</td>
<td></td>
</tr>
<tr>
<td>( \Delta f )</td>
<td>150Hz</td>
<td>227Hz</td>
<td>241Hz</td>
<td></td>
</tr>
<tr>
<td>rms</td>
<td>45Hz</td>
<td>37Hz</td>
<td>42Hz</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test at 45kHz (250Hz perturbation)</th>
<th>Average over 10 samples</th>
<th>Average over 100 samples</th>
<th>Average over 100 samples</th>
<th>Drift after 4min (averaged value)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean</td>
<td>45002Hz</td>
<td>44993Hz</td>
<td>44930Hz</td>
<td>( \Delta f = 44 \text{Hz} )</td>
</tr>
<tr>
<td>Max</td>
<td>45085Hz</td>
<td>45131Hz</td>
<td>45125Hz</td>
<td></td>
</tr>
<tr>
<td>min</td>
<td>44935Hz</td>
<td>44858Hz</td>
<td>44770Hz</td>
<td></td>
</tr>
<tr>
<td>( \Delta f )</td>
<td>150Hz</td>
<td>272Hz</td>
<td>355Hz</td>
<td></td>
</tr>
<tr>
<td>rms</td>
<td>52Hz</td>
<td>45Hz</td>
<td>52Hz</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test at 90kHz (500Hz perturbation)</th>
<th>Average over 10 samples</th>
<th>Average over 100 samples</th>
<th>Average over 100 samples</th>
<th>Drift after 4min (averaged value)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean</td>
<td>90018Hz</td>
<td>90010Hz</td>
<td>89981Hz</td>
<td>( \Delta f = 36 \text{Hz} )</td>
</tr>
<tr>
<td>Max</td>
<td>90178Hz</td>
<td>90180Hz</td>
<td>90222Hz</td>
<td></td>
</tr>
<tr>
<td>min</td>
<td>89836Hz</td>
<td>89891Hz</td>
<td>89742Hz</td>
<td></td>
</tr>
<tr>
<td>( \Delta f )</td>
<td>340Hz</td>
<td>288Hz</td>
<td>479Hz</td>
<td></td>
</tr>
<tr>
<td>rms</td>
<td>103Hz</td>
<td>63Hz</td>
<td>69Hz</td>
<td></td>
</tr>
</tbody>
</table>

The main source of inaccuracy in the perturbation generator instrumentation is the stability of the rotating speed. It is indeed difficult to state on the regularity of the rotating speed under a revolution since there is no real way, in the present case, to make sure that the pulses are evenly spaced over a full revolution. It is pretty obvious that any aerodynamic fluctuation in the test section will result in a tiny variation of the torque load. Therefore, a metallic disk was placed on the shaft to add some inertia and smoothen any rotating speed fluctuations under a single revolution.

Additionally, the long term stability of the perturbation generator was checked by monitoring the rotating speed, after a warm up period of 15 minutes, for different rotating speeds and different number of samples. Results are presented in Table 5.6. Statistical moments have been calculated out of each respective number of samples. First, the mean value obtained during the N-samples measurement is presented. Then the maximum and minimum values and the difference between them are reported. The standard deviation (or rms value) has finally been calculated and should be compared with the extreme values interval. It should also be noted that the frequencies reported in Table 5.6 correspond to the 360 pulses per revolution TTL signal and should therefore be divided by 360 and then multiplied by 2 (since the elliptical cam is symmetrical) in order to obtain values corresponding to the real perturbation frequency at which the shock will be excited.

The main result of this investigation is basically that the rotating speed fluctuations are, if we consider a 95% confidence interval (2 times rms values), about 0.2Hz, 0.5Hz,
5.4. MEASURING TECHNIQUES

0.6Hz, and 0.8Hz at 50Hz, 100Hz, 250Hz and 500Hz respectively. This corresponds, in percentage, to 0.004%, 0.005%, 0.024% and 0.0016% for the frequencies 50Hz, 100Hz, 250Hz and 500Hz respectively. It is also noteworthy that the rms value is much lower than the maximum amplitude of the fluctuations.

Additionally, a time drift over 4 minutes was calculated by measuring the variation of the average value over 100 samples. The results showed a time drift of 0.083Hz, 0.78Hz, 0.25Hz and 0.2Hz at 50Hz, 100Hz, 250Hz and 500Hz respectively, which is acceptable since the unsteady measurements will actually only take a few seconds.

5.4 Measuring techniques

In the present section will be presented the different measuring techniques used in this work. First, the measuring system will be introduced and followed by the acquisition method, data reduction, and post processing. Finally, an estimation of the accuracy or uncertainty of measurement related to each specific technique will be presented.

5.4.1 Steady state pressure measurements

5.4.1.1 Data acquisition system

Steady state pressure measurements will be performed using a highly parallel and modular data acquisition system. The 8400 digital pressure measurement system is a microprocessor-based system consisting of three separate components: the main chassis or System Processor (SP), a high speed input units interface, and a host interface as sketched in Figure 5.29.

Data from mixed Electronic Pressure Sensors (ESP) can be handled by separate input units, each one having its own firmware-controlled microprocessor, logic control and local memory. Any of those individual units thus receives its own unique scan list and sample time constraints, and asynchronously contributes, at its own rate, to the host’s received data stream. Averaging can also be done inside the unit, thus reducing the bandwidth handled by the host, which can be any computer that supports parallel (GPIB) and serial (IEEE-488) interfaces.

A Scanner Digitizer Unit (SDU) performs efficient analog-to-digital conversion from multiple local sensor scanners. The sampling rate of sensor scanning is about 50,000 channels/sec per SDU. Data from each scanner pressure port are received via the scanner interface and converted to a 16-bits raw data string. Different Pressure Standard Units (PSU) are available for different range of pressure measurements, typically ±7kPa, ±35kPa, and ±100kPa relative to atmosphere.

Each different PSU requires a single Pressure Calibrate Unit (PCU), which consists of pneumatic valves, servo controlled and a high accuracy pressure transfer standard. Under SP control, the PCU switches the calibration valves within the scanners to the calibrate...
position and applies five calibration pressures to each scanner. The SP reduces the calibration data, and calculates the coefficients of a fourth-order polynomial function, which is thereafter used as a calibration characteristic equation for each transducer. Either complete engineering-unit or faster raw data may be delivered to the host.

Further details about the PSI system itself can be found in the "error and accuracy evaluation" paragraph or directly in the PSI (1993) of course.

5.4.1.2 Data acquisition method

Steady state pressure measurement first consists in reaching the desired operating conditions and then start acquiring data. As described earlier, different valves control both the mass flow and the pressure level within the test section, and by adjusting one against the others, one can set up the inlet Mach number and Reynolds number. In regards to the range of the unsteady pressure transducers which is about 170kPa, it was decided to set up the total pressure and temperature in the settling chamber respectively to approximately 160kPa and 303K. The desired operating conditions were thereafter obtained by adjusting the outlet static pressure level.

Hence, the steady state pressure measurements were performed using exclusively the ±100kPa scanners of the PSI 8400 system. The sampling frequency and the number of

---

4The Kulite transducers can effectively stand over-pressure around two times their rated range (thus, up to 240kPa), but previous studies (Tijdeman and Seebass, 1980; Atassi et al., 1994b; Ferrand et al., 1996) pointed out that non-linear amplification of the pressure distribution on the surface could occur in unsteady transonic flows
samples were respectively set to 20Hz and 2000 samples for the two following distinct considerations. The first reason was to obtain a sampling time long enough to correctly describe at least one period of the lowest frequencies (of about 0.02Hz) observed during the air flow quality investigation. The second reason was to obtain a number of samples large enough to assume a statistical independence and a normal distribution of the results required for the later evaluation of statistical moments.

In addition to the wall pressure measurements, the total pressure and total temperature were measured in the settling chamber using the probes described in the respective instrumentation sections. Two static pressure taps located on both side walls (see wind tunnel instrumentation) were also used to measure the inlet static pressure in order to calculate the inlet Mach number.

An important remark concerning the adjustment of the operating conditions should be made before describing in details the measurement procedure for each respective test object. Considering the type of valve used, it was not possible to set up exactly repetitive operating conditions with a high accuracy. For instance, a slight change in the outlet valve position induces a large change (of about 2-300Pa) in the outlet static pressure. It was therefore a really sensitive and time consuming procedure to try reaching the exact operating conditions.

5.4.1.2.1 Two-dimensional sliding test object

Steady state pressure measurements in the two-dimensional nozzle were performed across the wind tunnel channel by traversing the instrumented pressure taps rows (see "Instrumentation section") from one side wall to the other. Each row of pressure taps was thus successively positioned at the same location in the test section according to the following layout. Assuming lower transversal gradients in the center of the channel, pressure measurements were performed every 5mm in this region, whereas a 2mm step was taken in the vicinity region, 20mm close to the side walls as illustrated in Figure 5.30.

![Figure 5.30: Experimental steady state pressure measurement layout on 2D test object](image)

It should be noted that, according to the sampling time and number of traverses, the overall measurement time for each operating point was about 3-4 hours, and that during
this time the atmospheric conditions might have changed and induced a very slight drift in the operating conditions, which are presented in Table 5.30.

<table>
<thead>
<tr>
<th>$p_{inlet}$ [kPa]</th>
<th>$T_{inlet}$ [K]</th>
<th>$M_{inlet}$ [-]</th>
<th>$p_{outlet}$ [kPa]</th>
<th>$Q_m$ [kg/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>160.29</td>
<td>303.3</td>
<td>0.641</td>
<td>118.05</td>
<td>3.497</td>
</tr>
<tr>
<td>160.24</td>
<td>303.2</td>
<td>0.656</td>
<td>116.11</td>
<td>3.541</td>
</tr>
<tr>
<td>160.03</td>
<td>303.2</td>
<td>0.671</td>
<td>114.07</td>
<td>3.579</td>
</tr>
<tr>
<td>159.85</td>
<td>303.3</td>
<td>0.683</td>
<td>111.95</td>
<td>3.607</td>
</tr>
<tr>
<td>159.81</td>
<td>303.2</td>
<td>0.692</td>
<td>110.00</td>
<td>3.628</td>
</tr>
<tr>
<td>160.11</td>
<td>303.3</td>
<td>0.650</td>
<td>108.01</td>
<td>3.650</td>
</tr>
<tr>
<td>160.09</td>
<td>303.1</td>
<td>0.702</td>
<td>106.07</td>
<td>3.662</td>
</tr>
<tr>
<td>160.27</td>
<td>303.2</td>
<td>0.701</td>
<td>104.09</td>
<td>3.662</td>
</tr>
<tr>
<td>160.20</td>
<td>303.2</td>
<td>0.701</td>
<td>102.19</td>
<td>3.665</td>
</tr>
<tr>
<td>159.94</td>
<td>303.3</td>
<td>0.701</td>
<td>100.18</td>
<td>3.665</td>
</tr>
<tr>
<td>160.49</td>
<td>303.1</td>
<td>0.700</td>
<td>97.82</td>
<td>3.664</td>
</tr>
</tbody>
</table>

Table 5.7: Operating conditions for steady state pressure measurements in 2D nozzle

5.4.1.2.2 Measurement procedure for the 3D test object

In the same way, the steady state pressure measurements in the three-dimensional nozzle were performed following the layout described in the test object instrumentation section and presented again in Figure 5.31. It should be noted that the black rectangle around the measurement area symbolizes the edges of the test object.

![Experimental steady state pressure measurement layout on three dimension test object](image)

Figure 5.31: Experimental steady state pressure measurement layout on three dimension test object

The measurement procedure therefore consisted in reaching an operating point and successively connect one of the seven quick connectors used in the instrumentation to the PSI system while carefully sealing the six others. The achieved operating conditions are subsequently presented in Table 5.8.
5.4. MEASURING TECHNIQUES

Table 5.8: Operating conditions for steady state pressure measurements in 3D nozzle

<table>
<thead>
<tr>
<th>$P_{\text{inlet}}$</th>
<th>$T_{\text{inlet}}$</th>
<th>$M_{\text{inlet}}$</th>
<th>$P_{\text{outlet}}$</th>
<th>$\dot{m}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>[kPa]</td>
<td>[K]</td>
<td>[-]</td>
<td>[kPa]</td>
<td>[kg/s]</td>
</tr>
<tr>
<td>160.09</td>
<td>302.9</td>
<td>0.549</td>
<td>128.09</td>
<td>3.180</td>
</tr>
<tr>
<td>160.06</td>
<td>303.5</td>
<td>0.569</td>
<td>125.99</td>
<td>3.249</td>
</tr>
<tr>
<td>160.24</td>
<td>303.4</td>
<td>0.588</td>
<td>124.31</td>
<td>3.314</td>
</tr>
<tr>
<td>160.05</td>
<td>303.5</td>
<td>0.607</td>
<td>121.99</td>
<td>3.374</td>
</tr>
<tr>
<td>159.73</td>
<td>303.5</td>
<td>0.624</td>
<td>119.82</td>
<td>3.421</td>
</tr>
<tr>
<td>159.92</td>
<td>303.5</td>
<td>0.638</td>
<td>117.89</td>
<td>3.478</td>
</tr>
<tr>
<td>160.05</td>
<td>303.4</td>
<td>0.653</td>
<td>116.04</td>
<td>3.526</td>
</tr>
<tr>
<td>160.06</td>
<td>303.5</td>
<td>0.668</td>
<td>113.99</td>
<td>3.566</td>
</tr>
<tr>
<td>159.70</td>
<td>303.5</td>
<td>0.676</td>
<td>112.00</td>
<td>3.581</td>
</tr>
<tr>
<td>159.87</td>
<td>303.5</td>
<td>0.685</td>
<td>109.89</td>
<td>3.608</td>
</tr>
<tr>
<td>159.92</td>
<td>303.4</td>
<td>0.691</td>
<td>107.73</td>
<td>3.625</td>
</tr>
<tr>
<td>160.08</td>
<td>304.4</td>
<td>0.689</td>
<td>105.88</td>
<td>3.633</td>
</tr>
<tr>
<td>160.09</td>
<td>305.1</td>
<td>0.689</td>
<td>103.99</td>
<td>3.624</td>
</tr>
<tr>
<td>159.60</td>
<td>304.7</td>
<td>0.689</td>
<td>101.87</td>
<td>3.619</td>
</tr>
<tr>
<td>159.79</td>
<td>303.4</td>
<td>0.692</td>
<td>99.66</td>
<td>3.626</td>
</tr>
<tr>
<td>159.85</td>
<td>303.4</td>
<td>0.688</td>
<td>97.99</td>
<td>3.618</td>
</tr>
</tbody>
</table>

5.4.1.3 Data reduction

The steady state pressure measurements were evaluated in the following way. First, the data was extracted from the different files and converted from a relative to an absolute pressure value. Then the first statistical moments were computed using the definitions from Bendat and Piersol (1992) and are presented in detail in Appendix H. The data was finally sorted and saved into an appropriate format for later visualization using the commercial program Tecplot (Amtec, 2001).

In addition to statistical definitions, the isentropic Mach number was calculated both at the inlet and on the bump surface for each operating point by using the mean values of the stagnation pressure and measured static pressure as follow:

$$M_{\text{iso}} = \sqrt{\frac{2}{\gamma - 1} \left( \left( \frac{P_{\text{inlet}}}{P_{\text{ts}}^{\text{iso}}} \right)^{\frac{\gamma}{\gamma-1}} - 1 \right)} = \sqrt{\frac{2}{\gamma - 1} \left( \left( \frac{P_{\text{inlet}}}{P_{s}^{\text{sc}}} \right)^{\frac{\gamma}{\gamma-1}} - 1 \right)}$$  (5.3)

Finally, all evaluated or computed data were sorted using their respective meshes presented in Figures 5.32 and 5.33, and then saved into Tecplot format files.

5.4.1.4 Uncertainty analysis

According to Swedish standard SMO (1981), any given experimentally measured quantity can be thought to contain two parts: a true value and an error term. The latter can be also divided into a bias, or systematic component, and a random component. By definition, bias errors act in a systematic fashion. Hence, if accurately quantified during
CHAPTER 5. EXPERIMENTAL MODEL

Figure 5.32: Experimental mesh for steady state pressure measurements on 2D test object

Figure 5.33: Experimental mesh for steady state pressure measurements on 3D test object

calibration procedure, such errors can be effectively suppressed.

It should be noted that different standards for uncertainty of measurement estimation exist and are simply based on different approaches of the concept of uncertainty. Although the definition of the different error estimations are not exactly similar, the final estimation of uncertainty is believed to be the same. As an example, the Swedish Standard defines a systematic and random error components as defined above, whereas the international standard BIMP (1997) prefers to define to an estimated and expected uncertainty values.

It is therefore a personal choice that the author made to follow such standard as more commonly admitted. The reader is advice to refer to other standardization guides (ISO, 1993), (BIMP, 1997), or (SMO, 1981) for further details on measurement uncertainty evaluation.

In the following paragraphs are presented the evaluation of pressure measurement uncertainty first related to the PSI system, and then including the overall measurement chain.
5.4. MEASURING TECHNIQUES

5.4.1.4.1 PSI system accuracy

A detailed working description and a fairly complete accuracy analysis related to the steady state pressure measurement system was conducted and is presented in Appendix I.

Finally, the uncertainty on the measurement for the respective pressure scanners used within the PSI 8400 system is the following:

- ESP range: $\pm 7kPa$ \Rightarrow $\Delta P_{abs}(\pm7kPa) = \pm2.94Pa$
- ESP range: $\pm 35kPa$ \Rightarrow $\Delta P_{abs}(\pm35kPa) = \pm14.7Pa$
- ESP range: $\pm 100kPa$ \Rightarrow $\Delta P_{abs}(\pm100kPa) = \pm42Pa$

5.4.1.4.2 Overall pressure accuracy

As shown above, the PSI 8400 system can measure pressures with a rather good accuracy. However, the measured values are all relative to the atmosphere and must therefore be added to the atmospheric pressure value delivered by the digital barometer. The overall pressure measurement accuracy has thus to account, as well, for the barometer accuracy.

As the errors related to both devices are statistically independent, the absolute error can be determined by the method of equal effects as demonstrated in Doebelin (1996):

$$f = f(x, y, \ldots) \Rightarrow \Delta f = \sqrt{\left(\frac{\partial f}{\partial x} \Delta x\right)^2 + \left(\frac{\partial f}{\partial y} \Delta y\right)^2 + \ldots} \quad (5.4)$$

Thus, for the absolute pressure reading, the full uncertainty is:

$$P_{abs} = P_{rel}^{PSI} + P_{rel}^{BARO} \Rightarrow \Delta P_{abs} = \sqrt{\left(\Delta P_{rel}^{PSI}\right)^2 + \left(\Delta P_{rel}^{BARO}\right)^2}$$

- ESP range: $\pm 7kPa$ \Rightarrow $\Delta P_{abs}(\pm7kPa) = \pm11.9Pa$
- ESP range: $\pm 35kPa$ \Rightarrow $\Delta P_{abs}(\pm35kPa) = \pm18.7Pa$
- ESP range: $\pm 100kPa$ \Rightarrow $\Delta P_{abs}(\pm100kPa) = \pm43.5Pa$

5.4.1.4.3 Uncertainty related to other aerodynamic variables

An estimation of the uncertainty value can as well be determined for other aerodynamic variables directly calculated from the steady state pressure measurements, like the isentropic Mach number:

$$M_{iso} = \sqrt{\frac{2}{\gamma - 1} \left[\left(\frac{P_t}{P_s}\right)^{\frac{\gamma+1}{2\gamma-1}} - 1\right]} \quad (5.5)$$

Thus,

$$\Delta M_{iso} = \sqrt{\left(\frac{\partial M_{iso}}{\partial P_t} \Delta P_t\right)^2 + \left(\frac{\partial M_{iso}}{\partial P_s} \Delta P_s\right)^2}$$

with

$$\frac{\partial M_{iso}}{\partial P_t} = \frac{1}{5P_t M_{iso}} \left(\frac{P_t}{P_s}\right)^{-\frac{1}{2}}$$
$$\frac{\partial M_{iso}}{\partial P_s} = \frac{1}{5P_s M_{iso}} \left(\frac{P_t}{P_s}\right)^{-\frac{\gamma+1}{2\gamma-1}}$$

(5.6)
Hence, the uncertainty corresponding to the calculated isentropic Mach number is a function of the Mach number itself, thus of the static and total pressure values (see equation 5.6). Consequently, assuming that both the total pressure in the settling chamber and the static pressure in the test section were measured with the same type of scanner, i.e. scanners with same pressure range, the isentropic Mach number uncertainty can be plotted versus the static pressure, for different total pressure values around the operating point, in Figure 5.34.

\[
\begin{array}{c}
\text{ESP range: } \pm 7\text{kPa} \quad \Rightarrow \quad \Delta M_{\text{iso}} = \pm 0.00018 \\
\text{ESP range: } \pm 35\text{kPa} \quad \Rightarrow \quad \Delta M_{\text{iso}} = \pm 0.00028 \\
\text{ESP range: } \pm 100\text{kPa} \quad \Rightarrow \quad \Delta M_{\text{iso}} = \pm 0.00065 \\
\end{array}
\]
5.4. MEASURING TECHNIQUES

5.4.2 Unsteady pressure measurements

5.4.2.1 Unsteady data acquisition system

The unsteady pressure measurements were performed using the K8000 system, a high frequency data acquisition and storage system developed by Kayser-Threde\(^5\) and featuring a dual bus based architecture. The main chassis of the system is a VMEbus acquisition unit located in a 19inch rack mountable housing. A functional overview of the K800 system is presented in Figure 5.35. The C/MDT200 data acquisition controller is responsible for the address and control signal generation which initiates the acquisition and data transfer from the addressed analog and digital input modules via the High Speed Data (HSD) bus. A maximum sustained sampling rate of 6.4MBytes/s for all channels is possible and thus allow real time data acquisition, monitoring and storage. A dual buffer controller, the C/DM2041 module, thereafter ensure the transfer of data from the HSD bus to a solid state memory, or to any mass storage device like hard drive via the VME bus. Data transfer from the VMEbus interface to the PCIbus interface located in the host computer is performed via the PCI200 PCI-VMEbus coupler.

![Functional overview of the K8000 system](image)

Figure 5.35: Functional overview of the K8000 system

Eight C/MBA101 DC Bridge amplifiers modules are directly connected to the VMEbus and provide highly integrated signal conditioning and data acquisition (Figure 5.36). Each module allows up to four independent signal conditioning with 100kHz bandwidth differential input, programmable input voltage (from \(\pm 5mV\) to \(\pm 5V\)), programmable calibration or running modes, an automatic offset correction, a 14 Bit Analog Digital Converter (ADC), and a low pass filter 48dB/Octave with programmable cut Frequency (Fc).

---

\(^5\)Kayser-Threde, Wolfratshauener strasse 48, 81379 München, Germany
Digitized data is acquired under control of the C/MDT200 Acquisition Controller, which can be programmed for sampling rates from 400 samples/s to the maximum value of 6.4M samples/s real time. As a result, all 32 channels can be sampled at up to 100kHz and stored to disk in real time. Besides, both a higher sampling rate and an oversampling are possible with respect to the 6.4M samples/s limit.

The acquired data is regularly transferred to the disk via a dual buffer module with 2x2MBytes. Each time a buffer has been filled up with acquisition data, the content of this buffer is transferred onto the disk, while the second buffer is filled with acquisition data. The minimum frame size is therefore 2MBytes, which also sets a minimum sampling time, depending on the acquisition rate.

The application software is running under the windows NT operating system installed on a standard PC. The user interface (see Figure 5.37) provides all control functions: amplifiers and acquisition settings, data monitoring, real time graphical display of one selected channel, disk storing, and off line file conversion to ASCII format.

5.4.2.2 Dynamic calibration of capillarity tubes

Introduction and objectives

Basically, a calibration is the establishment of a known relation or transfer function (TF) between the input or driving function and the output or response function. In the present case, the instrumentation holes on both 2D and 3D bumps were foreseen such that the Kulite transducers were located at different distances from the surface depending on the local thickness of the test object. As a result, the pressure waves travelling within the channel and propagating in each capillarity tube is reaching the transducers with different time delays and amplitude attenuations, which have to be established. This subsection is therefore concerned with the estimation of the transfer function of the pressure
5.4. MEASURING TECHNIQUES

(a) Sampling table selection
(b) Signal visualization

Figure 5.37: Unsteady pressure measurement system KT8000 software illustration

fluctuations through each capillarity tube.

Following the recent purchase of the unsteady pressure measurement system, a complete dynamic calibration method was developed and is presented in detailed in Appendix J. The following paragraphs will hence only present a summary of the dynamic calibration procedure. Although the development of such method constitute a whole and consistent part of the present research work, it was decided to develop it in detail in Appendix in order to clarify the overview of the experimental model.

First, a quick overview of the different calibration devices and methodologies used nowadays was performed, followed by a description of the device and analysis methodology chosen for dynamic calibration of the capillarity tubes. Finally, the data acquisition procedure and data reduction is presented, as well as the method used to account for the damping and phase-lag of the pneumatic line during the unsteady pressure measurements.

Dynamic calibration methodology

According to the above literature survey

As a result, it was decided to apply a periodic fluctuating pressure on each pressure tap, directly on the surface of the instrumented test object and measure both the input and response signals as sketched in Figure J.2.

The dynamic calibration apparatus basically consists of a small device, so called calibration head, which contains a small chamber opened at one end and connected to a Kulite transducer on the other end, as shown in Figure J.3. Two holes were drilled on the side of the calibration head in order to introduce and evacuate the pressure fluctuations. One of them is connected to an external source of pressure fluctuation, whereas the second one is simply connected to a long pipe which damps the fluctuations and avoids any reflected pressure waves back in the small chamber. The interface between the tap surface and the chamber’s opening is sealed with an O-ring to prevent any leakage. Besides, the design
was made symmetrical so that the pressure fluctuations on the surface and recorded by the transducer could assumed the same, which was thereafter experimentally confirmed. As a result, the fluctuating pressure within the chamber is measured simultaneously by the reference transducers in the calibration head, and a "test" transducer at the other end of the capillarity tube.

The acquisition procedure basically consisted in the following steps. First, a six points static calibration was performed on all transducers before and after the unsteady measurements in order to minimize the systematic error and check the influence of time.
shift on sensitivity and offset. Secondly, using the methodology and instrumentation introduced in the previous paragraph a fluctuating pressure was applied on the surface of the test object and simultaneously recorded by a reference transducer placed on the pressure tap and a test transducer placed underneath the bump, on the other side of the pneumatic line to be calibrated. This procedure was repeated for each pressure tap on both the 2D and 3D test objects, for different frequencies of perturbations (from 50Hz to 4kHz), and different jet pressures (10, 20, and 30kPa), i.e. different amplitude of perturbations. In total 4500 measurements points were taken to establish the transfer function as a function of the perturbation frequency and amplitude for all pressure taps on the test objects.

Dynamic calibration data reduction

The dynamic calibration data reduction consisted in two parts. The first task was to evaluate the damping and phase-lag for each capillarity tube as discrete functions of the perturbation frequency and amplitude. The main purpose of this part was to create a calibration database, which would thereafter be used to estimate the transfer function on any pressure tap, for any frequency and any amplitude of pressure perturbation.

The second part of the dynamic calibration data reduction consisted in fitting a model function to the dynamic calibration experimental data obtained for discrete frequency values. The objective was to provide a continuous function for the damping and phase-lag so that a value could be interpolated depending on the perturbation frequency and amplitude measured directly during the unsteady pressure measurements.

The practical details of the procedure for both tasks described above are further developed in Appendix J.

Error and accuracy of dynamic calibration

The accuracy estimation of the dynamic calibration is not a simple task as many different measurements and data reduction procedures are involved. A description of the measurement chain as well as a breakdown of precision errors is presented in the "Error and accuracy estimation" section further down.

5.4.2.3 Acquisition method

5.4.2.3.1 Common acquisition setup and procedure

Unsteady pressure measurements setup

The unsteady pressure measurements setup was made similar for both the 2D and 3D test objects in order to optimize the data reduction post treatment. The reference TTL signal from the downstream rotating rod was input on the first channel of the KT8000 system. Seventeen Kulite transducers placed underneath the test objects to measure the unsteady pressure distribution on the surface were then connected to the following channels. Finally, two extra Kulite transducers mounted in pipes were placed on both side walls
(y=0 and 100mm) to measure outlet pressure fluctuations at a mid-channel (z=60mm) and allow signal comparison with the pressure distribution on the surface. The sampling frequency was set to 40kHz with a low pass filter adjusted to 20kHz to avoid any bias effects while performing harmonic analysis. It should be noted that additional numerical low pass filtering is also performed during the post processing to cut resonance frequency effects. The high sampling frequency value was basically chosen to ensure a comfortable description of the fluctuating signal at high frequency of perturbation and increase the accuracy while performing the Fourier analysis and computing the amplitude and phase of higher harmonics. Four different frequencies of perturbation (50, 100, 250, 500Hz) were investigated on both 2D and 3D test objects and the sampling time (or the file size) was adapted depending on the perturbation frequency in order to obtain respectively 300, 600, 750, and 1500 unsteady cycles per measurement.

Common acquisition procedure

As already mentioned, a static calibration of all Kulite transducers was performed systematically before and after each measurement to suppress any possible systematic error in the transducers response. Illustrative results and calibration procedure are presented in the above presented “dynamic calibration” section (see Figures J.5 and J.6). Although small differences were found in the sensitivity and offset values between the prior and post calibrations, the drift of those coefficients was always smaller than the manufacturer specification and were thus included within the provided accuracy value. However, in order to further reduce the systematic error, an averaged value of the sensitivity and offset was calculated for each transducer.

The steady state operating conditions, around which the unsteady measurements should oscillate, were estimated by measuring the change in back pressure between the extreme positions (vertical and horizontal) of the downstream rod and thereafter set the outlet static pressure so that the averaged value would correspond to the desired operating conditions.

5.4.2.3.2 Data acquisition procedure on 2D sliding test object

Operating conditions

Steady state operating points were obtained by monitoring and averaging the flow conditions for extreme positions (vertical or horizontal) of the rotating rod. All operating conditions are presented in Table 5.9 together with expected values and unsteady parameters, which simply consist of an amplitude and frequency of perturbation.

Steady state pressure measurements were simultaneously performed in order to estimate the maximum shock motion range and obtain a comparison between the extreme operating points and steady state pressure measurement presented in Appendix M. Time averaged pressure distribution in the center of the channel are therefore presented in Figure 5.40 for extreme positions of the rotating rod together with steady state pressure distributions corresponding to the closest operating points available.
### 5.4. MEASURING TECHNIQUES

#### Steady State OPs

<table>
<thead>
<tr>
<th>Rod position</th>
<th>Steady State OPs</th>
<th>Unsteady OPs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>Value</td>
<td>Value</td>
</tr>
</tbody>
</table>

#### Unsteady OPs

<table>
<thead>
<tr>
<th>Rod position</th>
<th>Steady State OPs</th>
<th>Unsteady OPs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>Value</td>
<td>Value</td>
</tr>
</tbody>
</table>

### Table 5.9: Operating conditions during unsteady pressure measurements

<table>
<thead>
<tr>
<th>OP</th>
<th>( P_{in} )</th>
<th>( T_{in} )</th>
<th>( T_{out} )</th>
<th>( P_{out} )</th>
<th>( A )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>159.57kPa</td>
<td>303.5K</td>
<td>110.58kPa</td>
<td>108.00kPa</td>
<td>±1.25kPa</td>
</tr>
</tbody>
</table>

### Figure 5.40: Steady state pressure distribution for extreme rod position on unsteady OP No1

**Unsteady measurements procedure**

The measurement procedure on the 2D bump consisted in successively equipping one row of the test object with 17 transducers and traverse the channel with 4.5mm steps, then change the layout of the Kulite transducers and traverse again. As a result, a resolution of 4.5mm in the main stream direction and 5mm in the transversal direction was obtained. Measurements were repeated for all four perturbations frequencies. Besides, for each measurement, two extra transducers placed on both side walls at the outlet of the test section recorded the outlet static pressure in order to quantify the upstream propagating perturbations.

### 5.4.2.3.3 Data acquisition procedure procedure on 3D test object

#### Operating conditions
Similarly to the 2D test object, steady state operating conditions for extreme positions of the rod are presented in Table 5.9 together with the expected conditions and unsteady parameters.

Unsteady measurements procedure

The measurement procedure on the 3D bump simply consisted of inserting the fast response transducers into the instrumented test object following a specific layout and trigger the measurements for different frequency of perturbation as specified in Table 5.9. Two transducers were as well used to measure the back pressure fluctuations. The overall resolution covers the entire surface of the 3D bump with finer measurements underneath the shock position.

5.4.2.4 Unsteady pressure data reduction

General overview of data reduction and post treatment

Following the acquisition procedure described above, unsteady data were stored into binary files for different measurement layouts and different frequencies of perturbations. The unsteady pressure measurement data reduction basically consists of evaluating all data in both the time and frequency domain in order to compare, analyze and understand the information contained in the raw signals.

First, output voltage signals from the transducers placed underneath the test object are read and converted into pressure signals using the static calibration coefficients obtained prior to the measurements. Secondly, for each channel, the TTL signal from the motor is used as a reference to make an ensemble average (EA) of the time serie data (illustration in Figure 5.41(a)). The obtained single unsteady cycle then represents an average of all unsteady cycles. The mathematical formulations introduced here are further described in the following paragraphs to keep this overview as clear as possible. Thirdly, a Discrete Fourier Serie Decomposition (DFSD) is performed on the ensemble averaged signal computed previously and the amplitude and phase angle of the few first harmonics are evaluated. Simultaneously, a Fast Fourier Transform (FFT) is performed on the entire time fluctuating signal to evaluate all frequency components. At this point, the transfer function (TF) throughout each capillarity tube is evaluated depending on the respective amplitude of the fundamental (illustration in Figure 5.41(b) and 5.41(c)). Consequently, both the DFSD components (amplitude and phase of all harmonics) as well as the FFT signal (amplitude only) are corrected using the damping and phase lag values evaluated at the corresponding frequency. The time delay relative to the sequential sampling of the KT8000 system is also accounted here. Finally, each signal is reconstructed and compared to the uncorrected one (illustration in Figure 5.41(d)). As a result, the corrected unsteady pressure distribution and corresponding spectral content over the test object surface are stored in binary data files, Tecplot data files, or animation movies.

Ensemble averaging procedure
5.4. MEASURING TECHNIQUES

(a) Time serie and Ensemble Averaged pressure signals

(b) Evaluation of measured fundamental amplitude among dynamic calibration curves

(c) Interpolation of transfer function (damping and phase-lag) depending on amplitude of fundamental

(d) Corrected and reconstructed signals for different number of harmonics

Figure 5.41: Illustration of data reduction process during unsteady pressure measurements on 2D bump

The ensemble average procedure basically consists of overlaying the entire set of data, for a given channel, into a single reference period, which corresponds to the period of the imposed excitation. The benefit of such procedure is to suppress random signal noise. The acquisition method described previously greatly facilitates the procedure since each pair of periods is referenced by the TTL signal output from the motor of the back pressure perturbation generator and recorded together with the unsteady pressure measurements. Hence, the ensemble averaging is accomplished by calculating the mean of all $n^{th}$ point of each period, as expressed below and illustrated in Figure 5.42:

$$P_n = \frac{N}{M} \sum_{i=0}^{M+N} P_{n+N,i} \quad \text{with} \quad N = \frac{T}{\Delta t} = \frac{F_s}{F_p} \quad (5.7)$$
CHAPTER 5. EXPERIMENTAL MODEL

where: \( \overline{P}_n \) is the mean sample value at the \( n^{th} \) index of the dual period.

\( P_i \) is a data point in the original sample.

\( N \) is the number of sample per pair of unsteady periods.

\( M \) is the total number of samples.

In this investigation, the number of samples per (pair of) period is a function of the sampling frequency and the perturbation frequency and corresponds to 800, 400, 160 and 80 respectively to the perturbation frequency 50Hz, 100Hz, 250Hz, and 500Hz.

\[
\text{Figure 5.42: Illustration of Ensemble Averaging technique}
\]

The spread of the samples around the average signal is statistically analyzed to estimate the 95\% confident interval of the measured unsteady pressure. The reasons for the spread are turbulence, noise, external perturbations from the different parts of the wind tunnel, fluctuations in the rotating speed of the perturbation device...

Spectral analysis: Discrete Fourier Serie Decomposition

According to the Fourier theorem, any physical function that varies periodically with time with a frequency \( f \) can be expressed as an infinite superposition of sinusoidal components with integer multiples of the fundamental frequency.

Practically, a periodic function of \( t \), with period \( T \), can be expressed as the following summation:

\[
f(t) = \frac{a_0}{2} + \sum_{n=1}^{\infty} a_n \cos(n \omega t) + b_n \sin(n \omega t) \quad \text{where} \quad \omega = 2\pi f = \frac{2\pi}{T}
\]

(5.8)
5.4. MEASURING TECHNIQUES

with

\[ a_0 = \frac{1}{T} \int_{-\frac{T}{2}}^{\frac{T}{2}} f(t) dt \]  
\[ a_n = \frac{2}{T} \int_{-\frac{T}{2}}^{\frac{T}{2}} f(t) \cos(n\omega t) dt \]  
\[ b_n = \frac{2}{T} \int_{-\frac{T}{2}}^{\frac{T}{2}} f(t) \sin(n\omega t) dt \]

(5.9) \hspace{1cm} (5.10) \hspace{1cm} (5.11)

Using the orthogonality properties of the Sine and Cosine functions, the phase angle of each harmonic component can as well be calculated compared to the reference signal.

Spectral analysis: Fast Fourier Transfer

The Fourier transform is a generalization of the complex Fourier series in the limit as \( T \to \infty \). The sum is then replaced by an integral and the equations becomes:

\[ F(\omega) = \int_{-\infty}^{+\infty} f(t)e^{j\omega t} dt \]

(5.12)

The fast Fourier transform (FFT) is a discrete Fourier transform algorithm which reduces the number of computations needed for \( N \) points from \( 2^N \) to \( 2N \ln N \), where \( \ln \) is the base-2 logarithm. Although incredibly convenient for their fast computations, FFT algorithms have however a few drawbacks. First of all, the number of samples used to compute the FFT should be an integer multiples to power of two. Any different number of samples would result in a truncated signal and introduce non physical frequency components. Secondly, for a given power spectrum, aliasing (or FFT leakage) is a false translation of power falling in some frequency range \((-f_c, f_c)\) outside the range. Aliasing can, for instance, be caused by discrete sampling below the Nyquist frequency and results in non real frequency components. Aliasing can be reduced by apodization using tapering functions like the Hanning function, but unfortunately, at the expense of broadening the spectral response and thus leaking the energy content. However, a special function based on the Hanning window but only limited to the first and last periods (corresponding to the imposed perturbation frequency) was used to limit FFT leakage and improve the spectral response.

Dynamic calibration influence determination

The correction for dynamic behaviour basically consists of evaluating the damping and phase-lag introduced in the measurements due to the dynamic response of each capillarity tube. As seen in the "Dynamic calibration" section, the transfer function through each pneumatic line is a function of both the frequency and the amplitude of perturbation. As a result, the damping and phase-lag values are evaluated from the database built for each capillarity tube on each test object by interpolation at the measured amplitude of perturbation. The interpolated values are then fitted to a continuous model function in order to calculate the transfer function over the whole frequency spectrum. The procedure
is further described in the section on "dynamic calibration and illustrated in Figure J.9. The correction is thereafter applied by multiplying the measured pressure by the damping coefficient and adding the phase-lag to the measured phase angle for each harmonic in case of DFSD or continuously over the entire spectrum in case of FFT.

**Measurement equipment time-delay correction**

Similarly to the calibration of pneumatic lines, the electronic measuring equipment also should be checked for dynamic behaviour. In the present case, however, both the reference and pressure signals were measured using the same electronic and should theoretically have the same dynamic response. As discussed in the previous section, only the time delay due to the sequential sampling method of the unsteady pressure measurement system have to be accounted here. This delay basically depends on the total number of channels, the sampling frequency, and the relative position of the current channel within the recording frame. This time delay is automatically calculated, converted into a phase-lag, and applied to the measured phase angle for correction.

**Backups and unsteady visualizations**

All measured data, raw and corrected, are finally recorded into binary files for later post processing or visualizations. Especially, an interactive program allows the user to observe the frequency contents of the unsteady pressure distribution at any location on the bump surface by clicking on the time averaged pressure map. Animation movies were also created to display the time evolution of certain quantities like the corrected static pressure, the reconstructed pressure signal with different harmonics, the isentropic Mach number, the pressure fluctuations, etc.

**5.4.2.5 Accuracy - Error analysis**

Whilst the small errors associated with steady flow measurements can be simply defined by the accuracy of the individual instruments and sensors, the overall error introduced by the complex procedure for unsteady pressure measurements is unfortunately far less clear. Therefore, a qualitative, and quantitative when possible, description of the error introduced at each step of the measurement procedure is going to be discussed below starting by the sensor and including the measurement system and procedure, the different calibrations as well as the post treatment methods. An overview of the measurement chain is presented in Figure 5.43 and the respective error estimation is addressed in tables in the following paragraphs.

**Kulite transducers accuracy**

Introduced in the section on "unsteady instrumentation", the accuracy such fast response transducer corresponds to three types of behaviour: the non-linearity, the hysteresis, and the repeatability. The overall accuracy for the XCQ-062 type Kulite transducers was estimated around ±0.141% full scale, that is to say ±240Pa for a pressure
5.4. MEASURING TECHNIQUES

Figure 5.43: Illustration of the unsteady pressure measurement chain

range of 170 kPa.

Analog-Digital Conversion accuracy

Defined as the ratio between the smallest pressure change noticeable by the 16 bits Analog-Digital (AD) card and the maximum pressure acquired (160 kPa), the ADC accuracy is about ±0.003%.

Floating point resolution

Once the AD conversion is performed, data are coded into 32bits reals and the floating point resolution is thus $2^{-32} \approx \pm 10^{-10}$%.

Static calibration

The static calibration procedure, which consists of establishing a linear relationship between transducers output voltage and pressure, is composed of many step as illustrated in Figure 5.43 and detailed in Table 5.10 below.

Pressures are acquired by an accurate external reference transducer and the Kulite transducer under calibration. Output signals are then processed respectively by the portable calibration unit (DPI601) and the unsteady measurement system (KT8000). The data reduction then consists of a linear curve fitting, so called linear regression method, which bring the coefficients of the modelled function. The accuracy of the modelling technique was then calculated by measuring the residuals, that is the difference between the experimental values and the modelled function values. The curve fitting accuracy was
CHAPTER 5. EXPERIMENTAL MODEL

<table>
<thead>
<tr>
<th>I. Reference transducer &amp; Signal Processing Errors</th>
</tr>
</thead>
<tbody>
<tr>
<td>AMOUNT</td>
</tr>
<tr>
<td>±0.1%</td>
</tr>
<tr>
<td>±0.1%</td>
</tr>
<tr>
<td>±0.025%</td>
</tr>
<tr>
<td>±0.006%</td>
</tr>
<tr>
<td>±10⁻⁵%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>II. Kulite transducer &amp; Signal Processing Errors</th>
</tr>
</thead>
<tbody>
<tr>
<td>±0.1%</td>
</tr>
<tr>
<td>±0.1%</td>
</tr>
<tr>
<td>±0.003%</td>
</tr>
<tr>
<td>±10⁻¹⁰%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>III. Data Reduction Errors</th>
</tr>
</thead>
<tbody>
<tr>
<td>±0.002%</td>
</tr>
</tbody>
</table>

Table 5.10: Breakdown of precision errors during static calibration procedure found to be below ±0.002%.

Finally, the coefficients provided by the data reduction process have an accuracy corresponding to the Root-Sum-Square (RSS) of the accuracies presented in Table 5.10:

\[
\Delta a, \Delta b = \pm \sqrt{\sum \text{error}^2} = \pm 0.202\% \quad (5.13)
\]

**Dynamic calibration**

Similarly, the dynamic calibration procedure is illustrated in Figure 5.43 and main error sources are detailed in Table 5.11.

The final accuracy corresponds to the Root-Sum-Square (RSS) of all accuracies:

\[
\Delta a, \Delta b = \pm \sqrt{\sum \text{error}^2} = \pm 0.41\% \quad (5.14)
\]

**Overall unsteady pressure accuracy**

Finally, summing up all different accuracies during the unsteady pressure measurement process, including the static and dynamic accuracies, gives the following value:

\[
\Delta P = \pm \sqrt{\sum \text{error}^2} = \pm 0.48\% \quad (5.15)
\]

5.4.3 Hotfilm measurements

This section presents the hotfilm measurements. It first introduces the principles of such technique and gives an overview of the Constant Temperature Anemometer (CTA) System. It discusses the components of the system and briefly explains its operation and
5.4. MEASURING TECHNIQUES

I. Reference transducer & Signal Processing Errors

<table>
<thead>
<tr>
<th>AMOUNT</th>
<th>TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>±0.1%</td>
<td>Non Linearity &amp; Hysteresis</td>
</tr>
<tr>
<td>±0.1%</td>
<td>Repeatability (including temperature effects)</td>
</tr>
<tr>
<td>±0.003%</td>
<td>ADC (KT8000)</td>
</tr>
<tr>
<td>±10^-10%</td>
<td>floating point resolution</td>
</tr>
</tbody>
</table>

II. Calibrated transducer & Signal Processing Errors

<table>
<thead>
<tr>
<th>AMOUNT</th>
<th>TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>±0.1%</td>
<td>Non Linearity &amp; Hysteresis</td>
</tr>
<tr>
<td>±0.1%</td>
<td>Repeatability (including temperature effects)</td>
</tr>
<tr>
<td>±0.003%</td>
<td>ADC (KT8000)</td>
</tr>
<tr>
<td>±10^-10%</td>
<td>floating point resolution</td>
</tr>
</tbody>
</table>

III. Data Reduction Errors

<table>
<thead>
<tr>
<th>AMOUNT</th>
<th>TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>±0.202%</td>
<td>static calibration</td>
</tr>
<tr>
<td>±0.3%</td>
<td>Curve Fit Error (robust least square method)</td>
</tr>
</tbody>
</table>

Table 5.11: Breakdown of precision errors during dynamic calibration procedure

5.4.3.1 Data acquisition system

System overview

The IFA 300 System, presented in Figure 5.44, is a set of Constant Temperature Anemometers expandable to 16 channels. It provides up to 300kHz frequency response, depending on the sensor used. Each module is designed with a built-in thermocouple circuit for measuring fluid temperature and for making temperature corrections. All operations, including setup, calibration, and data acquisition are software-controlled via a RS-232 interface.

Data Acquisition and A/D Converter Board

The data acquisition and analysis software runs as a stand-alone program under Microsoft Windows©. The software basically selects the channel, measures cable and sensor resistances, sets sensor operate resistance, measures fluid temperature, switches between standby and run modes, and sets signal conditioner offset and gain.

The A/D converter board plugs directly into a computer. The software controls the A/D converter, selecting the sampling rate and the sample block size and performs all the calibration, acquisition, and analysis functions.

System description
A CTA is a bridge and amplifier circuit that controls a tiny wire or film sensor at constant temperature. As a fluid flow passes over the heated sensor, the amplifier senses the bridge off-balance and adjusts the voltage to the top of the bridge, keeping the bridge in balance. The voltage on top of the bridge can then be related to the velocity of the flow. The bridge voltage is sensitive to temperature as well as velocity and so the built-in thermocouple circuit can be attached to a thermocouple that can measure the fluid temperature. This temperature reading can then be used by the software to correct the results to minimize the effect of temperature.

Each IFA 300 unit contains one microprocessor which controls all functions, settings of the anemometer, and signal conditioner via an address and data bus. An RS232 interface is used to send commands from the computer to the microprocessor.

Each channel of anemometry contains a single bridge circuit and signal conditioner. The bridge circuit includes the Smartune technology (TSI, 1997) that automatically optimizes the frequency response and prevents oscillations which may damage the sensor. Therefore, the bridge does not require tuning for frequency response regardless of the type of sensor used or the length of the cable. Smartune constantly monitors the bridge voltage and feeds a signal back to the amplifier circuit, maintaining the frequency response based on the operating temperature and sensor type.

The signal conditioners (Figure 5.45) in the IFA 300 provide settings for filtering and increasing the bridge voltage gain to use the entire ±5V signal range. High-pass filters

Figure 5.44: Overview of the IFA300 Constant Temperature Anemometer System
5.4. MEASURING TECHNIQUES

available are 0.1Hz, 1Hz, and 10Hz. These filters are used when only velocity fluctuation measurements are needed since mean voltage information and thus actual velocity is removed from the signal. Offset settings available are 0 to 10V in 10mV steps. Offset and gain can be used to utilize entire ±5V signal range. Offset must be used when actual bridge voltage is greater than 5 volts. Gains available range from 1 to 1000. Low-pass filters allow the removal of high frequency signals which are out of the range of interest and to eliminate aliasing. Thirteen low-pass filter settings are available from 10Hz to 1MHz.

5.4.3.2 Hotfilm measurements principle

The principle involved in hotwire and hotfilm measurements is based on the relationship between the rate of heat transfer, through forced convection, of the heated sensor in the colder fluid flow and the electric power needed to keep the sensor at the same temperature. In case of hotfilm measurements however, the output bridge voltage can directly be correlated to the wall shear stress as commonly seen in literature (Hanratty and Campbell, 1983; Bellhouse and Schultz, 1966) in the form:

\[
\frac{I^2R}{\Delta T} = C_1 (\rho \tau_w)^{\frac{1}{2}} + C_2 \tag{5.16}
\]

where \( I \) is the current, \( R \) the resistance of the sensor, \( \Delta T \) is the temperature difference in relation to the fluid, \( \rho \) is the air density, \( \tau_w \) is the wall shear stress, and \( C_1 \) and \( C_2 \) are constants depending on the geometry of the sensor, the flow properties and the heat lost to the substrate. Theoretically, those constants have to be determined by a complicated calibration procedure (Hanratty and Campbell, 1983; Bellhouse and Schultz, 1966; Pope, 1972). However, several publications (Pucher and Göhl, 1987; Hodson, 1984) have demonstrated that the boundary layer condition can be recognized simply by signal comparisons. In the present investigation, a quantitative determination of the wall shear stress is not required but the heat transfer coefficient, proportional to the wall shear stress, can be easily estimated from the output bridge voltage according to a method originally proposed by Wolff and Fottner (2000) and presented below.
The results of thin film measurements are usually evaluated separately according to the DC and AC part of the signal. The continuous voltage, $E$, corresponds to the time averaged heat transfer, whereas the AC voltage, $e$, describes the fluctuating part of the heat transfer. In order to isolate and only consider the part of the heat transfer which is actually dissipated to the fluid by convection, a zero flow measurement has to be performed additionally. Indeed, a simple heat rate balance (see equation 5.18) shows the contribution of conductive, radiative and inertial heat flux terms, which can be eliminated by similar measurements conducted under zero flow conditions at the same overheat ratio.

Heat rate balance

\[
\dot{d}Q_e = \dot{d}Q_{fc} + \dot{d}Q_c + \dot{d}Q_r + \dot{d}Q_s \quad (5.17)
\]

Zero flow heat rate balance

\[
\dot{d}Q^0_e = \dot{d}Q^0_{fc} + \dot{d}Q^0_c + \dot{d}Q^0_r + \dot{d}Q^0_s \quad (5.18)
\]

Therefore, the forced convective heat flux is proportional to:

\[
\dot{d}Q_{fc} = \dot{d}Q_c - \dot{d}Q^0_c \quad (5.19)
\]

Furthermore, the heat generation rate by an electric current of the hotfilm is given by:

\[
\dot{d}Q_e = I^2 R_w = \frac{E^2}{R_w} \quad (5.20)
\]

where $E_w$ is the voltage drop over the gauge, illustrated in Figure 5.46(b), and $R_w$ the resistance of the heated sensor under working conditions.
5.4. MEASURING TECHNIQUES

The voltage drop, $E_w$, over the gauge can be determined by Kirchhoff's law for the electric cycle of the gauge arm:

$$E_w = \frac{R_w}{R_1 + R_{cl1} + R_{cl2} + R_w} E \quad (5.21)$$

The resistance, $R_w$, of the heated sensor can be determined considering the bridge setup and the temperature sensitivity of the gauge resistance $s$.

$$s = \frac{(R_w - R_c)}{(T_w - T_c)} \implies R_w = s \Delta T + R_c = s \Delta T + (R_{gauge} - R_{cl1} - R_{cl2}) \quad (5.22)$$

where $R_c$ is the resistance of the sensor under zero flow condition, and $R_{cl1}$ and $R_{cl2}$ are the resistance of the cables between the sensor and the CTA, including the hotfilm leads, electric cables and multiplexer as illustrated in Figure 5.46(b).

Practically, the temperature sensitivity of the gauge resistance was measured around 0.022 $\Omega /K$ for a couple of sensors and assumed constant for all sensors. The cold resistance of the sensor was measured at the total temperature value (303 $K$) of the flow around 6 to 8 $\Omega$ depending on the sensor.

Now, considering the heat transfer law and the surface area of a typical hotfilm sensor given by the manufacturer, $A = 0.1477 mm^2$, the heat transfer coefficient can finally be calculated:

$$\dot{Q}_e = k A (T_w - T_c) \implies k = \frac{1}{A \Delta T} \left[ \frac{E^2_w}{R_w} - \frac{(E_0^0)^2}{R_0} \right] \quad (5.23)$$

$$\quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \text{(5.24)}$\n
Similarly, the fluctuating part of the heat coefficient is evaluated as follow:

$$k' = \frac{1}{A \Delta T \frac{e_w^2}{R_w}} \quad (5.25)$$

5.4.3.3 Measurement procedure

Acquisition procedure

Hotfilm measurements were performed on the 2D test object in the middle of the channel for different operating conditions as listed in Table 5.12. The acquisition procedure first consisted in measuring the "cold" resistance of each sensor as well as the cable resistance of each gauge independently. The operating resistance determining the temperature at which the sensors should be heated up was then evaluated by using an overheat ratio of 1.3.

A zero flow condition measurement was conducted in order to eliminate the contribution of other heat flux terms in the overall heat rate balance and keep only the part related to forced convection as explained in the previous paragraph.

A preliminary test was systematically conducted for each operating point in order to optimize the amplifiers and offset values for each channel of the CTA. The measurements
CHAPTER 5. EXPERIMENTAL MODEL

Data reduction

Bridge output voltage data were processed as described in the previous paragraph and the time averaged heat transfer coefficient was calculated for each sensor over the 2D test object. The Root Mean Square of the voltage fluctuations was also post treated to calculate the heat transfer coefficient fluctuations.

5.4.3.4 Accuracy - Error analysis

The assessment of the hotfilm measurements accuracy is extremely difficult and almost meaningless considering the fact that no calibration was performed. Indeed, a qualitative comparison of signals was preferred to a quantitative determination of the wall shear stress. As a result, a discussion over the different sources of error and their influence on the signal behaviour is presented below.

The main source of error concerns the determination of the sensor cold resistance. Indeed, a slight difference to the real value has a direct consequence on the operating temperature. As a result, the sensor is not heated at the same overheat ratio as the others and a too high current is delivered by the CTA with the risk to burn the sensor. The chance to overestimate the sensor’s resistance is real when using a resistivity meter with prongs directly applied on the sensor. No accurate method such as short cutting the sensor in hotfilm measurements has been be developed at this moment. Besides, some friction over the copper leads of the gauges occurred while sliding the 2D test object and resulted in changing the resistance of the cable during the measurement. This effect was not taken into account and corrected during the measurement campaign and a large scatter of the results seem to have occurred. As a result, the overheat ratio cannot be assumed the same for all sensors.

<table>
<thead>
<tr>
<th>$P_{\text{inlet}}$ [kPa]</th>
<th>$T_{\text{inlet}}$ [K]</th>
<th>$P_{\text{outlet}}$ [kPa]</th>
<th>$M_{\text{inlet}}$ [kg/s]</th>
<th>$Q_{m}$ [kg/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>159.93</td>
<td>303.9</td>
<td>116.18</td>
<td>0.642</td>
<td>3.539</td>
</tr>
<tr>
<td>160.15</td>
<td>303.6</td>
<td>114.02</td>
<td>0.658</td>
<td>3.581</td>
</tr>
<tr>
<td>160.25</td>
<td>304.6</td>
<td>112.14</td>
<td>0.670</td>
<td>3.603</td>
</tr>
<tr>
<td>159.72</td>
<td>303.5</td>
<td>110.03</td>
<td>0.677</td>
<td>3.631</td>
</tr>
<tr>
<td>159.75</td>
<td>303.7</td>
<td>107.98</td>
<td>0.684</td>
<td>3.654</td>
</tr>
<tr>
<td>160.03</td>
<td>303.4</td>
<td>106.08</td>
<td>0.688</td>
<td>3.668</td>
</tr>
<tr>
<td>160.02</td>
<td>302.9</td>
<td>103.95</td>
<td>0.698</td>
<td>3.669</td>
</tr>
<tr>
<td>160.10</td>
<td>303.6</td>
<td>102.11</td>
<td>0.701</td>
<td>3.668</td>
</tr>
<tr>
<td>159.90</td>
<td>303.6</td>
<td>100.11</td>
<td>0.701</td>
<td>3.668</td>
</tr>
<tr>
<td>160.01</td>
<td>303.6</td>
<td>97.86</td>
<td>0.699</td>
<td>3.669</td>
</tr>
</tbody>
</table>

Table 5.12: Operating conditions for hotfilm measurements on 2Dbump

were finally performed at a sampling frequency of 2kHz during a sampling period of 128 seconds.
Another uncertainty in the post treatment concerns the assumption that the temperature resistivity coefficient of the sensor is similar for all sensors. Indeed, although the material is strictly similar, some small manufacturing inaccuracies might slightly change the resistance as a function of the temperature, which has a direct influence on the overheat ratio and thus the heat transfer coefficient calculation.

Besides, the hotfilm sensor array was operated without evaluating the thermal interaction among sensors. Haselbach and Nitsche (1996) demonstrated the strong effect of an increasing thermal boundary layer thickness on the convective and conductive heat losses of each single sensor within an array due to an increasing number of upstream sensor being operated. They showed that the possible measurement error in terms of \( \Delta \tau_{w, all} \) increases asymptotically with the number of sensor operated upstream of the sensor considered, but also decreases asymptotically with the distance between the sensors. However, considering the low overheat ratio chosen and the distance between each sensor being operated (15mm), the influence of thermal interaction was neglected here.

Finally, considering the results presented in Appendix M and especially the difficulty to accurately detect any flow structure such as a shock or a separation, unsteady measurements were not performed and shall require a better understanding of the hotfilm signals.

5.4.4 Conventional Schlieren flow visualization

5.5 Visualization system

Principle of shock visualization method

The conventional Schlieren method is based, as introduced earlier, on the fact that the speed of light, and consequently the index of refraction, varies with the density of the medium it is passing through. From the basic laws of optics, a changing index of refraction has two effects on a light ray. First, it leads to a rotation of the wave fronts. Secondly, it introduces a phase shift between the different rays. The Schlieren method takes advantage of the first property to visualize regions within the flow field with non uniform density distribution.

A detailed description of the conventional Schlieren technique working principle and optical setup is presented in Appendix K.

Image acquisition system

Both the steady and unsteady Schlieren visualizations were recorded using the MotionScope PCI 8000S CCD camera. This high speed digital imaging system can record a sequence of digital black and white images of an event at a speed of 60 to 8000 frames per second. The system also provides an electronic shutter control of image exposure that allows the reduction of each frame exposure’s time in order to eliminate image blurring due to motion. Finally, the high speed CCD digital camera features the possibility to display the state of two TTL signals directly on the recorded pictures. This feature was
used to locate the reference of each perturbation cycle by monitoring the output TTL signal from the pressure perturbation generator.

### 5.5.0.1 Image acquisition methods

Whereas the optical set-up described up to now is similar for both steady state and unsteady visualizations, the Schlieren pictures were acquired in two different ways since the objectives, acquisition methods and data reduction differ from one case to another.

#### Steady state acquisition method

In steady state visualizations, the objectives are basically to establish statistical moments of the shock wave position for different operating conditions in order to characterize both the mean position and the scatter of the results, that is to say the standard deviation and the confidence interval in which the shock can be though to be found at, say, 95%. Consequently, a certain number of samples (or frames) is needed in order to, first be statistically independent, that is to say reach a Gaussian density probability, and secondly "capture" the lowest frequencies which compose the shock motion.

Practically, the CCD camera was set to record 5,000 frames at the lowest frequency, 50Hz, with a shutter speed of 20x, i.e an exposure time of 1ms. The measuring time was about 120 seconds to be consistent with the pressure measurements described earlier. Thereafter, the experiments basically consisted in setting up the desired operating point and then launch the acquisition. Table 5.13 gives the different experimental operation conditions at which the visualizations were recorded.

<table>
<thead>
<tr>
<th>$P_{\text{inlet}}$ [kPa]</th>
<th>$T_{\text{inlet}}$ [K]</th>
<th>$P_{\text{outlet}}$ [kPa]</th>
<th>$M_{\text{ISO}}$ [-]</th>
<th>$Q_{m}$ [kg/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>160.13</td>
<td>304.4</td>
<td>114.06</td>
<td>0.660</td>
<td>3.64</td>
</tr>
<tr>
<td>160.09</td>
<td>304.4</td>
<td>112.07</td>
<td>0.673</td>
<td>3.68</td>
</tr>
<tr>
<td>159.89</td>
<td>304.4</td>
<td>110.11</td>
<td>0.682</td>
<td>3.69</td>
</tr>
<tr>
<td>159.94</td>
<td>304.5</td>
<td>107.83</td>
<td>0.690</td>
<td>3.72</td>
</tr>
<tr>
<td>160.08</td>
<td>304.5</td>
<td>105.94</td>
<td>0.693</td>
<td>3.73</td>
</tr>
<tr>
<td>159.89</td>
<td>304.4</td>
<td>104.01</td>
<td>0.694</td>
<td>3.73</td>
</tr>
<tr>
<td>160.24</td>
<td>304.4</td>
<td>102.01</td>
<td>0.694</td>
<td>3.74</td>
</tr>
<tr>
<td>160.05</td>
<td>304.4</td>
<td>99.99</td>
<td>0.694</td>
<td>3.73</td>
</tr>
<tr>
<td>160.20</td>
<td>304.4</td>
<td>98.17</td>
<td>0.694</td>
<td>3.73</td>
</tr>
<tr>
<td>160.34</td>
<td>304.4</td>
<td>96.06</td>
<td>0.694</td>
<td>3.74</td>
</tr>
</tbody>
</table>

Table 5.13: Operating conditions for steady state Schlieren visualizations

#### Unsteady acquisition method

Unsteady visualizations have a different objective since the oscillating motion of the shock wave is now of interest. Consequently the sampling frame rate is much higher than
### 5.5. Visualization System

#### Steady State OPs

<table>
<thead>
<tr>
<th>OP</th>
<th>$P_{in}$ [kPa]</th>
<th>$T_{in}$ [K]</th>
<th>$P_{out}$ [kPa]</th>
<th>Expected values</th>
<th>Frequency &amp; Amplitude of perturbations</th>
</tr>
</thead>
<tbody>
<tr>
<td>OP1</td>
<td>159.75</td>
<td>303.7</td>
<td>103.48</td>
<td>159</td>
<td>50, 100, 250, 500 Hz, $A=\pm2.34$ kPa</td>
</tr>
<tr>
<td>OP2</td>
<td>160.25</td>
<td>303.7</td>
<td>108.16</td>
<td>160</td>
<td>50, 100, 250, 500 Hz, $A=\pm1.24$ kPa</td>
</tr>
<tr>
<td>OP3</td>
<td>160.07</td>
<td>303.3</td>
<td>106.04</td>
<td>160</td>
<td>50, 100, 250, 500 Hz, $A=\pm1.49$ kPa</td>
</tr>
</tbody>
</table>

Table 5.14: Operating conditions and sampling parameters for unsteady Schlieren visualizations

in the steady state case, and basically depends on the perturbation frequency, i.e. the frequency at which the shock is excited. At least 20 samples, i.e. frames, are needed to satisfactorily describe a perturbation cycle and calculate the five first harmonics of the perturbation frequency by Fourier decomposition. The sampling frame rate was set to 1, 2, 4, and 8 kHz for the perturbation frequencies 50, 100, 250, and 500 Hz respectively in order to obtain around 20 samples per perturbation cycle. The sampling time was set up, through the number of frames, so that at least a few hundred cycles could be recorded in order to get a correct ensemble averaging. Table 5.14 gives all experimental operation conditions and recording parameters for the unsteady Schlieren visualizations. Whatever the selection of frame rate was, the shutter speed was set to 20x, therefore corresponding to an exposure time of 50, 25, 12.5, and 6.125 μs for the frame rate 1, 2, 4, and 8 kHz respectively.

As mentioned in the previous subsection, the visualizations were manually triggered using the hand held trigger button of the CCD camera, and the TTL output signal from the optical encoder of the perturbation generator device was connected to the data input of the camera. As a result, each time the motor has completed a revolution, the marker appearing in each frame turns either black or white, basically signaling the start of a new perturbation cycle. This marker will be later used during post processing to sort the pictures and make an ensemble average of the unsteady shock position.

#### 5.5.0.2 Image processing and data reduction

The image post processing and data reduction were performed using a set of tools developed under Matlab to enhance the quality of the pictures, process the images to extract quantitative values of an event (i.e. shock wave position), and post treat the data to gather information about a physical phenomenon, using statistical or Fourier decomposition methods.

In the following paragraphs are discussed both the objectives of the post treatment and the tools developed in order to achieve them.
5.5.0.2.1 Steady state post processing

Objectives of steady state data reduction

As mentioned in the previous subsection, the objectives in steady state visualizations are basically to characterize a non deterministic signal, typically the steady state shock position, from a time series, using statistical or Fourier decomposition methods.

The first task will therefore be to establish the statistical moments of the shock wave position for all operating conditions in order to determine the mean position, the standard deviation and the 95% confidence interval of the shock wave position.

The second step would be to determine the content of a Fourier transform of the "quasi-steady" shock motion. Indeed, natural disturbances and noise emission surely have an influence on the shock position, and a correlation between the present visualization technique and the high frequency pressure measurements would certainly be of a great interest towards the understanding of the physical phenomena occurring in the test section facility.

Post treatment tools

A few post treatment tools were developed using Matlab and its image processing toolbox to perform the following functions:

- Enhance the quality of each acquired picture.
- Determine the instantaneous shock position as a function of the channel height (vertical direction) by detecting the sudden change of light intensity.
- Calculate the first central moments to establish the mean shock position, the 95% confidence interval as well as the extreme shock position.
- Plot and save the above statistical information on disk.
- Calculate, for a given coordinate in the vertical direction, the frequency content of the "quasi-steady state" shock position.
- Plot and save the above calculated FFT information on disk.

Presentation of the data

Once post treated, the Schlieren pictures have been modified in order to include statistical information from the data reduction process. As a result, steady state Schlieren visualization pictures are presented as illustrated in Figure 5.47. The scale in the bottom enables the determination of the shock position, whereas the five lines on the picture represent the mean shock position, the standard deviation of the shock fluctuations and the most extreme shock positions.
5.5. VISUALIZATION SYSTEM

5.5.0.2.2 Unsteady post processing

Objectives of steady state data reduction

The objectives in high speed Schlieren visualizations basically consist of determining the unsteady motion of the shock throughout the height of the channel. The latter is to be analyzed with the aim at detecting regions of shock motion amplification and their mechanisms, as well as the overall phase lag of the shock wave. Being of a particular interest for aeroelasticity study, the relative phase lag between different positions in the height of the channel should also be checked up. Naturally, the frequency content of the time varying shock motion is of high important and shall be looked at by calculating both the FFT and the DFT of the time varying shock motion.

Post treatment tools

Similarly to the steady state post treatment, a few post treatment tools were developed to perform the following functions:

- Enhance the quality of each acquired picture.
- Determine the instantaneous shock position and create a time serie of the shock position for a given set of coordinates.
- Calculate on one hand the FFT and evaluate the frequency content of the time varying shock position.
- Plot and save to disk the frequency content determined by FFT.
• Detect, on the other hand, whether the marker corresponding to the TTL pulse level output from the motor, is black or white in order to determine the relative shock position during the unsteady cycle.

• Decompose the complete time serie into a single cycle by using the "TTL marker" and making an ensemble average over the entire set of data i.e time fluctuating shock position.

• Compute the Discrete Fourier Serie Decomposition and evaluate the amplitude and phase for the few first harmonics.

• Plot and save to disk the above calculated information.

• Repeat the process for different coordinates in the vertical direction.

Presentation of the data

The different steps of the post treatment procedure are presented in Figure 5.48. On the left hand side (Figure 5.48(a)) are shown the time fluctuating shock position as it was detected by the post treatment tools, the ensemble averaged signal corresponding to two perturbation cycles, and the amplitude and phase values of the first five harmonics of a Fourier decomposition of the shock motion. On the right hand side (Figure 5.48(b)) is the amplitude of the complete Fourier transform of the shock motion for different location in the channel height.

5.5.0.3 Error and accuracy

To determine the level of uncertainty of a visualization technique is not as straightforward as for a pressure measurement system where a quantitative value can be established to characterize the uncertainty of the measurements. As a result, the different sources of error and the relevant parameters to the accuracy of the visualization technique are discussed rather presented and discussed in Appendix K whereas the most important concepts and quantitatively measurable accuracy are presented below.

Aerodynamical resolution

Rather than a quantitative resolution, the aerodynamical resolution is more a physical consideration to be aware of while post processing and analyzing the pictures. Indeed, the bright and dark areas revealed by Schlieren visualizations actually correspond to density gradients integrated through a certain width, including boundary layers on the side walls and possible three-dimensional shock configuration. In order to overcome this problem, the focusing Schlieren method should be recommended as future work. Although it requires a much more sensitive set-up than the conventional Schlieren method, it presents several advantages among which the possibility to focus over a region with a rather small value of depth of focus, thus allowing three-dimensional visualizations. Further details about this method can be found in Weinstein (1991).
5.5. VISUALIZATION SYSTEM

(a) Ensemble Averaging of Unsteady Schlieren visualization
(b) Amplitude and phase angle of shock motion throughout channel height

Figure 5.48: Illustration of unsteady Schlieren post treatment

CCD camera Resolution

The CCD camera resolution directly contribute to the geometrical resolution and can be quantitatively established. Basically, a Schlieren window with 150mm width and 130mm height was resolved with a 480x420 pixels image giving 3.2 pixels per mm (a geometrical resolution of 0.31mm per pixel). Any other frame rate thereafter uses the same geometrical resolution of 0.31mm per pixel but a different frame resolution like for instance in unsteady visualizations where 8000 frames per second only permit Schlieren window of 160x30 pixels.

Just as Schlieren technique integrates density gradients in space, through the channel width, it also integrates in time the unsteady behavior of the density field. However, a fairly high value of the electronic shutter speed was used both in steady state and unsteady visualizations and provided a exposure frequency 20 times higher than the sampling frequency as mentioned in the acquisition method section.

5.5.1 Oil Paint flow visualization

Surface visualizations are often used to gather information about the flow structures within the boundary layer. The oil paint flow visualization technique is especially suited
to view streamlines directly on the surface and identify flow structures such as separation and reattachment lines, reverse flows, vortices and even the boundary layer thickening.

### 5.5.1.1 Streamlines visualization method

Streamlines visualizations are usually performed by gluing small tufts or spraying a thin layer of oil-paint mix directly onto the surface before the experiment. The problem is that the oil-paint mix tends to disappear before the regime can stabilize or even before the operating conditions can be reached. The solution applied here is to directly inject, through small holes at the inlet of the bumps, a mix of oil and white paint at the inlet of the test section once the operating conditions have been reached and the regime is stable. A quite good spread of the paint could be achieved by having individual injection devices (plastic needle) and evenly spreading the injection holes through the channel’s width. The main advantages of this technique are, as mentioned, to allow the regime to stabilize after setting up the correct operating conditions, which is a long and delicate procedure, and control independently the amount of paint injected into each hole in order to obtain better results while minimizing the interaction between the paint and the flow field.

### 5.5.1.2 Experimental setup and procedure

The experimental setup simply consist of connecting a set of plastic needles, used as oil-paint mix tanks, to the test objects using some flexible pipes as illustrated in Figure 5.49(a). The position marks were directly drawn onto the test objects (see Figure 5.49(b)) in order to locate the different flow structures on the coordinate axis. Finally, a digital video camera was used to record the flow visualization through the upper plexiglas window as seen in Figure 5.49(c).

![Figure 5.49: Experimental setup for oil visualizations](image-url)
The experimental procedure thereafter consist in setting up the correct operating points and slowly inject the paint through the pressure tap holes. The different operating conditions were also recorded using the steady state pressure measurement system, and are presented in tables 5.15 and 5.16 for the 2D and 3D test objects respectively.

<table>
<thead>
<tr>
<th>$P_{\text{inlet}}$ [kPa]</th>
<th>$T_{\text{inlet}}$ [K]</th>
<th>$T_{\text{s}}$ [K]</th>
<th>$P_{\text{outlet}}$ [kPa]</th>
<th>$M_{\text{inlet}}$ [−]</th>
<th>$Q_{\text{m}}$ [kg/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>160.05</td>
<td>303.4</td>
<td>127.95</td>
<td>0.536</td>
<td>3.176</td>
<td></td>
</tr>
<tr>
<td>159.92</td>
<td>303.4</td>
<td>125.85</td>
<td>0.557</td>
<td>3.239</td>
<td></td>
</tr>
<tr>
<td>160.03</td>
<td>303.4</td>
<td>123.91</td>
<td>0.576</td>
<td>3.312</td>
<td></td>
</tr>
<tr>
<td>160.16</td>
<td>303.5</td>
<td>122.18</td>
<td>0.592</td>
<td>3.368</td>
<td></td>
</tr>
<tr>
<td>159.88</td>
<td>303.4</td>
<td>119.94</td>
<td>0.608</td>
<td>3.428</td>
<td></td>
</tr>
<tr>
<td>159.86</td>
<td>303.5</td>
<td>117.85</td>
<td>0.624</td>
<td>3.482</td>
<td></td>
</tr>
<tr>
<td>159.68</td>
<td>303.3</td>
<td>115.87</td>
<td>0.638</td>
<td>3.520</td>
<td></td>
</tr>
<tr>
<td>160.30</td>
<td>303.4</td>
<td>113.91</td>
<td>0.654</td>
<td>3.582</td>
<td></td>
</tr>
<tr>
<td>159.72</td>
<td>303.4</td>
<td>111.96</td>
<td>0.661</td>
<td>3.589</td>
<td></td>
</tr>
<tr>
<td>159.85</td>
<td>303.3</td>
<td>109.90</td>
<td>0.670</td>
<td>3.617</td>
<td></td>
</tr>
<tr>
<td>159.82</td>
<td>303.5</td>
<td>107.84</td>
<td>0.677</td>
<td>3.633</td>
<td></td>
</tr>
<tr>
<td>159.72</td>
<td>303.4</td>
<td>105.83</td>
<td>0.680</td>
<td>3.638</td>
<td></td>
</tr>
<tr>
<td>159.68</td>
<td>303.5</td>
<td>103.72</td>
<td>0.680</td>
<td>3.638</td>
<td></td>
</tr>
<tr>
<td>159.77</td>
<td>303.5</td>
<td>101.98</td>
<td>0.680</td>
<td>3.640</td>
<td></td>
</tr>
<tr>
<td>159.77</td>
<td>303.3</td>
<td>100.11</td>
<td>0.680</td>
<td>3.640</td>
<td></td>
</tr>
<tr>
<td>160.03</td>
<td>303.4</td>
<td>98.08</td>
<td>0.680</td>
<td>3.646</td>
<td></td>
</tr>
</tbody>
</table>

Table 5.15: Operating conditions for oil visualizations on 2Dbump

<table>
<thead>
<tr>
<th>$P_{\text{inlet}}$ [kPa]</th>
<th>$T_{\text{inlet}}$ [K]</th>
<th>$T_{\text{s}}$ [K]</th>
<th>$P_{\text{outlet}}$ [kPa]</th>
<th>$M_{\text{inlet}}$ [−]</th>
<th>$Q_{\text{m}}$ [kg/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>160.05</td>
<td>303.4</td>
<td>127.95</td>
<td>0.536</td>
<td>3.176</td>
<td></td>
</tr>
<tr>
<td>159.92</td>
<td>303.4</td>
<td>125.85</td>
<td>0.557</td>
<td>3.239</td>
<td></td>
</tr>
<tr>
<td>160.03</td>
<td>303.4</td>
<td>123.91</td>
<td>0.576</td>
<td>3.312</td>
<td></td>
</tr>
<tr>
<td>160.16</td>
<td>303.5</td>
<td>122.18</td>
<td>0.592</td>
<td>3.368</td>
<td></td>
</tr>
<tr>
<td>159.88</td>
<td>303.4</td>
<td>119.94</td>
<td>0.608</td>
<td>3.428</td>
<td></td>
</tr>
<tr>
<td>159.86</td>
<td>303.5</td>
<td>117.85</td>
<td>0.624</td>
<td>3.482</td>
<td></td>
</tr>
<tr>
<td>159.68</td>
<td>303.3</td>
<td>115.87</td>
<td>0.638</td>
<td>3.520</td>
<td></td>
</tr>
<tr>
<td>160.30</td>
<td>303.4</td>
<td>113.91</td>
<td>0.654</td>
<td>3.582</td>
<td></td>
</tr>
<tr>
<td>159.72</td>
<td>303.4</td>
<td>111.96</td>
<td>0.661</td>
<td>3.589</td>
<td></td>
</tr>
<tr>
<td>159.85</td>
<td>303.3</td>
<td>109.90</td>
<td>0.670</td>
<td>3.617</td>
<td></td>
</tr>
<tr>
<td>159.82</td>
<td>303.5</td>
<td>107.84</td>
<td>0.677</td>
<td>3.633</td>
<td></td>
</tr>
<tr>
<td>159.72</td>
<td>303.4</td>
<td>105.83</td>
<td>0.680</td>
<td>3.638</td>
<td></td>
</tr>
<tr>
<td>159.68</td>
<td>303.5</td>
<td>103.72</td>
<td>0.680</td>
<td>3.638</td>
<td></td>
</tr>
<tr>
<td>159.77</td>
<td>303.5</td>
<td>101.98</td>
<td>0.680</td>
<td>3.640</td>
<td></td>
</tr>
<tr>
<td>159.77</td>
<td>303.3</td>
<td>100.11</td>
<td>0.680</td>
<td>3.640</td>
<td></td>
</tr>
<tr>
<td>160.03</td>
<td>303.4</td>
<td>98.08</td>
<td>0.680</td>
<td>3.646</td>
<td></td>
</tr>
</tbody>
</table>

Table 5.16: Operating conditions for oil visualizations on 3Dbump

5.5.1.3 Error and accuracy

Basically, the only but not the least error possible during oil paint visualizations is the interaction between the flow and the paint. Ideally, the layer of paint on the surface should be as thin as possible. However, in certain region of the flow field, the paint accumulates due to strong flow decelerations or reverse flows and thus interact or even change the flow pattern. It should also be remembered that the streamlines observed on the surface correspond to flow structures within the viscous sub-layer of the boundary layer, and should not be confused with outer flow structures.

5.6 Conclusion

This chapter was dedicated to the description of the overall experimental apparatus, including the air supply facility, the test objects design and instrumentation, as well as the different measuring techniques used in the present study.
CHAPTER 5. EXPERIMENTAL MODEL

The test section of the wind tunnel was redesigned in order to meet the requirements for the SBLI investigation. Two and three-dimensional test objects were also designed and instrumented with pressure taps and hotfilm sensors. Several measuring techniques were presented by introducing systematically the operating system and discussing the acquisition method, data reduction, as well as estimations of the measurement accuracy and errors. As a result, steady state and unsteady pressure measurements were extensively described, as well as thermal anemometry, high speed Schlieren and oil-paint visualizations.
Chapter 6

Steady State Results

6.1 Introduction

The present chapter aims at describing the steady state flow field within the two-dimensional (2D) and three-dimensional (3D) nozzles presented in the "Experimental model" part of this thesis. Both experimental and numerical steady state investigations were performed over the two geometrical configurations, for several different operating conditions, using the Computational Fluid Dynamic (CFD) tool and different measuring techniques described in chapters 4 and 5 respectively.

The main objective of the steady state campaign is to obtain a clear and accurate description of the flow field structures over a wide range of operating points, from high subsonic to transonic and fully choked nozzle flows. The need to establish a complete characterization of the basic flow structures like shock wave location towards the operating point, namely the outlet static pressure value, is typically of interest in the present chapter. The underlying goal is to build a database and framework in order to select the best suitable unsteady configurations, enable a quasi-steady comparison with the unsteady campaign as well as characterize the numerical strengths and deficiencies of the CFD tool to simulate shock boundary layer interactions (SBLI).

A list of practical objectives towards the steady state campaign is presented below:

- Relate basic flow structures to the operating conditions, namely the outlet static pressure. Typically of interest are, for instance, the shock location, the separated regions location and size, the amplitude and spectrum of natural (steady state) shock oscillations, flow streamlines on the bump surface or streaklines in the free stream, vortices location, size and origin, side wall effects or 2D approximation limits,
- Confirm steady state results by comparing several experimental techniques with each others,
- Determine the inlet and outlet boundary conditions in order to compute qualitative simulations of the nozzle flow,
- Enable comparison between experiments and numerical simulations in order to obtain a deeper understanding of the flow field and/or outline the numerical deficiencies
of the CFD tool,

- Enable experimental comparison between steady state and quasi-steady unsteady cases for any operating point.

First, the experimental flow investigation is presented as it is more likely to represent the physic (within the set of assumptions and errors respective to each measuring technique) and should, thus, be considered as the reference in further comparison. For each test object a global flow description is presented followed by a discussion on the influence of the operating conditions, i.e. the outlet static pressure value, based on observations relative to each measuring techniques.

The second part of this chapter is dedicated to numerical simulations and comparison with experimental data. The numerical boundary conditions and meshes are first presented, followed by observations related to the above introduced flow phenomena. Secondly, a comparison between experimental and numerical results is performed with a systematic discussion on possible reasons to explain the eventual differences. In order to enhance the comparison between experimental and numerical data, result illustrations have been placed side by side in Appendix M, followed by two-dimensional plots for a more quantitative analysis. Finally, a conclusion over the steady state campaign is performed with a summary of the results obtained.

6.2 Experimental investigation

6.2.1 Introduction

The present section aims at describing, on both the 2D and 3D bump test objects respectively, the evolution of the different flow structures while the shock moves downstream in the diffusor and gets stronger, that is to say when the outlet static pressure decreases. The experimental steady state measurement campaign was performed using different measuring techniques over a wide range of transonic operating points, from subsonic to fully choked nozzle flow. Typical phenomena of interest are for instance the shock location and size, the interaction zone between the shock and the boundary layers the location and size of separated regions.

6.2.2 Two dimensional nozzle flow investigation

Results from the steady state measurement campaign performed on the 2D bump using several measuring techniques over a wide range of operating conditions are presented in figures M.5 to M.23 in Appendix M.

6.2.2.1 Global flow description

In quasi one dimensional analysis, the static pressure distribution in a nozzle is directly related to the section area. The flow accelerates in the convergent part and either decel-
6.2. EXPERIMENTAL INVESTIGATION

erates or re-accelerates in the divergent part depending on the flow velocity at the throat as sketched in figure 6.1.

- $P_{\text{out}} > P_{\text{critic}}$: Subsonic outlet.
- $P_{\text{out}} = P_{\text{critic}}$: Sonic throat, subsonic outlet, nozzle choked.
- $P_{\text{ns}} < P_{\text{out}} < P_{\text{critic}}$: Sonic throat, subsonic outlet, normal shock in diffuser (expansion no longer isentropic).
- $P_{\text{ns}} = P_{\text{out}}$: Normal shock at the exit.
- $P_{\text{ad}} < P_{\text{out}} < P_{\text{ns}}$: Supersonic and isentropic flow inside, oblique shocks at outlet (over-expanded flow).
- $P_{\text{out}} < P_{\text{ad}}$: Supersonic and isentropic flow inside, expansion waves at outlet (under expanded flow).

Figure 6.1: Quasi-one dimensional nozzle flow theory

In a two-dimensional analysis, the previous mechanisms are still present but local curvatures introduce new local trends. Indeed, a flow basically accelerates over a convex surface and decelerates over a concave surface, which leads to a conflicting situation between global and local trends as illustrated in figure 6.2. Especially, the influence of the side wall boundary layers has the same effect as a change in the curvatures. A separation or a sudden boundary layer thickening thus changes both the global section area evolution and the local flow gradients. The resulting behaviour obviously depends on the predominance of one trend compared to the other. As a result, the flow structure is directly influenced by the transition between strong mean flow gradients close to the lower wall and smoother gradients on the upper wall.

Figure 6.2: Two dimensional nozzle flow analysis
6.2.2.2 Steady state pressure distribution

Pressure distribution

The pressure distribution measured on the bump surface in the middle of the channel is plotted in figure 6.3 for all operating points. As the back pressure decreases, the flow reaches sonic velocity close to the throat, a sonic pocket develops over the bump, and a shock wave appears in the divergent part of the nozzle. As the outlet static pressure further decreases, the shock moves downstream and grows in strength. It is noteworthy that the displacement of the shock is not linear with the back pressure variation but rather depends on the local curvatures. For back pressure values above 106kPa, the shock impacts on the convex part of the bump and moves on to the concave curvatures for back pressure values below 104kPa. Whereas the change of the outlet static pressure is regular, the displacement of the shock seems lower when the outlet static pressure value is around 106kPa. At this operating point, the shock impacts slightly upstream of the inflection point of the curvatures. Additionally, although a clear pressure rise can be observed for outlet static pressure values above 102kPa, a smooth pressure evolution with no sudden pressure rise can be observed for lower back pressure values. This seems to be the result of the lambda foot of the shock impacting in the concave part of the diffuser rather than the lambda configuration itself as it already occurs for operating point with higher back pressure values.

![Steady State Experimental Results on 2D Sliding Bump](image)

Figure 6.3: Experimental steady state pressure distribution at mid-channel for all operating points

Shock location

As pressure measurements were performed on the surface underneath a turbulent boundary layer, the strong discontinuity is replaced by a diffused gradient over a larger area. The thicker the boundary layer, the larger the "influence area" of the shock. As shown in Delery and Marvin (1986), the boundary layer characteristics, laminar or turbulent, shape factor and displacement thickness, play an important role regarding the SBLI.

In the present case, the lower wall boundary layer goes under a favorable pressure gradient over the convex part of the bump, is accelerated and gets thinner upstream of
6.2. EXPERIMENTAL INVESTIGATION

the shock, which reduces the extent of the upstream domain of influence. Nevertheless, the diffuse pressure gradient through the BL mentioned above is clearly observable in figure 6.4 as the pressure rise features a certain length which seems to increase as the BL thickening increases for strong shock configurations, i.e. at low back pressure values. It should be noted that the upstream influence (propagation of the information) of the presence of the shock is performed through two mechanisms. First the subsonic part of the BL underneath the shock is a region where downstream pressure information can propagate through. The shock is thus “felt” upstream of its real location. Secondly, as a result of the sudden and strong APG through the shock, the BL thickening leads to a system of oblique front shock waves, which contributes to gradually increase the pressure distribution on the surface already far upstream of the shock location. This second mechanism tends to dominate when the inlet Mach number increases and the lambda shock system further develops.

![Figure 6.4: Experimental steady state shock location at mid-channel for all operating points](Image)

**Bi-dimensionality of the flow**

The pressure distribution over the bump surface is plotted in Appendix M at 25%&75% and 10%&90% of the channel width for each operating point. At high back pressure values, from 118kPa down to 110kPa, the curves collapse almost perfectly (see figures M.8(c) and M.8(e) for instance), which means a two dimensional flow pattern. However, for lower outlet static pressure (below 110kPa), the pressure distributions does not collapse as nicely (see figures M.18(c) and M.18(e)), which denotes a non symmetry in the flow field either due to different flow structures or a strong interaction with non symmetrical side wall boundary layers. Besides, for such operating points, the shock occurs slightly upstream on the each side of the channel (distributions at 10% and 25%) than in the center. This effect creates a V-shape of the pressure distribution (and the shock) over the bump as seen in Figure M.18(a). It is believed that a strong interaction occurs between the shock wave and the side wall boundary layers. It should be noted that the boundary layer on the lower wall are influenced by strong local gradients originating the curvature of the bump.
whereas the side wall boundary layers go under the global flow gradients influence (acceleration in contractions and deceleration in diffusors). Indeed, the lower wall boundary layer is strongly accelerated over the convex surface and thus stabilized when interacting with the shock wave whereas the side wall boundary layers do not accelerate as much and remain thicker. The resulting effect of the strong negative pressure gradient through the shock is then more critical and lead to a larger thickening of the side wall boundary layers.

Influence of separated regions

For operating points with low back pressure values, the pressure distribution downstream of the shock location (figure M.20(c)) is slightly "flattened" (lower values than expected), which is similar to a slight acceleration superimposed to the strong flow deceleration observed in this region. It is believed that a slight change of curvatures, due to a separation for instance, would have this resulting effect. Besides, the pressure distribution at 10%, 25%, 75%, and 90% of the channel (see figure 6.5 compared to figure 6.3) show even more important trends, which correspond to larger separations in the side wall lower corners.

Figure 6.5: Experimental steady state pressure measurements at different width locations for all operating points

An interesting observation concerns the pressure distributions at 10% and 90% for the operating point with outlet static pressure 106kPa in Figure 6.5. Although it should move downstream like the global trend in the center of the channel, it actually moves upstream of the "previous" operating point (at $P_{out} = 108kPa$). It is believed that the shock strength and position, as well as the interaction between the side wall boundary layers and the local curvatures at this exact location create a particular V-shape flow configuration. Indeed, for other operating points with lower back pressure values, the V-shape pressure distribution is not as clear as the one observed for this OP (figure M.17(c)).
6.2. EXPERIMENTAL INVESTIGATION

Anticipating the forthcoming analysis of flow visualizations on the 2D bump surface, the extent of separated regions at different locations in the channel’s width could be roughly estimated and are presented in Table 6.1. The comparison with the pressure distributions shown in figure 6.5 and 6.3 confirmed the obtained results.

<table>
<thead>
<tr>
<th>$P_{out}$ [kPa]</th>
<th>$X_{sep}$ [mm]</th>
<th>$X_{reat}$ [mm]</th>
<th>L [mm]</th>
<th>$X_{sep}$ [mm]</th>
<th>$X_{reat}$ [mm]</th>
<th>L [mm]</th>
<th>$X_{sep}$ [mm]</th>
<th>$X_{reat}$ [mm]</th>
<th>L [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>98</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>100</td>
<td>130</td>
<td>30</td>
</tr>
<tr>
<td>100</td>
<td>100</td>
<td>120</td>
<td>20</td>
<td>100</td>
<td>130</td>
<td>30</td>
<td>100</td>
<td>140</td>
<td>40</td>
</tr>
<tr>
<td>102</td>
<td>85</td>
<td>135</td>
<td>50</td>
<td>88</td>
<td>130</td>
<td>42</td>
<td>85</td>
<td>130</td>
<td>45</td>
</tr>
<tr>
<td>104</td>
<td>78</td>
<td>120</td>
<td>42</td>
<td>78</td>
<td>125</td>
<td>47</td>
<td>75</td>
<td>120</td>
<td>45</td>
</tr>
<tr>
<td>106</td>
<td>65</td>
<td>130</td>
<td>65</td>
<td>73</td>
<td>120</td>
<td>47</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>108</td>
<td>62</td>
<td>115</td>
<td>53</td>
<td>70</td>
<td>115</td>
<td>45</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>110</td>
<td>60</td>
<td>105</td>
<td>45</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 6.1: Separation and reattachment location based on pressure measurements

is noteworthy that this behaviour is not observed on for the ”next” OP (figure 6.5) with outlet static pressure 104kPa.

And so, for high back pressure values, the sonic pocket is limited to a small region confined at the throat of the nozzle which does not even reach the side walls. As the outlet pressure decreases, the sonic pocket extends downstream to a location which is farther than the shock position observed on pressure distributions in the center of the channel (figure M.17(c) in comparison with figure M.18(c) for instance). Furthermore, the shock line observed on the isentropic Mach number distribution extends even more downstream close to the side walls. This observation is of course unrealistic and denotes a large error in the total pressure assumption, and thus higher losses in the side wall corners.

Isentropic Mach number

The isentropic Mach number distribution presented in Appendix for all OPs illustrates the sonic pocket shape and size over the 2D bump surface (see figure M.7(c)). The black lines on the plots represent the sonic lines and shock locations calculated with the isentropic formula based on the static pressure measured on the surface and the assumption that the total pressure is constant between the settling chamber and the test section. However, although this assumption is correct for subsonic flows, it becomes erroneous for transonic flows with strong shock waves. Nevertheless, if the isentropic Mach number is not suited to determine the exact shock position, it gives a fairly good idea about the sonic pocket size and shape.

Standard deviation

The standard deviation of steady state pressure measurements basically represents the Root Mean Square (RMS) of fluctuations between the instantaneous and time averaged...
pressure value. It basically reveals region where pressure fluctuations are either lower or higher than the average. Figure 6.6 presents different plots of the standard deviation over a few operating points.

- At a back pressure of 118kPa, slightly higher pressure fluctuations can be observed at several particular locations: at the beginning of the bump (between x=0mm and 10mm), at the shock location (around x=40mm), and close to the side walls downstream of the shock (between 60mm and 120mm). As explained previously, the BL undergoes an unfavorable pressure gradient on a concave surface and becomes unstable, which exhibits higher pressure fluctuations. Accordingly, the pressure fluctuations are lower on convex surfaces of the bump as the BL is accelerated and thus stabilized. Furthermore, the pressure fluctuations located close to the side walls, at x=50mm, denote corner vortices originating the interaction between the side wall BL and the bump’s curvatures. Besides, it is interesting to note that the downstream influence of those corner vortices is not symmetric, which shows that the side wall BLs are not either.

- At 112kPa, as the shock grows in strength, the pressure fluctuations are more important in intensity. The interaction with the side wall BLs is also more important, clearly non symmetric and extends farther both downstream and in the center of the channel.

- At 110kPa, the V-shape observed on the mean pressure distribution can now be seen on the standard deviation plot. Larger and symmetrical interaction zones can also be observed in the corner downstream of shock location and limit the two-dimensionality of the flow to approximately 30% of the channel width. Besides, the low pressure fluctuations level just downstream of the shock suggests that no separation occurs yet at this operating point.

- At 106kPa, the V-shape is still present and larger pressure fluctuating regions can be seen in the lower wall corners as well as downstream of the shock in the center of the channel, suggesting large separations.

- At 102kPa, a lambda shock system has developed and the intensity of the pressure fluctuations increases underneath the shock. However, the size and intensity of the fluctuations in the lower wall corners actually decrease.

- At 98kPa, the V-shape observed previously has disappeared and the shock is straight through the channel’s width. It should be noted that at this operating point, the shock impacts on the concave curvatures of the diffuser.

**Skewness and flatness**

The skewness and flatness coefficients respectively represent the degree of asymmetry and peakedness of a distribution. They help characterizing a signal by statistical analysis and comparison with known probability distributions. For the instance, the skewness and
6.2. EXPERIMENTAL INVESTIGATION

Figure 6.6: Standard deviation of experimental steady state pressure measurements

flatness of a normal distribution are respectively 0 and 3. The exact definitions were presented in chapter 5 and illustrate in figure H.1. The skewness and flatness coefficients of steady state pressure measurements over the 2D bump have been plotted in figures 6.7 and 6.8 respectively for a few operating points. The contour colors have been simplified in order to visualize positive and negative values within a certain threshold. For the skewness plots, regions in orange or light blue denote pressure fluctuations with a skewness value within ±200 Pa, whereas red and dark blue zones indicate a skewness with absolute value over ±200 Pa.

First of all, it is interesting to note that pressure signals on most of the 2D bump surface have a low skewness value, close to 0, thus to a normal probability distribution. Secondly, many of the flow structures can be recognized and assimilated to positive or negative skewness values. For instance, the pressure signals underneath the shock always feature a high skewness value as a strong pressure rise slightly oscillates over the surface whereas separations or BL thickening seem to be characterized by low positive skewness values as illustrated in figure 6.7(d) close to the side walls. Although, it is not straightforward to correlate the asymmetry of the pressure signals to the different flow structures, it is, however, interesting to get an idea about the pressure signals in some regions of the flow.

Similarly, the contour colors were simplified for the flatness plots and regions in orange or light blue denote pressure fluctuations with flatness values around 3000. Although no
clear flow structures, except the shock, can be observed, it is interesting to note that most of the pressure signals over the 2D bump have a more or less constant flatness value around 3000. A high flatness is characteristic of a narrow distribution around the mean value.

6.2.2.3 Conventional Schlieren visualizations

Low speed Schlieren

Low speed Schlieren visualizations with time-averaged shock positions, confidence intervals and extreme shock positions are presented in Appendix M for each operating point. At the highest back pressure value, although the pressure measurements (via the isentropic Mach number) revealed a small sonic pocket at the throat of the bump, the presence of a BL diffuses the small density gradients and no shock can be seen at this operating point. As the outlet static pressure decreases, the sonic pocket and the shock grow in size and strength respectively, and the high density gradients appear through Schlieren visualizations. A lambda foot shock structure appears at 106kPa on the bump surface whereas the nozzle flow is not yet choked. The shock then blocks the entire channel at 108kPa and another lambda shock structure appears on the upper wall at 98kPa. It should be reminded that the Schlieren technique reveals high density gradients through
6.2. EXPERIMENTAL INVESTIGATION

Figure 6.8: Flatness of experimental steady state pressure measurements

...the whole test section, including side walls effects on the density field. Those effects tend to bend the shock as it interacts with the side wall boundary layers and give the illusion of a certain thickness of the shock which is not representative of the reality. However, the post treatment method only detects positive light intensity variations, from black to bright, and the downstream edge of the darken shock image is assumed as the real shock position. The time-averaged steady state shock position was then determined by statistical analysis of instantaneous shock positions.

The mean shock positions on the 2D bump, presented in figure 6.10(a) together with pressure measurement results for each operating point, corresponds to the prolongation of the mean shock position determined in the channel, right above the surface. As mentioned earlier, it is difficult to state where exactly is the shock position on the surface since the pressure rise is diffused through the lower wall boundary layer. From Schlieren visualizations however, it appears that the mean shock position is located within this region where the pressure rises, and actually corresponds more to the end of the pressure jump. The fact that the pressure starts to rise slightly upstream of the position where the shock was detected by Schlieren visualizations can be explained by two phenomena: The propagation of pressure information through the subsonic part of the boundary layer, and the presence of oblique compression waves originating the sudden thickening of the boundary layer. It is indeed interesting to note that, for low back pressure values, the start of the pressure rise exactly corresponds to the beginning of the lambda shock structure.
Furthermore, the pressure rise on the surface underneath the shock remains quite sharp as long as the downstream "leg" of the lambda foot is small, which seems to be the case as long as the shock wave is located on the convex curvatures of the bump as illustrated in figure 6.9(a). As soon as the shock impacts on the concave surface of the diffuser, the lambda shock structure seems rather composed of regular oblique compression waves which explains the smooth pressure distribution seen in Figure 6.9(b) at 98kPa. It seems, consequently, that the structure of lambda shock depends on the pressure gradients in the boundary layer or close to the surface.

\[(a) \quad P_{out}^{\text{foot}} = 110\, \text{kPa}\]
\[(b) \quad P_{out}^{\text{foot}} = 98\, \text{kPa}\]

Figure 6.9: Schlieren visualizations of lambda foot shock structure for steady state operating points

In order to characterize the steady state fluctuations of the shock, a 95% confidence interval (assuming a normal probability distribution) was plotted as a function of the channel’s height for all different operating points in figure 6.10(b). For high back pressure values, from 114kPa to 108kPa, the shock oscillates with the same amplitude, of about 4 to 5mm, throughout the channel’s height. As soon as the lambda shock structure appears on the lower wall, at 106kPa, the amplitude of fluctuation of the shock increases from 5mm at the foot of the shock \((z=25\, \text{mm})\) to 10mm in the middle of the channel \((z=70\, \text{mm})\). The same tendency is observed on the next two operating points with lower back pressure values. At 104kPa for instance, the shock fluctuates (at 95%) within a 8mm and 20mm long interval respectively at the lower and upper wall (at \(z=33\, \text{mm}\) to 20mm at \(z=115\, \text{mm}\) exactly). This confidence interval somehow represents a degree of stability of the shock since the incoming pressure perturbations are supposed to be the same throughout the channel’s height. As a result, the region close to the upper wall, where the mean flow gradients are small, seems to be more exciting towards the shock fluctuations than the region with strong curvatures, thus high mean flow gradients. As observed, the variation in the amplitude of the shock fluctuations is maximum for 104kPa, but then decreases as the shock boundary layer interaction gets stronger on the upper wall. At 100kPa and 98kPa for instance, the amplitude of fluctuations of the shock remains constant around 10mm throughout the channel’s height. A strong shock boundary layer interaction, characterized by a lambda shock structure seems then to stabilize the shock fluctuations.

High speed steady state Schlieren

As discussed in the "Air flow quality" section in chapter 5, it is difficult to keep the operating conditions strictly stationary and perturbations originating the different wind
6.2. EXPERIMENTAL INVESTIGATION

(a) Time-averaged shock positions  (b) 95% Confidence Intervals of steady state shock positions

Figure 6.10: Schlieren visualizations of steady state shock positions for all operating points

tunnel components propagate in the test section. In order to correlate the "inherent" pressure perturbations to the quasi-steady shock oscillations, high frequency Schlieren visualizations and pressure measurements were performed simultaneously. Two fast response transducers were located on the upper wall, upstream and downstream of the 2D bump location (at x=-30mm and x=180mm). The acquisition system, procedure, and data reduction methods follow the explanations presented in the experimental model chapter. The instantaneous shock position was recorded, together with the pressure fluctuations, at a frequency of 1kHz and 16kHz respectively. A low pass filter was applied to the pressure measurement to avoid bias errors, and Fourier analysis was performed on the time fluctuating signals. The amplitude of the FFTs are presented for three different operating points in figure 6.11.

First of all, the time averaged shock position as well as the 95% confidence interval (assuming a normal probability distribution) were calculated based on the present high speed Schlieren visualizations, for each presented operating point, and confirm the results presented in figure 6.10 above. At 112kPa, the FFT content of both pressure signals are very similar. At this operating point, the nozzle is not yet choked and the perturbations propagate freely upstream and downstream the channel. The similitude with the FFT content of the shock motion is striking. Most of the largest peaks (at 25Hz, 35Hz, 45Hz, 105Hz and 350Hz) can be observed on both FFTs. At 106kPa, the shock has still not reached the upper wall but differences already appear on the pressure measurement signals. The pressure fluctuations are now globally smaller upstream, which could mean that a part of the downstream perturbations cannot propagate beyond the shock location. The similitude with the shock motion is still present, especially regarding the peaks at 25Hz and 35Hz. At 100kPa, the flow is completely choked, and downstream perturbations can no longer propagate upstream. As a result the FFT content of the upstream pressure signal only corresponds to downstream propagating perturbations. It is then interesting to note the similarity between the FFT of the "natural" shock motion and the FFT of the...
CHAPTER 6. STEADY STATE RESULTS

Figure 6.11: High speed Schlieren visualizations of steady state shock oscillations

(a) $P_{out}^a = 112\text{kPa}$

(b) $P_{out}^b = 106\text{kPa}$

(c) $P_{out}^c = 100\text{kPa}$
6.2. EXPERIMENTAL INVESTIGATION

downstream pressure signal. Downstream perturbations seem therefore to have a much more important impact on the quasi-steady behaviour of the shock. As a conclusion, the similitude in the frequency composition between the pressure and the shock motion signals showed that pressure fluctuations and shock motion are tightly correlated. Besides, it seems that downstream perturbations have a more important effect on the shock motion than upstream perturbations.

6.2.2.4 Oil visualizations

Separation bubble

Oil-paint visualizations is well suited to display reversed flows and streamlines over a surface. This technique was therefore performed for each operating point and pictures of the flow patterns over the 2D bump are presented in Appendix M. For high outlet static pressure values, from 118kPa to 114kPa, the flow is attached to the surface and a slight boundary layer thickening can be observed on the side wall corners around x=45mm. At 112kPa, a white strip can be observed at x=55mm and corresponds to white paint accumulating in the region due to the strong deceleration underneath the shock location. At 110kPa, the negative pressure gradient created by the sudden pressure rise is strong enough to reveal a separation bubble (at x=62mm) and create large boundary layer thickening in the side wall corners. At lower back pressure values, the flow fully separates and large vortices originate the strong interaction between the shock and the side wall boundary layers as illustrated in figure 6.13. The location of the separation and reattachment points have been plotted in figure 6.12 together with the pressure rise region. It seems that the separation occurs slightly downstream of the beginning of the pressure rise as logically assumed for a shock induced separation. The size of the separated region increases until a maximum of 50mm long at 102kPa. For lower outlet static pressure values, the size of the separation decreases until the flow completely reattaches at 96kPa (not shown in Appendix M). It is noteworthy that for operating points with back pressure between 100kPa and 98kPa, the shock impacts on the concave surface of the diffuser and a global flow deceleration precedes and attenuates the sudden pressure rise through the shock. Besides, the large lambda foot shock structure, composed by a set of oblique compression waves as seen in figure 6.9(b) also diffuses the negative pressure gradient, which might be the cause for the separation to disappear at extremely low back pressure operating points.

Flow structure description

Oil visualization pictures with outlined separated regions are presented in figure 6.13. The flow on the lower wall of the channel features two or three vortices depending on the operating point. The first vortex is really small and originates the interaction between the curvatures and the side wall boundary layers. Located on each side wall, upstream of the separation bubble, it triggers the thickening of the side wall boundary layer. A second vortex originates the interaction between the shock and the side wall boundary layer and is located on each side of the separated region. Finally a third vortex appears when the side wall boundary layer thickens early enough to impact on the shock, which happens only at 102kPa and 100kPa. Furthermore, the size and length of the separated zone in
CHAPTER 6. STEADY STATE RESULTS

Figure 6.12: Oil visualizations of steady state shock induced separation for all operating points

the side wall corners seem approximately the same throughout the different operating points (from 108kPa to 100kPa). Finally, the flow is two dimensional over 90% of the channel’s width until the lambda shock structure appears. Then, the interaction between the side walls and the shock becomes intense and the flow separates in the middle of the channel. The interaction between the separated flow, the shock wave and the thick side wall boundary layers thereafter reduces the two dimensionality of the flow to a narrow strip of about 20mm in the center of the channel. It is noteworthy that, at 98kPa, as the separation in the middle of the channel nearly disappears, the size of the separation in the corners greatly decreases as well. This is probably due to structure and location of the shock, a large lambda foot over a concave surface which creates smoother pressure gradients.

6.2.2.5 Hot-film measurements

The heat transfer coefficient over the 2D sliding bump was calculated for each operating point according to the method presented in the experimental chapter. The results have been plotted in Appendix M together with the beginning and the end of both the pressure rise (dashed line) and the separated region (continuous line). It should be noted that the hotfilm measurement results have a fairly poor accuracy mainly due to the determination of the cold resistance of each sensor and the deterioration of the hotfilm array while sliding the 2D test object. However, a qualitative analysis by direct signal comparison is still of interest. Besides, a smoothed curve has been added to the scattered data in order to give a better idea of the general trends.

Heat transfer coefficient

Although most of publications in the literature simply present the output bridge
6.2. EXPERIMENTAL INVESTIGATION

voltage results, it was decided to rather analyze the heat transfer coefficient as it is directly proportional to the wall shear stress. As a result, the heat transfer coefficient, $h$, presents an evolution closely related to the behaviour of the flow field and the local mean flow gradients. Typically, it decreases on concave curvatures and increases on convex curvatures as the flow respectively decelerates and accelerates. Globally, the heat transfer coefficient reaches a minimum slightly before the beginning of the bump, at $x=0\text{mm}$, and strongly increases to a maximum located upstream of the shock. Then, depending on the presence of a shock induced separated region, the heat transfer coefficient suddenly decreases and remains more or less constant over a plateau. If no separation occurs, the heat transfer coefficient slowly decreases to a second minimum around $x=100\text{mm}$ as observed for 116kPa, and finally increases again until the end of the 2D bump.

Interestingly the sudden drop of $h$ coincides more or less with the beginning of the pressure rise and the location of the separation point. However, the scatter of the results is such that the detection of the separation point through a sudden change of the heat transfer coefficient value would be nearly impossible without the help from other experimental techniques.

Heat exchange fluctuations

The heat transfer coefficient fluctuations were determined using the Root Mean Square value of the output bridge voltage and are presented in figure 6.14. Although the values seem doubtfully low, a qualitative analysis of the signals shows that high heat transfer fluctuations occur at specific locations where the boundary layer seems unstable. Indeed, the fluctuations increases until the beginning of the bump as the flow is under a negative pressure gradient close to the surface, and suddenly drop to a low level while the flow is accelerated over the bump. It is interesting to note that the fluctuations start to increase again at the location of the pressure rise and reach a maximum at the location of the reattachment point. Those observations are in agreement with results achieved by Hodson (1984) and Pucher and Göhl (1987). The fluctuations thereafter decreases towards the end of the bump.

Flow patterns detection

Although the above analysis was able to roughly locate separation and reattachment point locations with the help of other measuring techniques, it appears difficult, from hotfilm signals alone, to accurately locate a separated region. Hotfilm results however might give a significant hint on whether the boundary layer separates or not. As pointed out by Hodson (1984), the main weakness of hotfilm measurement technique is its inability to independently and accurately detect regions of zero value wall shear stress. A possible reason might be the only availability of results at discrete location rather than a continuous evolution of the heat transfer coefficient along the surface.

Hotfilm measurement are nevertheless of a high interest as they allow the determination of the heat transfer coefficient, which is proportional to the wall shear stress, without any calibration. Further measurement campaigns would be needed in order to further
understand hotfilm signals and maybe automatically detect flow structures which would allow unsteady measurements and the characterization of unsteady separated region behaviour.

6.2.2.6 Boundary Layer measurements

Introduction

Inflow boundary information are of high interest in order to characterize and understand Shock Boundary Layer Interaction. Within the framework of this project, inlet boundary layer profiles were investigated on the lower wall of the rectangular channel duct (without any test object) by Sigfrid (2003) using hotwire and PIV techniques. Although the measurements do not exactly correspond to the same operating conditions as the ones presented in this work, the results are considered relevant enough to get a rather good estimation of the inlet boundary layer on the lower wall and are therefore presented below in comparison with other BL measurements.

At an early stage of the project, a 3D Laser-Two-Focus equipment was put into service at the division and a short measurement campaign was performed in the wind tunnel facility with the aim at determining, whether or not, the L2F technique was suitable to boundary layer measurements in such high speed flow configuration. As a result, flow velocity measurements were performed on all four walls, at both inlet (x=-90mm) and outlet (x=200mm) stations, in the rectangular channel duct. Unfortunately, as a preliminary test of the the system, the laser measurements were performed at a lower pressure level (lower density thus lower Reynolds number). The operating conditions under which the hotwire and L2F measurements were performed are presented in Table 6.2 below. A short description of the working principle of the L2F system is presented in Appendix L.

<table>
<thead>
<tr>
<th></th>
<th>$P_{in}^t$ [kPa]</th>
<th>$P_{in}^s$ [kPa]</th>
<th>$T_{in}^t$ [K]</th>
<th>$T_{in}^s$ [K]</th>
<th>$U_{in}^\infty$ [m/s]</th>
<th>$\rho_{in}^\infty$ [kg/m$^3$]</th>
<th>$M_{in}$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hotwire</td>
<td>137.8</td>
<td>101</td>
<td>304</td>
<td>278.1</td>
<td>227.8</td>
<td>1.265</td>
<td>0.681</td>
</tr>
<tr>
<td>L2F</td>
<td>121.8</td>
<td>90</td>
<td>304</td>
<td>278.8</td>
<td>225</td>
<td>1.124</td>
<td>0.672</td>
</tr>
</tbody>
</table>

Table 6.2: Operating conditions during Hotwire and L2F measurements in the empty test section

Although the laser measurements gave a good estimation of the BL thicknesses on all four walls, the results also showed that the L2F system is not really suited for BL measurements. Indeed, the BL profile was found to evolve mostly within the first millimeter close to the surface. Unfortunately, the velocity profile could not be measured close enough to the wall due to the size and shape of the measurement volume and the amount of reflected light while rotating one beam around the other in the vicinity of the surface. Besides, the alignment and reference setup of the laser beams are extremely sensitive to perform and have a high influence on the results, especially in BL measurements. However, the achieved results are still of interest to estimate other BL characteristic parameters like displacement and momentum thicknesses as well as the shape factor. As a result, a short introduction to the working principle of a L2F system is first presented, followed by the
6.2. EXPERIMENTAL INVESTIGATION

<table>
<thead>
<tr>
<th>Laser-Two-Focus</th>
<th>Hotwire</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lower wall</td>
<td>Upper wall</td>
</tr>
<tr>
<td>Inlet</td>
<td>Outlet</td>
</tr>
<tr>
<td>$\delta_{99}$</td>
<td>11.5</td>
</tr>
<tr>
<td>$\delta^*$</td>
<td>0.83</td>
</tr>
<tr>
<td>$\theta$</td>
<td>0.64</td>
</tr>
<tr>
<td>$H$</td>
<td>1.29</td>
</tr>
</tbody>
</table>

Table 6.3: Experimental boundary layer characteristics

BL measurement results for both hotwire and L2F techniques.

BL measurement results

Vertical and horizontal traverses were performed using the L2F system at both inlet and outlet, in the center of the rectangular channel duct. The boundary layer profiles were then extracted and are presented in figures 6.15 and 6.16 respectively for L2F and Hotwire measurements. Additionally, BL characteristic parameters have been calculated according to the definitions introduced in equations 2.4 and 2.5 reminded below. The results are reported in Table 6.3.

- BL thickness: $\int_{0}^{\delta} \frac{U}{U_{\infty}} dy = 0.99$
- Displacement thickness: $\delta^* = \int_{0}^{\infty} \left(1 - \frac{U}{U_{\infty}}\right) dy$
- Momentum thickness: $\theta = \int_{0}^{\infty} \frac{U}{U_{\infty}} \left(1 - \frac{U}{U_{\infty}}\right) dy$
- Shape factor: $H = \frac{\delta^*}{\theta}$

From L2F and hotwire measurements, it appears that the BL on the lower wall at the inlet of the test section is about 12mm thick whereas the BL on all other walls are slightly thinner (10 to 11mm thick). The main difference however comes when comparing the displacement thicknesses and shape factors. The BLs on the side walls seem to have a higher displacement thickness and shape factor, which is more critical in terms of stability towards flow separation. Indeed, a turbulent BL is very likely to separate when the value of the shape factor exceeds 2.4. The side wall BLs are thus more sensitive to any negative pressure gradient as mentioned in the previous paragraphs. The dissymmetry in the inflow BLs could originate a general non uniformity in the incoming flow field or simply the dual contraction design. Indeed, the two small settling chamber probably does not fully dissipate the flow gradients originating the last corner of the pipe system. In addition, the double contraction design certainly has an influence on the dissymmetry between side wall BLs and the other BLs. Observing the behaviour of the different BLs between the inlet and outlet stations, it is noteworthy that the BLs on the side walls thicken much more than on the upper and lower walls. As seen in Figure 6.15, it is actually difficult to tell where exactly the edge of the side wall BLs is located at the outlet. Indeed, the
thickening of all BLs is equivalent to a narrowing of the effective section area and the flow slightly accelerates in the free stream, which induces a continuous increase of the velocity on the side wall BLs.

Figure 6.16 shows a breakdown of hotwire measurements into velocity and velocity fluctuations. The temperature profile was however not measured and a Walz’s distribution\(^1\) was assumed, based on the total temperature value in the settling chamber. Although the velocity profile seems in good agreement with the L2F measurements, the velocity fluctuations level is definitely underestimated due to a too long hotwire probe, thus unable to resolve the smallest turbulent scales in the vicinity of the wall (Sigfrid, 2003).

\subsection{6.2.3 Three dimensional nozzle flow investigation}

The analysis concerning the influence of local and global mean flow gradients presented for the 2D nozzle is still valid here but much more complex as the local curvatures vary both in streamwise and spanwise directions.

Considering the complexity of the flow field over the 3D bump geometry, a global description of the flow structures is first performed at one operating point, and comparison to this reference case is thereafter presented to describe the changes for other operating conditions, i.e. different back pressure values.

The reference case is the one presented in figures M.31 and M.32, in Appendix M, obtained for an outlet static pressure value of 116kPa (inlet conditions being the same as for all operating conditions as listed in Table 5.8). Anticipating forthcoming explanations, this case was chosen as the steady state operating point for unsteady measurements and simulations, and should therefore be carefully investigated.

\subsubsection{6.2.3.1 Global flow description}

Results from steady state pressure measurements over the 3D bump surface at the OP with \(P_{\text{out}}^{\text{ref}}=116\text{kPa}\) are presented in figure 6.17 together with the geometrical location of the leading edge (LE), throat line and trailing edge (TE). The high and low curvature profile were also plotted, respectively at \(y=100\text{mm}\) and \(y=0\text{mm}\), in order to correlate the flow behaviour to the overall shape of the 3D nozzle.

Clearly, a three-dimensional pattern can be observed on all four subfigures. At the LE, as the flow decelerates while approaching the bump, the spanwise variation of the curvatures creates a pressure gradient through the width of the channel, which deviates the incoming flow already upstream of the beginning of the bump as seen in figure 6.18(a). It is noteworthy that the velocity deficit within the side wall BL interacts with the strong curvatures, which results in a corner vortex just upstream of the bump. Next to this vortex is located a “stagnation” point where the flow decelerates the most and thus corresponding to the highest pressure location in the nozzle. Further downstream, at the throat line

\(^1\)Temperature distribution corresponding to a ZPG laminar BL
location, can be observed a low pressure region corresponding to a supersonic pocket as the flow strongly accelerates over the convex surface of the bump. It is interesting to note that this supersonic flow region tends to follow the throat line and extends downstream as it spreads itself through the width of the channel. Consequently, the shock, which can be observed by a sudden rise in the pressure distribution in figure 6.17(c), is inclined in the spanwise direction as clearly seen by stagnating paint underneath the shock in Figure 6.18(a).

The isentropic Mach number is presented in figure 6.17(c) and show the extent of the sonic pocket on the surface. The blackened lines correspond to sonic iso-contours and the downstream one therefore to the shock location as experimentally measured. It should however be mentioned that the assumption to use the upstream stagnation pressure to calculate the isentropic Mach number on the bump surface is no longer valid downstream of the shock, which leads to a shift in the displayed shock position. As the real total pressure decreases through a shock discontinuity, the static pressure corresponding to the sonic iso-line is also lower. According to the pressure distribution, the shock should physically be located upstream of the black line shown in figure 6.17(c).

According to the Schlieren visualization (figure 6.18(b)) and the isentropic Mach number distribution (figure 6.17(c)), the shock wave spreads itself, in the spanwise direction, over the whole width of the channel but does not reach the upper wall. The nozzle is therefore not choked and the supersonic pocket only grows over 25% of the local channel height. This configuration corresponds fairly well to the ideal configuration drawn from the design objectives described in the experimental model chapter and sketched in figure 5.4.

Although the extent and shape of the shock throughout the height of the channel (as seen from Schlieren visualization in Figure 6.18(b)) would suggest a weak SBLI and no shock induced separation, the maximum Mach number value is about 1.45 on the high curvature side of the 3D bump. The stagnating paint observed in figure 6.18(a) is therefore believed to originate a small separation bubble underneath the shock location. The flow reattaches almost immediately downstream of the separation bubble.

Furthermore, the standard deviation of steady state pressure measurements over the 3D bump has been plotted in figure 6.17(d) and show three different regions of higher (only slightly at this OP) pressure fluctuations. At the TE first, as the flow is decelerated the BL becomes more instable and pressure fluctuations increase in this region. Secondly, under the shock location as the shock oscillates with a low amplitude due to free stream perturbations or incoming BL turbulence behaviour. And thirdly in a region located downstream of the shock, slightly in the corner, on the high curvature side of the channel. This "higher" fluctuation zone basically corresponds to a corner separation region. At this OP, the separation is not very large but does exist as seen in Figure 6.18(a) and clearly originates the interaction between the destabilized side wall BL (as it just went through a strong APG) and the bump curvatures.

The skewness and flatness are presented in figure 6.17(e) and 6.17(f) respectively. Characteristic zones have been plotted rather than all iso value contours. As a result, the
skewness of steady state pressure measurements, which represents the degree of asymmetry of a distribution, appears to be organized in four different zones. Most of the steady state pressure measurements on the surface seem to have a low positive skewness value (between 0 and 400), whereas isolated regions, namely around the shock location and in the downstream corner separated zone, feature low but negative values (between -400 and 0). Additionally, high negative skewness values (below -400) can be observed immediately downstream of the shock location. Those observations are useful to better characterize the steady state pressure signals and establish a correlation between pressure signals and flow structures. For instance, the region of high negative skewness value seems to correspond to the reattachment of the separation bubble underneath the shock. On the other hand, the flatness of steady state pressure measurements, which characterizes the degree of peakedness of a distribution, appears to be rather high (between 2000 and 4000) all over the 3D bump surface. This result illustrates the fact that pressure signals feature many sharp peaks, which is agreement with the air flow quality investigation presented in the previous chapter.

6.2.3.2 Influence of outlet static pressure value

The present subsection will now emphasize on the influence of the outlet static pressure value and describe the changes occurring in the different flow structures. Results from the steady state measurement campaign performed on the 3D bump using several measuring techniques over a wide range of operating conditions are presented in Appendix M, from figures M.25 to M.47.

6.2.3.2.1 Mach number distribution

As observed in figure M.24(b) (in Appendix M), the isentropic Mach number distribution at high back pressure value, typically above 128kPa, shows an entire subsonic flow field over the 3D bump surface. As the outlet static pressure decreases, a small sonic pocket appears on the high curvature side (y=100mm), at the throat line location, and already covers around 25% of the channel width at 126kPa(figure 6.19). For even lower values of the outlet static pressure, the sonic pocket further develops along the shock line and spreads itself towards the low curvature side wall. It is noteworthy that the shock is fairly well aligned with the throat line as designed and validated during the numerical simulations (Bron, 1997).

However, as mentioned above, the calculation of the isentropic Mach number distribution is then not correct any longer as the shock grows stronger, implying that the downstream sonic line does not represent the shock location any more. Considering the streamwise pressure distribution, the shock should physically be located somewhere upstream of that line. It is however difficult to accurately foresee where exactly the shock is since the original strong discontinuity is replaced by a smooth pressure gradient on the surface under the BL. However, the start of the pressure rise can be considered as a the shock location since it corresponds to the location of the first oblique shock, originating the BL thickening, and therefore represents the upstream extent of the interaction zone. Accordingly, a couple of observations can be drawn from the evolution of the isentropic
6.2. EXPERIMENTAL INVESTIGATION

Mach number as the outlet pressure decreases. First, for all OPs the shock has a certain transversal angle and is never completely perpendicular to the main flow direction. Secondly, a double shock configuration appears for low outlet static pressure values (below 106kPa). Indeed the isentropic Mach number on the surface slightly decreases downstream of the throat line location, and immediately increases to an even higher value meaning a second shock stronger than the first one. This behaviour is best observed in Figure 6.19 at lowest back pressure values (typically at 102kPa and 98kPa). From a global perspective, it seems as the first shock always occurs along the throat line whereas the second shock moves downstream and grows stronger in the diffusor as the outlet static pressure further decreases.

6.2.3.2.2 Schlieren visualization

The streamwise location and vertical extent of the 3D shock wave can be observed directly on Schlieren visualizations in Figure 6.20. For high back pressure value (say above 124kPa) the density gradients are too low to be detected by conventional Schlieren method. At lower outlet static pressure values, a small blackened line first appears on the bump at the throat line location, around x=30mm. As the back pressure value further decreases, the previous mark which represents high density gradients moves slightly downstream and also thickens in the streamwise direction. As the conventional Schlieren technique integrates the mean flow gradients through the whole width of the channel, the large black mark (more than 10cm large as seen in Figure 6.20(d)) is a consequence of the bend shock configuration observed in the previous paragraph.

For low outlet static pressure values (below 106kPa), the extent of the black mark becomes thinner in the free stream, which basically corresponds to a less pronounced transversal inclination of the shock through the channel’s width. However, it is important to note that this is not the case in the region close to the lower wall. As well seen in Figure M.43(a), the shock seems to keep its transversal inclination on the 3D bump surface and adopts a more 2D configuration in the free stream. It is also interesting to note that although the double shock structure could be observed at the same OP from pressure measurements, the first shock, probably too weak, does not appear on Schlieren visualizations.

For very low back pressure values, the (second) shock has developed into a large lambda configuration with a set of upstream oblique shocks. It is however difficult, from conventional Schlieren visualizations only, to clearly state whether there is a set of oblique shocks or if the same oblique shock is inclined through the width of the channel. Nevertheless, the lambda shock system clearly features a 3D structure with large separation on the bump surface as seen in figures 6.20(g) and 6.20(h). It is also difficult to say, from Schlieren visualizations, whether the separation occurs through the whole width of the channel or only a part of it. Finally, it should be noted that, as the outlet static pressure decreases, the interception point, where the oblique front shock, the normal rear shock and the oblique main shock meet, moves vertical towards the upper wall as the same time as the normal rear shock becomes weaker. Indeed it seems as the observed trend would lead to a normal shock configuration throughout the whole channel if the shock would move further downstream.
6.2.3.2.3 Oil-paint visualizations

The evolution of the different flow structures as a function of the back pressure is best illustrated by oil visualizations presented in figure 6.21 for a few OPs. At high outlet static pressure value, a thin white zone can be seen on the high curvature side of the 3D bump and corresponds to a separation bubble where the oil-paint mix stagnates. Similarly to the observations on the isentropic Mach number, the separation bubble grows, spreads itself through the channel width towards the opposite side wall and moves downstream as the back pressure decreases. Simultaneously, at the location (around x=60mm) where a large BL thickening could be observed, appears a corner separation (see figure M.29(e)) which thereafter grows in length and size as reported in Table 6.4 and illustrated in figure 6.21 (in details in Appendix M).

At low outlet static pressure values, the interaction between the strong lambda shock system, the side wall BLs and the corner separation creates a really complex flow structure as best illustrated in figure M.43(e). As the back pressure decreases, the shock reaches the low curvature side and interacts with the side wall BL, which results in a clockwise vortex as seen in Figure M.39(e). The combination of the strong APG through the shock and the curvatures destabilizes the BL which is close to separate as seen in Figure 6.21(f) where the flow downstream of the shock location still has a positive but very low streamwise velocity. When the lambda shock structure appears, for an outlet static pressure value of about 104kPa, the interaction becomes extremely complex and a double separation pattern with multiple vortices is created as observed in Figure M.43(e). It seems as the shock induced separation originating the front oblique shock interacts with the side wall BL, which results in corner vortices as showed in Figure M.46(c). As the back pressure further decreases, the clockwise vortex observed at x=80mm in the center of the channel in Figure M.43(e) migrates towards the corner separation on the high curvature side of the bump (figure 6.21(g)) and interacts with the reversed flow. As a result, a counter clockwise vortex appears next to it (figure 6.21(h)) and both vortices thereafter merge into each others (figure 6.21(i)). At this outlet static pressure value, a large separation covers the entire channel width and extends from approximately x=90mm to x=175mm as seen in the medallion of figure 6.21(i).

6.2.3.2.4 Pressure measurements

Pressure distribution

Picture 6.22 presents the steady state pressure distribution for all OPs at different
spanwise locations. The evolution of the pressure distribution can then easily be analyzed as a function of the back pressure for a given position in the channel width. Appendix M presents in details the pressure distribution individually for each OP and all locations in the spanwise direction.

From a global observation, it is interesting to note the following. For high outlet static pressure values and all spanwise positions the pressure distribution over the 3D bump surface is fairly regular in the sense that all curves are regularly spaced which denotes a continuous and smooth evolution between one OP to another. This observation is valid until a back pressure value of 114kPa. Beyond this value, i.e. for lower values, the pressure distribution downstream of the shock location is influenced by the separation and a progressive inflection of the curves occurs as the back pressure further decreases as well illustrated in figure 6.22(b). This observation is in agreement with the oil-paint visualizations which showed a large increase of the corner separation from an outlet static pressure value of 114kPa as seen in Figure 6.21(c) and reported in Table 6.4.

Besides, on all locations through the channel width and for outlet static pressure values below 106kPa, the decrease in pressure following the first shock denotes a reacceleration of the flow and the presence of a second shock. This reacceleration of the flow occurs at different streamwise positions depending on the spanwise location, and seems to follow the throat line. Both above observations are also in agreement with the analysis of the isentropic Mach number performed previously and illustrated in Figure 6.19.

Furthermore, it is interesting to compare the pressure distributions and the flow structures observed with oil visualizations. As a result the vortices do not seem to have a large influence on the pressure distribution. A few observations however seem to correlate a large inflection of the pressure distribution curves to the presence of a vortex as seen for instance in pictures M.46(c) and M.46(f) at x=90mm and y=90mm. This result is explained by the fact that both clockwise and counter clockwise vortices are also seen as ejection points which logically leads to a lower pressure distribution on the 3D bump surface.

**Standard deviation**

The standard deviation (or RMS) of the steady state pressure measurements is plotted in Figure 6.23 and basically represents the level of fluctuations relatively to an ensemble or (in our case) a time-averaged value. As mentioned in the previous subsection, three different regions of higher RMS value can be observed at high outlet static pressure. Namely, the upstream concave curvature at the LE of the bump, the region underneath the shock, and the downstream concave surface close to the TE of the bump. These regions feature a higher level of fluctuations basically because of the deceleration of the flow. The BL is suddenly under an APG. The skin friction decreases, the shape factors increases, the turbulence bursting are enhanced and the velocity fluctuations, thus the pressure fluctuations, increase. As the outlet static pressure decreases the shock and the flow deceleration gets stronger. For a back pressure value of 114kPa the corner separation on the high curvature side suddenly grows and the pressure fluctuations increase proportionally in this region as observed in Figure 6.23. Additionally some large fluctuations level are located
CHAPTER 6. STEADY STATE RESULTS

underneath the shock as it slightly oscillates (due to turbulence effect and the presence of various perturbations in the flow field) and passes over a measurement pressure tap.

As the back pressure further decreases the pressure fluctuations increases together with the extent of the corner separation region and reaches the opposite side wall. This observation is in total agreement with the analysis on the oil-paint mix visualizations. Finally for very low outlet static pressure values the pressure fluctuations are limited to the low curvature side of the bump, leaving a low fluctuation level on the high curvature side.

Skewness and flatness coefficients

Similarly, the skewness and flatness coefficients of the steady state pressure measurements are presented in pictures 6.24 and 6.25 respectively in order to better characterize the pressure signals on the bump. It should be reminded that the skewness and flatness represents respectively the degree of asymmetry and peakedness of a distribution.

In a general perspective it seems as the steady state pressure distribution on the 3D bump features positive and negative low skewness values. Although no general and systematic behaviour could be observed it seems, for certain OPs, that high positive skewness values are to be found underneath the shock wave location (Figure 6.24 at 110kPa) whereas negative values can be associated to BL thickening or separation (Figure 6.24 at 14kPa). Similarly high flatness coefficient values seem to be found underneath the shock location (Figure 6.25 at 110kPa) whereas an average value of 3000 is found all over the 3D bump surface with no special variation at locations where separations are expected.

Although it is tempting to associate values or trends of the skewness and flatness parameters to certain flow structures observed at some OPs, no general conclusion could be clearly drawn.
6.2. EXPERIMENTAL INVESTIGATION

Figure 6.13: Separated flow pattern with steady state oil visualization

(a) $P_{out} = 108\text{kPa}$  
(b) $P_{out} = 106\text{kPa}$  
(c) $P_{out} = 104\text{kPa}$  
(d) $P_{out} = 102\text{kPa}$  
(e) $P_{out} = 100\text{kPa}$  
(f) $P_{out} = 98\text{kPa}$
CHAPTER 6. STEADY STATE RESULTS

Figure 6.14: Heat transfer coefficient fluctuations over 2D bump
6.2. EXPERIMENTAL INVESTIGATION

Figure 6.15: Boundary layer measurements performed with a Laser-Two-Focus system

Figure 6.16: Inlet boundary layer profiles on lower wall, Hotwire measurements (Sigfrid, 2003)
CHAPTER 6. STEADY STATE RESULTS

Figure 6.17: Experimental steady state pressure measurements on 3D bump (weak shock configuration)

Figure 6.18: Experimental flow visualization on 3D bump (weak shock configuration)
6.2. EXPERIMENTAL INVESTIGATION

Figure 6.19: Steady state isentropic Mach number from pressure measurements in 3D nozzle
Figure 6.20: Steady state conventional Schlieren visualizations in 3D nozzle
6.2. EXPERIMENTAL INVESTIGATION

(a) $P_{\text{out}} = 126\, kPa$

(b) $P_{\text{out}} = 122\, kPa$

(c) $P_{\text{out}} = 118\, kPa$

(d) $P_{\text{out}} = 114\, kPa$

(e) $P_{\text{out}} = 110\, kPa$

(f) $P_{\text{out}} = 106\, kPa$

(g) $P_{\text{out}} = 102\, kPa$

(h) $P_{\text{out}} = 100\, kPa$
CHAPTER 6. STEADY STATE RESULTS

Figure 6.21: Steady state oil-paint visualization in 3D nozzle

(i) $P_{out} = 98kPa$

Figure 6.22: Steady state pressure distribution for all operating points at different $y$ locations
Figure 6.23: Standard deviation (RMS) from steady state pressure measurements in 3D nozzle
Figure 6.24: Skewness from steady state pressure measurements in 3D nozzle
Figure 6.25: Flatness from steady state pressure measurements in 3D nozzle
6.3 Numerical investigation

6.3.1 Introduction

This section aims at presenting the details of the numerical computations performed on the 2D and 3D nozzles, including a description of the different meshes and applied inlet boundary conditions.

From a general concern, the choice and selection of the grid mesh as well as a "physically" correct set of inlet boundary conditions are two essential aspects when performing numerical computations. Indeed, although no shape constrain is imposed in the finite volume formulation, the generation of the mesh grid should respect the assumptions associated to the formulation. First, the volume variation between adjacent cells should not exceed the ratio of $1.3^2$. Secondly, the shape and node distribution should be performed in respect to the mean flow gradients distribution, which supposes a prior knowledge of the flow structure. Thirdly, in order to minimize the error on the volume calculation\(^3\) of each cell, the grid distribution should, as far as possible, be orthogonal inside the domain and, more important, perpendicular to external boundaries. Indeed, the imposition of physical properties at the boundaries of the computational domain would be altered in case of inaccuracies or errors introduced in the calculation of the numerical fluxes through those boundaries.

Furthermore, as mentioned above, the knowledge of the inlet and outlet physical flow properties, as well as the evolution of some turbulent properties inside the domain, is essential to compute a quantitatively correct simulation of the flow field. Conservative variables (used internally by the CFD tool) are usually calculated from primitive (or aerodynamic) variables measured at the inlet and outlet stations of the test section using different available measuring techniques. Typically, the stagnation pressure and temperature as well as the flow angle are imposed on each node at the inlet whereas a static pressure distribution is set at the outlet. As a result, the determination of the complete flow field (pressure, temperature and velocity profiles in both direction in case of a 3D calculations) is required at the inlet whereas the static pressure distribution only is needed at the outlet. Furthermore in case of turbulent flow calculations using a two equations model, some knowledge of the incoming turbulence is also required. Typically the turbulent kinetic energy, $k$, as well as the turbulent dissipation\(^4\) distributions at the inlet boundary are imposed.

Unfortunately, no preliminary experimental data was available for the initialization of the numerical simulations and the numerical boundary conditions were therefore obtained by simulating the development of BLs within a rectangular duct. The following section thus aim at presenting the computation used to determine the inlet boundary conditions for the nozzle flow simulations and thereafter the 2D and 3D calculations over the 2D and

\(^2\)Value commonly accepted in order to satisfy the assumption that the conservative properties are constant inside each cell

\(^3\)Performed by evaluating the Jacobian, $\sqrt{g}$, of the geometrical transformation (see numerical method in Appendix A)

\(^4\)Specific dissipation, $\omega$, for Wilcox’s model
6.3. NUMERICAL INVESTIGATION

3D nozzle geometries.

6.3.2 Computation initialization: Rectangular duct simulation

As mentioned above, the computation of a rectangular duct flow was performed in order to initiate the calculations on the nozzle geometries. The idea was to simulate the development of BLs in a wall bounded flow similar to the one encountered upstream of the test section.

6.3.2.1 Computational grid and boundary conditions

Physical geometry

The main idea of the present calculation was to simulate the development of the BLs in a wall bounded flow similarly to the experimental case, i.e. from the settling chamber to the test section. However considering the lack of information on the pressure and velocity fields within the settling chamber, the computing cost for a full scale simulation compared to the expected quality (and relevance) of the results, it was decided to restrict the computational domain to the part of the facility comprised between the throat of the wind tunnel contraction and the inlet of the test section. Ideally the free stream velocity upstream of the contraction is very small and the acceleration through the wind tunnel contraction thinners the BL so that considering the development of a BL from the throat of the contraction appeared to be a fairly good compromise. The advantage of such a computation is also to provide a set of numerically stable variables rather than to specify, directly in the nozzle computations, theoretical BL profiles which would not necessarily be coherent with each others. There usually is, in such case, a rapid and sometimes important evolution of the different variables (most often turbulent quantities) in the first few planes of the domain in order to balance the aerodynamic quantities in respect to the discretized equations of the numerical model. Naturally, this simulation does not pretend to reproduce the physical inlet flow field and BL profiles, but it is believed to give a fairly good approximation of the main flow structure at the inlet. The exact inlet boundary conditions, i.e. pressure, temperature and velocity profiles as well as fluctuating velocity components, should be measured accurately in the future in order to enable a qualitative comparison between numerical and experimental results.

Actually, anticipating forthcoming results, the present calculation appeared to provide a fairly consistent set of inlet boundary conditions although it was performed before any experimental results was obtained, with a very few hints on the flow characteristics within the wind tunnel test section.

Practically, the length of the rectangular duct was chosen slightly longer (50cm) than the physical distance (37cm) between the contraction throat and the inlet of the test section in order to have the possibility to select different boundary conditions and especially thicker BL profiles if needed in future nozzle computations. As a result, an intermediate plane was actually used as inlet condition for the nozzle simulations. The mesh grid and
boundary conditions used in the rectangular calculations are specified below.

Mesh grid

Due to obvious symmetries, only a quarter of the rectangular duct geometry was computed. The physical dimension of the numerical domain were thus 50mm large, 60mm high and 500mm long. The geometry was discretized using a structured H-grid comprising 20x35x42 nodes respectively in the streamwise, spanwise and vertical directions as illustrated in figure 6.26. The grid density perpendicular to the walls was chosen as an geometrical expansion serie with factor 1.3 in respect to the finite volume formulation. The size of the first node which is critical in terms of computing resources, viscous sub-layer description and skin friction calculation on the walls, was chosen about $10^{-5}m$ as a good compromise. The resulting dimensionless parameter $y^+$ was calculated around 3.5, which is about the value specified by Wilcox (1993b) in order to use the $k-\omega$ turbulence model.

![Mesh grid in 3D duct](image)

**Figure 6.26: Mesh grid in 3D duct**

Boundary conditions

First, numerical adherent adiabatic boundary conditions were specified on the side walls whereas symmetry was used to treat the boundaries opposite to the side walls and account for symmetries in the geometry.

Secondly, the inlet and outlet planes were treated as free boundaries and aerodynamic primitive variables were specified. However, as the aim of the duct calculation was to simulate the development of the BLs in a wall bounded compressible flow, a uniform flow field profile was specified at both the inlet and outlet. Finally, after a few iterations on the boundary selection, the inlet total pressure and temperature, and outlet static pressure
were set to 160kPa, 303K and 124.65kPa respectively in order to obtain an outlet free stream Mach number of about 0.62.

Free stream turbulence initialization

The free stream turbulent variables, \( k_\infty \) and \( \omega_\infty \), were obtained by using the decay formulation (see equations 6.1) for turbulent level, turbulent kinetic energy and turbulent specific dissipation rate. For a given free stream flow velocity and turbulence level, iterations on the specific dissipation value were performed in order to obtain a low turbulence decay at the outlet of the rectangular duct and match a turbulence level value of about 0.5% measured in the same wind tunnel in different applications (Wiers, 2002; Svensdotter, 1998; Sigfrid, 2003).

\[
\begin{align*}
\text{Tu decay:} & \quad T_u(x) = \sqrt{\frac{2 k_0}{3 T_u^2} \left( 1 + \frac{C_{\omega^2}}{t_0} \frac{\omega_\infty}{x} \right)}^{-\frac{3}{2}} \\
\text{k decay:} & \quad k(x) = k_0 \left( 1 + \frac{C_{\omega^2}}{t_0} \frac{\omega_\infty}{x} \right)^{-\frac{3}{2}} \\
\text{\omega decay:} & \quad \omega(x) = \frac{1}{C_{\omega^2} x^{\frac{1}{2}}} \\
\text{with} & \quad C_k = 0.09 \\
& \quad C_{\omega^2} = 0.075 \\
& \quad k_0 = \frac{1}{2} T_u^2 \lambda_\infty \frac{U_\infty}{2}
\end{align*}
\]  

(6.1)

6.3.2.2 Rectangular duct simulation results

Overall outflow characteristics

The outlet flow plane has been plotted in figure 6.27 for different variables. An approximately 7mm thick BLs have developed on all side walls of the duct and interact with each others in the corner as observed on the streamlines plotted together with the axial velocity. As a result two counter rotating vortices have appeared in the corner but do not seem to generate a high level of turbulent kinetic energy. A region of low density is located very close to the wall and in the corner of the rectangular duct as the flow is incompressible in part of the BL. The BLs seem to have a high momentum as the low velocity region is located very close (less than 1mm) to the wall as observed on the isentropic Mach number distribution.

Boundary layer profiles

Detailed BL profiles are presented in figure 6.28 for different variables. As mentioned above the BL features a high momentum as the axial flow velocity remains at a fairly high value very close to the side walls. The incompressible BL parameters, as defined by equations 2.4 and 2.5, were calculated to be \( \delta = 7.08 \text{mm} \), \( \delta^* = 0.61 \text{mm} \), \( \theta = 0.48 \text{mm} \), which gives a shape parameter of about 1.25. It is interesting to note that the compressible BL parameters, calculated by integrating the density together with the velocity profile, would give a raise to the shape parameter to a value of 1.41, closer to the critical separation value of 2.5.
6.3.3 Two dimensional nozzle flow simulations

6.3.3.1 Introduction

Both 2D and 3D RANS simulations were performed over the 2D nozzle geometry. The objectives were first to perform unsteady simulations using 2D calculations and secondly to determine the similarities and differences obtained using both approaches. Indeed, in the perspective of unsteady 2D RANS calculations, it is important to determine how 2D and 3D calculations respectively differ from experimental results. Besides, the advantage of performing unsteady 2D simulations allows parametric studies which would take a huge amount of time and CPU resources if performed using 3D calculations.

In all following numerical calculations, the fluid is modelled as a viscous perfect gas. The specific heat ratio is \( \gamma = 1.4 \), and the perfect gas constant, \( R \), equals 287 \( J.kg^{-1}.K^{-1} \). The laminar dynamic viscosity and the thermal conductivity are assumed constant and respectively equal \( \mu = 1.81 \times 10^{-5} \ kg.m^{-1}.s^{-1} \) and \( k = 2.54 \times 10^{-2} \ m.kg.K^{-1}.s^{-3} \).

The following subsections aim at presenting both 2D and 3D computations over different operating conditions, as well as the different meshes used. The similarities or differences relative to each approach are also discussed.
6.3. NUMERICAL INVESTIGATION

6.3.3.2 2D RANS computations

6.3.3.2.1 Computational meshes

Four different H-grid meshes, each comprising 150x84 nodes, were created with adapted grid density in order to correctly describe the mean flow gradients, BLs and shock structure in 2D RANS calculations. As a result the upper and lower BL were described using respectively 28 and 33 nodes. Each mesh is presented together with the steady state results in Appendix M, from Figure M.1 to M.4.

Figure 6.28: Outflow boundary layer profile in 3D rectangular duct

Figure 6.29: Dimensionless vertical coordinate $y^+$ for 2D RANS calculation (case with $P_{out}^{*}=110$ kPa)

In order to ensure the grid independency towards the Wilcox two equations turbulent
model, the node distribution normal to the walls was set in respect to the dimensionless $y^+$ coordinate. As a result, the $y^+$ value does not exceed 5 as shown in figure 6.29.

### 6.3.3.2.2 Global flow description

Table 6.5 presents the different operating conditions for 2D RANS simulations of the 2D nozzle.

<table>
<thead>
<tr>
<th>$P_{\text{inlet}}$ [kPa]</th>
<th>$T_{\text{inlet}}$ [K]</th>
<th>$M_{\text{inlet}}$</th>
<th>$P_{\text{outlet}}$ [kPa]</th>
<th>$Q_m$ [kg/s]</th>
<th>Mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>160</td>
<td>303</td>
<td>0.644</td>
<td>120</td>
<td>3.88</td>
<td>2D-NS-1</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.661</td>
<td>118</td>
<td>3.96</td>
<td>2D-NS-1</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.676</td>
<td>116</td>
<td>3.96</td>
<td>2D-NS-1</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.687</td>
<td>114</td>
<td>3.97</td>
<td>2D-NS-1</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.692</td>
<td>112</td>
<td>3.98</td>
<td>2D-NS-1</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.693</td>
<td>110</td>
<td>3.99</td>
<td>2D-NS-2</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.694</td>
<td>108</td>
<td>3.99</td>
<td>2D-NS-2</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.694</td>
<td>106</td>
<td>3.99</td>
<td>2D-NS-3</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.694</td>
<td>102</td>
<td>3.99</td>
<td>2D-NS-4</td>
</tr>
</tbody>
</table>

Table 6.5: Operating conditions for 2D RANS calculations in 2D nozzle

The iso-Mach number contours in the 2D nozzle channel as well as the pressure distribution on the 2D bump surface are presented in Appendix M together with a comparison with experimental pressure measurement results.

A global flow structure description is presented below and focusses on the shock location, the separated region extends and the comparison between experimental and numerical pressure distribution on the 2D bump.

### Shock location

The shock location on the 2D bump surface is plotted in figure 6.30 respectively for experiments and 2D RANS calculations. Clearly, for the same OP, the 2D RANS simulation gives a shock position far downstream (from 5mm at $P_{\text{out}}^{\text{expt}}=118\text{kPa}$ to 30mm at $P_{\text{out}}^{\text{ns}}=106\text{kPa}$) of the experimental shock location. The lack of side wall boundary layers is directly responsible of this observation. Indeed, the presence of side wall BLs creates, in the experimental case, an acceleration of the flow and thus accentuates the pressure gradient in the diffusor. As the outlet static pressure is manually adjusted (in both numerical and experimental cases) the pressure immediately downstream of the shock is higher in the experimental case, which is why the shock is located upstream in the experiments. Besides, although the global trend is similar, the difference between the numerical and experimental shock position increases as the back pressure decreases. This observation results from the increasing thickening of the BL with decreasing back pressures. As the
shock grows and moves downstream, the SBLI gets stronger and all BLs thicken more, eventually until separation occurs which accentuates even more the phenomenon. It is also possible, to a certain extent, that the two-equations turbulent model does not predict the correct level of losses in the BLs and in the SBLI region. However, the explanation could not, by itself, justify the differences observed in the shock location.

Figure 6.30: Shock position - Comparison between experiments and 2D RANS

Separation

The extent of the separated region is plotted in figure 6.31 for experiments and numerical 2D RANS simulations as a function of the back pressure. The upstream and downstream curves represent respectively the location of the separation and reattachment points on the bump surface. It seems clear, by comparing the shock position and the location of the separating point, that it is a shock induced separation. Following the shock behaviour, the separation predicted by numerical simulations always occurs upstream of the experimental location. Accounting for the fact that the presence and the thickening of the side wall BLs have the effect to move the shock location upstream (compared to a case without side wall BLs) the global trends concerning the extent of the separated region seem fairly similar. It seems however that the 2D RANS simulations slightly underestimated the length of the separated region.

Static pressure distribution

The steady state pressure distribution on the 2D bump surface is plotted for all OPs of the 2D RANS calculations in figure 6.32. It is interesting to note that for constant variations of the outlet static pressure, the distance between each successive shock location is not regular. This is basically due to the curvatures and possibly to the presence of separation or a strong SBLI region.
The effect of the separation on the pressure distribution is well seen for the OPs with a back pressure value equal to 108kPa and 106kPa. The pressure distribution curves slightly "flattens" downstream of the shock. This corresponds to a lower pressure gradient and thus a slight local acceleration as the separation suddenly gives a raise to the displacement thickness of the BL. This effect is however much more pronounced in the experimental case as the thickening of the side wall BLs takes part in the process. It is also interesting to note that for very low outlet static pressure (below 104kPa), the separation disappears and the pressure distribution does not show any strong discontinuity any more. This behaviour is very similar to the experimental case although it does not occur at the same OP.

Figure 6.32: 2D RANS steady state pressure distribution on 2D bump

6.3.3.3 3D RANS computations

6.3.3.3.1 Computational meshes

The computational meshes used in the 3D RANS calculations of the 2D nozzle were constructed using two of the 2D meshes presented in the previous subsection. The 2D
meshes were then piled up in the spanwise direction using the same grid density distribution in the BL as for the 2D meshes. Besides, due to the geometrical symmetry, only half of the 2D nozzle was discretized. The two different 3D meshes, comprising each 150x84x35 nodes, are presented in figure 6.33. The number of nodes used to describe the upper, lower and side wall BLs were respectively 28, 33 and 28.

![Figure 6.33: Computational meshes for 3D RANS calculations of the 2D bump](image)

### 6.3.3.3.2 Global flow description

The different operating conditions for the 3D RANS calculations over the 2D bump are presented in Table 6.6.

<table>
<thead>
<tr>
<th>$P_{inlet}$ [kPa]</th>
<th>$T_{inlet}$ [K]</th>
<th>$A_{inlet}$ [-]</th>
<th>$P_{outlet}$ [kPa]</th>
<th>$Q_m$ [kg/s]</th>
<th>Mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>160</td>
<td>303</td>
<td>0.638</td>
<td>118</td>
<td>3.74</td>
<td>3D-NS-1</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.654</td>
<td>116</td>
<td>3.79</td>
<td>3D-NS-1</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.668</td>
<td>114</td>
<td>3.84</td>
<td>3D-NS-1</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.681</td>
<td>112</td>
<td>3.88</td>
<td>3D-NS-2</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.690</td>
<td>110</td>
<td>3.91</td>
<td>3D-NS-2</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.695</td>
<td>108</td>
<td>3.93</td>
<td>3D-NS-2</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.698</td>
<td>106</td>
<td>3.93</td>
<td>3D-NS-2</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
<td>0.698</td>
<td>104</td>
<td>3.93</td>
<td>3D-NS-2</td>
</tr>
</tbody>
</table>

Table 6.6: Operating conditions for 3D RANS calculations in 2D nozzle

The iso-Mach number contours in the 2D nozzle channel as well as the pressure distribution and streamlines over the 2D bump surface are presented in Appendix M together with a comparison with experimental results.

A global flow structure description is presented below and focusses on the shock location, the extent of the separated regions and the comparison between experimental and numerical results on the 2D bump as the back pressure decreases.

**Shock location**

The shock location, estimated form the steady state pressure distribution in the
center of the channel, is presented in figure 6.34 as a function of the outlet static pressure for experiments and both 2D and 3D RANS calculations.

![Figure 6.34: Shock position - Comparison between experiments, 2D RANS and 3D RANS](image)

Clearly, at high outlet static pressure values, the agreement between experiments and 3D RANS calculation is fairly good which confirm the fact that the presence and behaviour of the side wall BLs are important regarding the steady state shock location. As observed between the 2D and 3D RANS calculations, the side wall BL thickening is responsible for the important shift in the shock location. Besides, although the inlet boundary conditions do not exactly correspond to the experimental conditions, a fairly good agreement is observed at high back pressure values. The present observation could have three different meaning. Either the inlet boundary conditions do not have such a large impact as preliminary thought, or their influence is limited as long as the SBLI remains weak, or the numerical inlet boundary conditions obtained from the rectangular duct calculation appeared to be a fairly good estimation of the real inlet conditions. Regarding this last remark, it should be pointed out that, although the BL thickness and displacement thickness were quite different from the experimental case, the BL shape factor calculated from the computed BL in the rectangular duct was very close to the experimental value. As stated by Delery and Marvin (1986) this parameter is the most important regarding SBLI.

However, as the outlet static pressure decreases, the numerical shock location differs from the experimental one. Indeed for a back pressure value below 114kPa, the shock obtained by 3D RANS calculations is systematically located downstream of the experimental shock position. Besides, the difference between the experimental and numerical shock position increases as the back pressure further decreases. Following the same explanation regarding the influence of side wall BLs, it seems as the numerical simulations underestimate the magnitude of the BL thickening when the SBLI gets stronger. Consequently, the sudden increase of all BL displacement thickness and therefore the slight acceleration induced by the reduction of the equivalent section area are both under estimated. As a
result the changes in the pressure gradient are under predicted, which leads to a lower pressure value immediately downstream of the shock and therefore a more downstream shock position.

It should be noted that, from the OP with \( P_{\text{out}} = 110 \text{kPa} \) (at which a significant difference between the experimental and numerical shock position can be observed), a strong interaction takes place between the side wall BLs and the shock as illustrated by a “V” shape of the shock seen on the experimental isentropic Mach number distribution plotted in figure M.13(c). The difference with the numerical isentropic Mach number plotted in figure M.13(d) is obvious and illustrates perfectly the under estimation of the side wall SBLI and the resulting BL thickening.

Separation zones

The location of the separation and reattachment points in the mid-channel separation has been plotted as a function of the back pressure in figure 6.35 for experimental and numerical cases. Similarly to previous results, the comparison between the shock location and the separation point position suggests a shock induced separation. Although the global trends are similar, the numerical calculation clearly under estimate the length of the separation.

![Figure 6.35: Separation region location - Comparison between experiments, 2D RANS and 3D RANS](image)

The streamlines over the 2D bump surface, obtained with 3D RANS simulations, are presented in Appendix M together with oil visualizations performed at the same OP, in order to directly compared the extent of the mid-channel and corner separations. As mentioned above, for back pressure values above 112kPa, the flow structure and shock position from numerical simulations are very similar to the experiments. This situation corresponds to a weak SBLI and only a small corner separation can actually be observed (figure M.11(f)). However, as the back pressure further decreases, the SBLI gets stronger.
and a flow separation appears on the bump in the center of channel (figure M.13(f)). Simultaneously the corner separation grows in size and length. It is interesting to note that, from 112kPa, the structure of the corner separation differs between numerical calculations and experiments. Indeed the 3D RANS simulations predict a double vortex configuration in the corner separation (as seen in figure M.15(f) or M.17(f)) which is not observed on the oil paint visualizations. The flow structure is actually quite complex and involves the interaction between the shock and a 3D flow pattern as observed in figure 6.36.

![Figure 6.36: Streamlines on 3D RANS steady state calculation on 2D bump (P_{out}=108kPa)](image)

In order to analyze complex 3D separated flow patterns, Legendre (1977) introduced a limited number of elementary singularities into the family of skin friction lines of an isolated obstacle in a 3D flow. These points are of two kinds, nodal points and saddle points, but lead to different types of singularities as illustrated in figure 6.37 and explained below:

- **A nodes (of attachment or separation)** is basically a singularity where all the skin friction lines, except one, have a common tangent,
- **An isotropic node** is defined as a particular type of nodal point,
- **A focus (of separation or reattachment)** occurs when there is no common tangent line any longer. An infinity of streamlines spiral around such a point. Usually a focus of separation on a plane surface leads to an ejection of particles perpendicularly to this surface whereas a focus of reattachment leads to an injection of particles.
- **A saddle point** is characterized by a situation where only two skin friction lines can merge into each others. All other skin friction lines seem to avoid this singular point.

As observed in figure 6.36, the interaction between the shock and the side wall BL generates a complex system of vortices with multiples particle ejections and reattachment into the main flow. At this OP ($P_{out}=118kPa$) a set of particles originating the upstream BL flow is convected around the corner separation and impacts directly on the side wall at the isotropic node location noted $IN_{reat}$ in figure 6.36(left). Most of the particles are then convected further downstream but a few are actually trapped in the
6.3. NUMERICAL INVESTIGATION

Figure 6.37: Skin friction elementary singularities (Legendre, 1977)

Recirculation and travel upstream towards the focus of separation ($F_{sep}$) located on the side wall. Spiraling around the node location, the particles are finally ejected from the side wall BL and enter the domain of influence of the vortex which forms the focus of reattachment ($F_{reat}$) on the lower wall. Converted towards the lower wall, the followed particle is thereafter attracted by the vortex originating the focus of separation on the lower wall and is ejected and disappear downstream in the main stream flow.

Static pressure distribution

The steady state pressure distribution in the center of the channel ($y=50\text{mm}$), obtained by 3D RANS calculations over the 2D bump surface, is presented for all OPs in figure 6.32. Similarly to the results obtained by 2D RANS calculations, the effect of the separation on the pressure distribution is well seen at the OP with a back pressure value equal to 108kPa. The pressure distribution curves slightly “flattens” downstream of the shock, which corresponds to a change of the local curvatures due to a slight acceleration as the separation suddenly gives a raise to the displacement thickness of the BL.

6.3.3.4 Experimental-Numerical comparison

Two flow configurations, with different shock structures, have been selected as steady state OP for unsteady flow investigations and a more detailed comparison between experimental and numerical results is presented in this subsection.

6.3.3.4.1 Weak shock configuration case

The steady state shock wave in the 2D nozzle is presented in figure 6.39 for experimental visualization, 3D RANS (at mid-channel plane) and 2D RANS calculations. It should be noted that the outlet static pressure had to be adjusted for the 2D RANS numerical simulations in order to match the experimental shock location on the bump. As already
stated in the previous paragraphs, for the same OP the numerical shock is systematically located downstream of experimental shock position. For the weak shock structure shown in figure 6.39, the difference in back pressure value actually gives an estimation of the numerically underestimated losses. It seems obvious that the 2D RANS calculation presents such a large difference with the experiments since the friction on the side wall and interaction between the shock and the side wall BLs is simply not taken into account. The 3D RANS calculation on the other hand gives a better estimation of the level of the losses at this OP but still does not predict the correct shock position for stronger SBLI.

Using the continuity equation at the outlet ($\dot{Q}_m = \rho_2 V_2 A_2$), the reduction of section area due to BL thickening can be estimated for numerical simulations by calculating the change of section necessary to obtain the experimental mass flow under the same numerical outlet conditions. For the 2D RANS simulation, in which no side wall BL is specified, the change of section area was estimated around 11.15 $\text{cm}^2$, equivalent to a BL with a displacement thickness of 4.64 mm on each side wall (assuming that the upper and lower BL remain the same).

As observed in figure 6.40, although the agreement between experiments and 3D RANS is fairly good, a closer look to the pressure distribution curves reveal that the numerical...
6.3. NUMERICAL INVESTIGATION

calculation slightly under estimates the level of losses as the numerical shock is located about 2mm downstream of the experimental one. Downstream of the shock the pressure distribution for experiments and 3D RANS calculation collapse fairly nicely as the flow does not feature any separated zone yet. Besides, whereas the outlet static pressure value is quite different for 2D RANS calculation, it should be noted that the pressure distribution agrees fairly well to the experiments. A direct comparison of the flow field can be made by looking at figures M.11(e) and M.11(f).

![Steady State Experimental-Numerical Comparison on 2D Bump](chart)

Figure 6.40: Experimental and numerical comparison of pressure distribution on 2D bump (strong shock configuration $P_{out}=112kPa$)

6.3.3.4.2 Strong shock configuration case

Figure 6.41 presents the steady state shock structure for experimental visualization and viscous numerical simulations (at mid-channel plane for the 3D RANS simulation) for a lower back pressure value. The shock has moved downstream and nearly reaches the upper wall. A small lambda shock system has even appeared as seen from Schlieren visualization in figure 6.39(a).

Although a fairly good agreement on the shock location and general flow structure is achieved, the experimental shock position could only be matched by raising up the outlet static pressure value for both numerical simulations this time. Accounting for upper and lower BLs, the reduction of section area due to BL thickening was estimated for the 2D RANS simulation to be around $15.46cm^2$, which is to a side wall BL with a 4.56mm displacement thickness. For the 3D RANS, which already features outlet side wall BLs, the change of section area was estimated around $8.75cm^2$ and is equivalent to an increase of the displacement thickness of 3.63mm on each side wall BL.

Although the outlet static pressure value differ between the experiments and the calculation, the flow structure is fairly similar. The iso Mach number contours as well as the small expansion downstream of the shock are similar for both calculations. However, a direct comparison of the flow structure on the bump surface (figures M.17(e) and M.15(f)) reveals a quite different pattern with an under estimated separation length in the center of the channel and in the corners.
CHAPTER 6. STEADY STATE RESULTS

Figure 6.41: Steady state strong shock structure in 2D nozzle

Figure 6.42: Experimental and numerical comparison of pressure distribution on 2D bump (strong shock configuration $P_{out} = 106kPa$)

The steady state pressure distribution at mid-channel ($y=50mm$) over the 2D bump surface is plotted in figure 6.42 for experimental results and viscous numerical simulations. Although the curves collapse fairly well regarding the shock location, they differ downstream of it. Indeed, experimental results present a smoother pressure recovery, which denotes a change of local curvatures (towards a more convex surface) due to a separated flow region. This phenomenon is even stronger closer to the wall (as seen from pressure distribution at $y=10mm$) and denotes a large BL thickening or a separation of the flow in the corners. Probably due to larger side wall BLs and the interaction with the shock, the pressure rise occurs more upstream in the region close to the side walls.

6.3.4 Three dimensional nozzle flow simulation

6.3.4.1 Computational meshes

Considering the geometrical characteristics of the nozzle, i.e. varying length and thickness of the bump through the channel width, a 3D mesh was created to account for the span-wise evolution of the mean flow gradients and bent shock configuration. The 3D viscous
mesh was created preliminary computations performed by Bron (1997) during the design process of the 3D bump. Although the calculations were performed on a coarse grid using an algebraic turbulent model, the obtained results provided good indications on the high mean flow gradients regions and the extent of the shock displacement for a certain range of outlet static pressure variation. A few iterations between calculations and mesh corrections were thereafter performed in order to adapt the mesh to different steady state solutions. As a result the 3D mesh presented in figure 6.43 comprises 150x84x70 nodes with higher grid density in the BLs and around the shock location. The grid density was adapted to account for BL thickening on each wall. Consequently the upper, lower and side wall BL are described using 22, 30, and 22 nodes at the inlet and 25, 33, and 26 nodes at the outlet respectively. Finally, it should be noted that the mesh was designed to describe steady and unsteady flow behaviour for weak to moderate shock configurations, i.e. at high outlet static pressure value. The mesh is therefore not well adapted for strong shock configuration in which a large lambda shock system is present.

The present paragraph aims at describing the steady state flow field in a particular configuration, namely for a weak shock configuration, corresponding to a back pressure value of 116kPa. This configuration was chosen because of its close similarity with the ideal configuration presented in figure 5.4 in the experimental model chapter. Besides, for higher and lower back pressure value, namely 118kPa and 114kPa as shown in figure M.29 and M.33, the sonic pocket covers either part or the entire width of the channel.
steady state solutions represent the extreme quasi-steady configurations of an unsteady calculation with a downstream perturbation of amplitude ±2 kPa, which is a priori the configuration of interest, and should therefore be referred to while analyzing the unsteady results.

For the present weak shock configuration, the flow structure obtained with 3D RANS calculation is fairly similar to the experiments as seen in figure M.31 where a direct comparison is possible. The numerical simulation, however, allows to deepen the analysis on the flow field, visualize precisely the extent of the sonic pocket and shock wave (figure 6.44 left) and further understand the structure of the separated region. It is noteworthy that the sonic pocket actually does not reach the low curvature side wall as shown in iso-Mach number contours in figure M.31(d). Indeed, as mentioned previously, the assumption to use the inlet stagnation pressure as total pressure in the isentropic formula is not correct any longer downstream of the shock wave. As a result, the extent of the sonic pocket (and thus shock location) as observed in Figure M.31(d) is over estimated. Figure 6.44 (right) presents the pattern of the separated flow downstream of the shock described with the elementary singularities introduced by Legendre. A brief analysis of the streamlines pattern is proposed to better understand the behaviour of flow field within the separated region. First it is interesting to note that a set of particles originating the upstream flow very close to the corner goes round the separation area and directly impact on the isotropic node of reattachment \( (J_N_{reat}) \). From this point, according to the streamline pattern, some particles are convected downstream whereas some others are trapped in the recirculation zone and are convected back upstream. Among those latter particles, some travel far upstream and are re-injected into the main flow stream whereas other are directed towards the focus of separation \( (F_{sep}) \) and are ejected from the side wall into the free stream. This injection of particles seems to participate in the creation of a large vortex which convect flow particles towards the focus of reattachment \( (F_{reat}) \) on the lower wall. Particles trapped in this large vortex are then convected towards the lower wall and thereafter spiral on the lower wall, around the center of the vortex, towards the outside, and are eventually re-injected into the main flow. It also seems more likely that this large vortex originates the shear flow which seen on the side wall, at the saddle point \( (S) \) location.
6.3. NUMERICAL INVESTIGATION

6.3.4.3 Influence of outlet static pressure

Figure 6.47 shows the evolution of the overall flow field over the 3D bump as the outlet static pressure decreases. More precisely, the extent of the sonic pocket and streamlines in the lower wall and side wall BLs are presented and the 3D flow pattern within the corner separated region is described using Legendre elementary singularities.

For high back pressure values (between 118kPa and 114kPa) the solution obtained by 3D RANS calculation is very similar to the experiments concerning the shock location and shape (as seen from the pressure distributions at different spanwise locations in figures M.30, M.32 and M.34 and the isentropic Mach number distributions shown in figures M.29, M.31 and M.33), as well as the location, extent and flow pattern of the corner separation (by comparison between oil paint visualizations and streamlines from 3D RANS in figures M.29(e), M.31(e) and M.33(e)). Especially, the oil visualization performed in the corner separation region at $P_{\text{out}}^*=114$ kPa and presented in figure 6.45 shows a very similar flow structure than the one described in the previous paragraph. From a global perspective, for high back pressure, although the sonic pocket grows and extends both downstream and spanwise towards the opposite side wall, the structure for the flow field within the separated region does not change drastically and is fairly similar to the description given previously.

For lower outlet static pressure values (from 112kPa and below), the difference between 3D RANS calculations and experiments becomes important and the overall flow structure differs in many aspects. As observed on pressure distributions in figures M.36 to M.44, the numerical prediction of the shock location differs from experimental measurements or visualizations to such a large extent that no direct comparison can be performed any longer. Some similarities however seem obvious and are commented during the following description of the flow structure, obtained by 3D RANS simulations. As the back pressure decreases, the sonic pocket grows in size and extends both upstream and downstream. A complex flow separation then appears in the middle of the channel (at $P_{\text{out}}^*=112$ kPa) and quickly extends both downstream and spanwise as the 3D bend shock moves downstream.
The flow pattern within the corner separated region also evolves and the elementary singularities described previously tend to migrate downstream whereas new singularities appear and interact with each others. More precisely, a new focus of separation \( F_{\text{sep}} \) first appears at \( P_{\text{out}} = 112 \text{kPa} \) (figure 6.47(d)) on the side wall immediately downstream of the shock and probably results from the interaction of the sudden thickening of the BL and main flow field. Then, for a back pressure of 112kPa, a second focus of separation appears next to the first one (figure 6.47(e)) and both thereafter merge into each others as observed in figure 6.47(f) at \( P_{\text{out}} = 110 \text{kPa} \). At the same time, the lower wall recirculation has quickly extended downstream and towards the low curvatures side wall and a clear similarity with experimental visualizations at \( P_{\text{out}} = 104 \text{kPa} \) (figure M.43(e)) can be observed regarding the reversed flow pattern. It is also interesting to note the formation of a saddle point (S) and a focus of reattachment \( F_{\text{reat}} \) on the lower wall, close to the low curvature side wall, as a result of the interaction between the free stream and the reversed flow at \( P_{\text{out}} = 108 \text{kPa} \) (figure 6.47(g)). For very low outlet static pressure values, namely 106kPa and 104kPa, a vortex and focus of separation have appeared on the low curvature side wall and result from the SBFI as the shock at this location is also getting strong. It is noteworthy that a supersonic tongue, as described by Seddon (1976) and further experimentally investigated by others (Kooi, 1978; East, 1976; Abiss et al., 1976), can be observed close to the high curvatures side wall.

Finally, in order to give an idea of the flow structure within the separated region, the trajectory path of a fluid particle is presented and described in figure 6.46 for an outlet static pressure value of 106kPa. A fluid particle originating the upstream flow \( \text{(1)} \) close to the high curvature side wall is transported downstream directly above the corner separation. However, the particle suddenly turns back while approaching the saddle point location on the side wall \( \text{(2)} \) and is convected upstream. This is due to the fact that the path of the particle is located underneath the skin friction line on which is located the saddle point. If the particle would have been located only a few millimeters above this line, it would have been convected further downstream in the diffusor. Trapped within the recirculation zone close to the side wall, the particle travels upstream to a location, downstream of the shock, where the side wall BL thickens \( \text{(3)} \). Following the deflection
6.3. NUMERICAL INVESTIGATION

Figure 6.46: 3D path for flow particule within separated region at $P_{out} = 106kPa$

do the BL flow, the particule is convected farther from the side wall but does not escape from the domain of influence of the separated flow and is now "caught" by the vortex originating the shear flow around the saddle point $\text{④}$. The particule thereafter spirals down towards the focus of reattachment located on the lower wall $\text{⑤}$ and is transported back upstream by the reversed flow towards the focus of separation $\text{⑥}$. Coming close to the vortex originating the focus of separation, the particule spirals up $\text{⑦}$ until it is caught again in the recirculating flow $\text{⑧}$. When the particule is finally convected downstream, apparently outside of the reversed flow region, it is suddenly attracted by another vortex, seems to spiral down to the lower wall $\text{⑨}$ and heads towards the focus of reattachment located on the lower wall close to the low curvature side wall $\text{⑩}$.

6.3.5 Conclusion

This chapter was dedicated to the description of the steady state results on both the 2D and 3D nozzle geometries. Experimental results obtained from various measuring techniques were analyzed and compared with each others. Simultaneously, 2D and 3D RANS calculations were performed on the same configurations. The evolution of different flow structures was described as a function of the shock position.

Results on the 2D bump showed that the flow remains quasi two-dimensional over 80% of the channel width as long as the shock is weak. However when the shock reaches a certain streamwise position, the superposition of the strong adverse pressure gradient and the curvatures gives a rise the boundary layer thickness, which initiates the formation of a lambda shock system and simultaneously triggers a separation of the flow. The interaction between the shock and the side wall boundary layers is then strong enough to create large vortices which contribute to the thickening of the corner separation and narrow the two-dimensionality of the flow to a 20% large strip in the middle of the channel. Despite a mismatch of inlet boundary conditions, comparison with numerical simulations showed a fairly good agreement for weak shock configurations but also a clear under-estimation of the losses for strong Shock Boundary Layer Interaction. The comparison between 2D and 3D RANS calculations revealed that the thickening of the side wall boundary layers was
CHAPTER 6. STEADY STATE RESULTS

essential in the shock position determination but numerically still largely under-predicted.

Results on the 3D bump showed similar trends with even more pronounced consequences as the three-dimensional geometry induces higher mean flow gradients and larger separations. Although the flow structures were qualitatively very similar for weak shock configurations, as soon as the shock gets stronger, numerical and experimental results differ in such an order of magnitude that no direct comparison is possible any more. The under-estimation of the boundary layer thickening and extent of separated region is, again, believed to be the main origin of such difference.
6.3. NUMERICAL INVESTIGATION

Figure 6.47: Evolution of sonic pocket and streamlines within separated flow region with decreasing back pressure
Chapter 7

Unsteady Results and Discussion

7.1 Introduction

The chapter is dedicated to the presentation and analysis of unsteady results obtained from high frequency pressure measurements, conventional Schlieren visualizations, as well as both 2D and 3D RANS simulations.

The first part of this chapter is dedicated to the unsteady investigation in the 2D nozzle. Experimental results from different operating conditions are analyzed and compared with each others in order to establish the respective influence of the downstream pressure perturbations and the mean shock location. A similar analysis is thereafter performed on numerical results and a comparison with experimental is finally conducted to ensure the validity of the analysis.

The second part of this chapter concerns the investigation of the 3D nozzle and presents, similarly, experimental and numerical results first analyzed separately and thereafter compared with each others.

7.2 Two-dimensional nozzle

7.2.1 Experimental results

7.2.1.1 Introduction

This section presents the experimental unsteady results including both conventional Schlieren visualizations of the shock motion over the height of the channel and unsteady pressure measurements on the surface of the 2D bump through the entire width of the test section. The different unsteady operating conditions were presented in the experimental model (chapter 5) in Table 5.9 and 5.14 for the pressure measurements and Schlieren visualizations respectively. As a summary, two steady-state operating conditions (weak and strong shock configuration) were investigated at four different perturbation frequencies. Furthermore, the OP with a strong shock configuration was also investigated for a lower amplitude of perturbation.
In the following paragraphs, the analysis has been structured in order to emphasize on the parametric influence of the mean shock wave position, as well as the frequency and amplitude of the outlet pressure perturbations. Rather than to analyze each curve individually, results have been presented in a systematic way by comparing data obtained for different parameters, depending on the parametric study being performed.

7.2.1.2 Analysis of unsteady pressure distribution at mid-channel

7.2.1.2.1 Quasi-steady description of the streamwise unsteady pressure repartition

Evolution of unsteady pressure amplification

Figure N.2(e) presents the amplitude of the dimensionless unsteady pressure distribution measured on the 2D bump surface in the middle of the channel (y=50mm) for the strong shock configuration. Considering the low amplitude level of higher harmonics and the accuracy on unsteady pressure measurements, only the fundamental and first harmonic of the perturbation frequency ($F_p=50\text{Hz}$ in the present case) were considered relevant and plotted. Besides, it should be reminded that the unsteady pressure distribution on the bump was normalized by the amplitude of the fundamental of the outlet static pressure fluctuations in order to obtain the amplification factor relatively to the perturbation introduced, and to enable a comparison between different amplitude of perturbations. Furthermore, the steady state pressure distributions for both extreme rod positions (vertical and horizontal) are also plotted in order to locate the respective upstream and downstream shock positions. As mentioned those positions would correspond to quasi-steady extreme positions of the shock, that is to say for very low frequency. However, as the frequency increases, the amplitude of motion of the shock wave is expected to decrease and the above positions do not exactly correspond to the extreme shock positions any longer, but they still give an idea about the shock location.

Due to physical restrictions on the test object, the instrumentation could only be placed between $x=-35\text{mm}$ and $x=185\text{mm}$. Nevertheless, the pressure amplification level at $x=185\text{mm}$ is about 1 for the fundamental, which means that the downstream pressure perturbations have propagated from the outlet location ($x=290\text{mm}$) to this location without any damping nor amplification. The upstream propagating perturbations are then amplified up to factor 3.6 between $x=180\text{mm}$ and $x=85\text{mm}$. It is noteworthy that the pressure amplification occurs outside the domain of motion of the shock. Anticipating forthcoming observations and comparisons with numerical calculations, this amplification is not due to the curvatures but rather to viscous or turbulent effects. Furthermore, a sharp and high level amplification can be observed underneath the shock location but simply corresponds to the periodic motion of the pressure rise (diffused discontinuity) through the BL. It is interesting to note the decrease of the pressure amplification between the peak underneath the shock location and the maxima at $x=85\text{mm}$. It is not clear what the attenuation of amplification corresponds to, but it should be noted that the BL strongly thickens and separates close to this location. Finally, the amplification level further upstream of the shock location is extremely small, almost negligible. This observation simply corresponds to the fact that downstream pressure perturbations are
7.2. TWO-DIMENSIONAL NOZZLE

blocked by the sonic flow region and do not propagate upstream of the shock location. As a matter of fact, some perturbations do propagate upstream through the subsonic flow part of the BL, but are rapidly damped in the shear flow.

The amplitude of the first harmonic, which represents a non-linear behaviour of the unsteady pressure distribution, also shows a slight amplification downstream of the shock location. Although the level of amplification for the first harmonic is not very high (value of 0.4 at \( x=100 \text{mm} \)), still it represents 40\% of the outlet perturbation. It should also be noted that the location of the maximum pressure amplification of the first harmonic does not coincide with the maximum level for the fundamental, which suggests, a priori, a different origin for each respective amplification.

**Evolution of unsteady pressure phase-angle**

Similarly, Figure N.2(f) presents the phase-angle distribution of the unsteady pressure measured on the 2D bump surface in the middle of the channel (\( y=50 \text{mm} \)) for the strong shock configuration. It should first be reminded that the outlet pressure signal on the side wall, at the location (\( x=290 \text{mm}, y=50 \text{mm} \) and \( z=60 \text{mm} \)), was taken as a reference\(^1\). Therefore, the phase-angle distribution for the fundamental and all harmonics was corrected by taking away the outlet reference value for each respective mode. As a result the phase-angle value plotted in Figure N.2(f) should be zero at \( x=290 \text{mm} \) for both the fundamental and the first harmonic. However, by prolongating the curves between \( x=185 \text{mm} \) and \( x=290 \text{mm} \) it seems as the phase-angle value is largely above zero. This is due to the fact that outlet unsteady pressure measurements were performed on both side walls whereas the rest of the measurements were performed on the 2D bump, i.e. on the lower wall. Although no difference was observed between the two pressure signals on the side walls, it is believed that due to the vertical non uniformity of the outlet flow field (higher velocity close to the upper wall) the downstream perturbations do propagate with a varying speed between the upper and lower walls and that a progressive phase-lag therefore exists between those locations. This results in an offset which is dependent on the outlet flow non uniformity (thus the OP), and the perturbation wavelength.

The phase-angle distribution for the fundamental varies linearly with the streamwise location. Indeed, from the most downstream measurement location (at \( x=185 \text{mm} \)) the phase-angle decreases regularly until the location \( x=150 \text{mm} \), which is typical of a single travelling plane wave with velocity \( c-U \). A change of slope would therefore mean a change travelling speed within the assumption of a single travelling plane wave. It is very interesting to note that the phase-angle does not decrease any longer at \( x=100 \text{mm} \) and even increase between \( x=70 \text{mm} \) and \( x=100 \text{mm} \). This latter behaviour basically corresponds to a plane wave travelling in the other direction and therefore suggests a more complex acoustic composition with upstream and downstream propagating waves with different amplitudes and phase-angle distributions. At this frequency, and for an mean flow velocity around 220m/s, the typical wavelength is about 2.4m and only plane wave can propagate within the channel. For higher perturbation frequencies the wavelength is

\(^1\)The pressure signals from both side walls were also compared with each others to ensure the validity of the reference pressure measurement.
smaller and multiple waves can propagate in the vertical or spanwise directions, making the physic of the interaction and the analysis much more complex.

Beside, a phase jump or phase-angle discontinuity can be observed in between the extreme shock positions for both the fundamental and the first harmonic. This phase shift can be critical regarding stability if the present unsteady pressure distribution is transposed onto an oscillating airfoil. Indeed, in such case the overall moment coefficient would be directly proportional to this phase shift underneath the shock and lead to an unstable situation over a range of phase shift values. The evolution of the phase-angle underneath the shock location is therefore of great interest in the present study and a parametric investigation will help understanding the different influences and trends.

Finally, the phase-angle distribution further upstream of the shock location is almost constant. This situation however does not correspond to anything physical since the amplitude of the unsteady pressure distribution is nearly zero at this location.

7.2.1.2.2 Influence of perturbation frequency

Evolution of unsteady pressure amplification

The experimental pressure amplification distribution for the same OP (strong shock configuration) but different frequencies of perturbation are presented in Figures N.2-N.8 (sub-figures e). The main modification of the unsteady pressure distribution, as the frequency increases, can be observed downstream of the shock location. Indeed, as described above for a perturbation frequency of 50Hz, the pressure perturbations are amplified while approaching the shock location and slightly damped immediately downstream of it. As the perturbation frequency increases, a reduction of the amplification level can first be observed for 100Hz, but more interesting, an unsteady pressure attenuation appears and counter acts the amplification peak downstream of the shock. It is also interesting to note that the location where the maximum attenuation occurs is getting closer to the shock as the frequency increases (at x=160, 125 and 90mm at the respective perturbation frequencies of 100, 250 and 500Hz). Although it is tempting to associate such attenuation to the presence of steady state flow structures (like separation or reattachment points) or the unsteady behaviour of the flow (oscillations of the reattachment point with possible phase-lag for instance), no clear correlation could yet be found.

The influence of the perturbation frequency on the amplitude of the first harmonic of the unsteady pressure distribution is somehow similar to the one described above. A slight attenuation can be observed at x=135, 110 and 80mm for $F_p=100$, 250 and 500Hz respectively. The particularity of amplitude of the first harmonic however is that a slight amplification (with a level of about 0.2-0.3) corresponds exactly to the amplitude attenuation of the fundamental.

Evolution of unsteady pressure phase-angle

The influence of the perturbation frequency on the phase-angle distribution can be seen in Figures N.2-N.8 (sub-figures f). As discussed in the previous paragraph, for very
low frequencies, the phase-angle distribution is linear from the outlet up to a location around \(x=150\text{mm}\), which corresponds to a single upstream travelling plane wave. However, between \(x=150\text{mm}\) and the shock location, the phase-angle distribution experiences a change of slope, which, in the assumption of a single plane wave, would correspond to a downstream propagating perturbation. As the frequency increases (first at 100\(\text{Hz}\)) this change of tendency further accentuates but the trends remain similar. For higher frequencies, however, the phase-angle distribution does not correspond to a single travelling plane wave and features a rapid phase change like for instance at \(x=120\text{mm}\) for the perturbation frequency of 250\(\text{Hz}\). At 500\(\text{Hz}\) the unsteady pressure phase-angle distribution on the 2D bump is even more complex and features a phase jump at \(x=95\text{mm}\) and a rapid change of the slope just downstream of the shock. It is interesting to note that those, more or less, strong variations in the phase distribution occur outside the domain of motion of the shock wave and thus correspond to an interaction the upstream propagating pressure perturbations and the steady state flow. Anticipating a forthcoming observation based on the comparison with numerical calculations, it seems indeed that the unsteady pressure distribution corresponds to the superposition and interaction of upstream and downstream propagating pressure waves with varying amplitude and phase distributions. An attempt to decompose the acoustic field will be presented and further discussed in the next sub-section related to numerical calculations.

### 7.2.1.2.3 Influence of shock location

#### Evolution of unsteady pressure amplification

A general trend about the influence of the shock location can be established for experimental results by comparing the pressure amplification distribution for two different OPs. Figures N.10-N.16 (sub-figures e) present the amplitude of the fundamental and first harmonic for a weak shock configuration and should be compared with Figures N.2-N.8 (sub-figures e) previously analyzed. The difference is indeed quit striking since almost no pressure amplification can be observed downstream of the shock location. A slight amplification however appears at high frequency but is absolutely not similar, both in shape and amplitude level, to the one described in the previous sub-section.

The pressure amplification peak on the surface underneath the shock location is however much higher and wider, which means a larger shock motion and somehow a thinner incoming BL. Indeed the fact that the peak for the fundamental reaches a level of 9 corresponds to a sharper discontinuity and thus a less diffused pressure gradient due to a thinner BL. On the other hand, the larger amplitude of motion of the shock wave is probably due to the main flow gradient configuration at this OP since the shock is weak and located around \(x=55\text{mm}\), i.e. close to the throat of the 2D nozzle.

The pressure amplification for the first harmonic does not seem to be influenced much by the mean shock location and corresponds downstream of it to approximatively 30\% to 40\% of the outlet amplification level.

#### Evolution of unsteady pressure phase-angle
CHAPTER 7. UNSTEADY RESULTS AND DISCUSSION

Figures N.10-N.16 (sub-figures f) present the unsteady pressure phase-angle distribution for the weak shock configuration. The comparison with Figures N.2-N.8 (sub-figures f) is also striking since, except in the vicinity of the shock location, the phase-angle distribution does not feature any sudden jump or inversion of the slope and therefore correspond better to the assumption of a single upstream propagating plane wave. It should be noted that, at this OP, the sonic pocket occupies only 25% of the channel height and that upstream propagating pressure disturbances are not completely blocked by the shock and partially propagate further upstream. This fact seems to have a fairly important impact on the phase-angle distribution.

The phase-angle distribution for the first harmonic, on the other hand, presents a more complex behaviour but is subject to discussion since the amplitude of the signal is very small and the phase-angle distribution has therefore no real meaning.

7.2.1.2.4 Influence of perturbation amplitude

Evolution of unsteady pressure amplification

Figures N.18-N.24 (sub-figures e) present the amplitude of the fundamental and first harmonic for the same strong shock configuration as previously described but with a lower perturbation amplitude. In this case, a different elliptical rod created a quasi-steady outlet static pressure variation of \( \pm 1.14 \text{kPa} \), compared to \( \pm 2.12 \text{kPa} \) for the previously used rod, thus approximatively half of the previous perturbation amplitude. However, a Fourier decomposition of the outlet reference pressure signals revealed that the amplitude of the fundamental were very similar for the two different rods used. As a result the "effective" amplitude of perturbation (corresponding to the respective amplitude of the fundamental) was lower than expected for the present parametric study.

The comparison with Figures N.2-N.8 (sub-figures e) showed the very same behaviour of the pressure amplification distribution with a slightly lower order of magnitude however. The same amplification and attenuation behaviour as discussed at the beginning of this subsection for the strong shock configuration could be observed at the very same locations, which leads to the conclusion that the amplitude of perturbation does not have any effect on the shape of the pressure amplification distribution but only a slight effect on the level of amplification.

Evolution of unsteady pressure phase-angle

Similarly, Figures N.18-N.24 (sub-figures f) present the phase-angle distribution for the strong shock configuration with a lower amplitude of back pressure perturbation and should be compared with Figures N.2-N.8 (sub-figures f). As a result, the very same observation can be made for the phase-angle distribution. The amplitude of perturbation does not have any effect on the phase-angle distribution.

Finally, it seems that the amplitude of perturbation has a very minor influence compared to the respective effects of the perturbation frequency or the mean shock location.
7.2. TWO-DIMENSIONAL NOZZLE

Concluding remarks

This sub-section was dedicated to the description of the unsteady pressure amplification and phase-angle distribution at mid-channel ($y=50\text{mm}$), as well as the determination of the influence of the mean shock location, and the perturbation amplitude and frequency.

The perturbation frequency was found to strongly modify both the pressure amplification and phase-angle distributions. An important amplification observed downstream of the shock wave at low frequency evolved into a pressure attenuation at higher frequency. The regularly decreasing phase-angle distribution, typical of a single upstream travelling plan wave, became fairly complex with increasing frequency and featured phase jumps and slope variations outside the shock motion extent.

The unsteady pressure distribution (amplification and phase-angle) was also found to depend on the shock location or actually rather on the intensity of the SBLI. For a weak shock configuration where the shock does not block the entire channel, the pressure amplification was nearly non-existent downstream of the shock, whereas an important amplification occurs in the case of a strong SBLI with nearly choked nozzle flow and a large shock induced separation. Concerning the phase-angle distribution, the influence of the mean shock location seems coupled to the perturbation frequency. Indeed, at low frequency, the shock location does not seem to have a large influence on the phase-angle distribution. On the contrary, at high frequency, the shock location have a huge influence and a strong shock configuration revealed a fairly complex phase-angle behaviour with important phase shifts.

The amplitude of perturbation, on the other hand, does not seem to have any influence except on the level of pressure amplification, which seems to be directly proportional to the amplitude of the outlet pressure perturbations. No other influence of the perturbation amplitude was however observed.

Side wall influence on unsteady pressure distribution

Basic quasi-steady influence of spanwise location

Evolution of unsteady pressure amplification

The pressure amplification distribution over the 2D bump for the strong shock configuration is plotted for the three first harmonic contributions separately in Figures N.1 (sub-figures c, e, g). Figures N.2 (sub-figures a,c,e,g,i) also present a 2D plot of the pressure amplification at specific spanwise locations. As observed, the level of the pressure amplification peak downstream of the shock (at $x=85\text{mm}$), already described in the previous sections, slightly decreases for the spanwise location $y=25\%$ and $y=75\%$. However another pressure amplification peak has appeared close to the side walls ($y=10\text{mm}$) and is located further downstream (at $x=110\text{mm}$) than at mid-channel. It should also be noted that the pressure amplification at $y=10\text{mm}$ decreases between the peak and the
shock location (x=60mm). It is not yet clear why a pressure amplification appears at this location but it is interesting to note, from steady state results at this spanwise location, that a large corner separation appears from x=75mm to x=130mm and that the center of the corner vortex is precisely located at x=110mm. It is also interesting to note that the region between the shock and the pressure amplification peak features, in the steady state flow field, a strong BL thickening, both in the middle of the channel (between x=70mm and x=85mm) and close to the side wall at y=10mm (between x=70mm and x=110mm). Despite the correlations between unsteady pressure amplification and steady state flow structures, the mechanism of the observed pressure amplification is not yet clear. Besides, although the unsteady pressure distribution seems fairly symmetrical around the middle of the channel, the pressure amplification at y=10mm and y=90mm features some small differences probably due to the different incoming side wall BLs as already noticed in the steady results chapter.

Evolution of unsteady pressure phase-angle

The spanwise evolution of the phase-angle distribution is presented for the shock configuration at low perturbation frequency in Figures N.2 (sub-figures b, d, f, h and j). Clearly, the different phase-angle distributions throughout the width of the channel look alike. Both the fundamental and first harmonic contribution feature the same streamwise phase evolution. The only observable difference concerns the location of the phase shift underneath the shock, observed slightly upstream close to the side wall due to the "V" shape configuration of the steady state shock.

7.2.1.3.2 Influence of perturbation frequency on spanwise pressure amplification

Evolution of unsteady pressure amplification

Figures N.2-N.8 (sub-figures a,c,e,g,i) present, for the strong shock configuration, the spanwise evolution of the pressure amplification for different perturbation frequencies. Basically the observations made at low frequency are also valid at higher frequencies. The different pressure amplifications and attenuations, previously described at mid-channel, can be observed as well at other spanwise locations. The pressure amplification downstream of the shock seems directly influenced by the spanwise evolution of the steady state flow field. Especially the steady state "V" shape of the shock (more upstream shock position and longer pressure recovery close to the side wall) seems directly correlated to the location of the pressure amplification peaks and phase shift underneath the shock. The influence of the perturbation frequency on the spanwise distribution of the pressure amplification seems therefore rather limited.

Evolution of unsteady pressure phase-angle

Similarly, Figures N.2-N.8 (sub-figures b, d, f, h, j) present the spanwise evolution of the unsteady pressure phase-angle for different perturbation frequencies. As already mentioned for low frequency, the influence of the spanwise location on the phase-angle
distribution is very limited and seems to rather depend on the side wall effects on the steady state flow field. For higher frequencies this observation is still correct but seems to induce larger changes on the phase-angle distribution. At 500Hz for instance (Figure N.8b), the difference between the mid-channel and side wall phase-angle distributions is much more important than at low frequency (Figure N.2b). This magnified effect with the perturbation frequency is probably related to the shorter wavelength of the upstream propagating perturbations. It should be noted that those differences are limited to the region immediately downstream the shock location and that, further downstream, the phase-angle distribution is not influenced either by the spanwise location nor by the perturbation frequency.

7.2.1.3.3 Influence of shock location on spanwise pressure amplification

Evolution of unsteady pressure amplification

Figures N.10-N.16 (sub-figures a,c,e,g,i) present, for the weak shock configuration, the spanwise evolution of the pressure amplification for different perturbation frequencies. At this OP, where the mean flow is quasi-2D over 80% of the channel and does not feature any mid-channel separation but just a side wall BLs thickening, the spanwise location seems to have a strong influence on the unsteady pressure amplification distributions. Indeed, the pressure amplification level increases downstream of the shock as the spanwise location gets closer to the side wall (Figure N.10a). This result is really surprising for this OP which, again, features a quasi-2D steady state flow field and does not have any large corner separation. However, a possible explanation for this behaviour might be related to the side wall BL thickening but the mechanisms of the observed pressure amplification are still unclear.

As the perturbation frequency increases, for the same OP, the influence of the spanwise location on the pressure amplification is similar to the one described above. A pressure attenuation can even be observed close to the side wall (y=10mm) at 500Hz (Figure N.16a).

Evolution of unsteady pressure phase-angle

The unsteady pressure phase-angle distribution for the weak shock configuration is presented in Figures N.10-N.16 (sub-figures b,d,f,h,j) for different spanwise locations and perturbation frequencies. At low frequency, the variations of the phase-angle distribution through the spanwise direction are apparent but very small. As the frequency increases, the evolution of the phase-angle distribution between the mid-channel and the side wall becomes more important. For instance, the difference between the phase-angle distribution at y=50mm and y=10mm at 250Hz (Figures N.14f and N.14b respectively) is much larger than at 50Hz (Figures N.10). Although it is not clear what, in the spanwise direction, makes those important changes, it is interesting to note that the phase-angle distribution close to the side wall at high frequencies is very similar to the distribution at mid-channel with the strong shock configuration. Especially some phase jumps downstream of the shock can be observed at x=85mm on the weak shock, at y=250mm and Fp=250Hz. The side wall BL thickening is however believed to be related to the observed
7.2.1.3.4 Influence of perturbation amplitude on spanwise pressure amplification

Evolution of unsteady pressure amplification

Figures N.18-N.24 (sub-figures a,c,e,g) present, for the strong shock configuration with a lower amplitude of perturbation, the spanwise evolution of the pressure amplification for different perturbation frequencies. First the spanwise evolution of the pressure amplification is very similar to the one observed in Figure N.2. Indeed, for different spanwise location, the pressure amplification level slightly decreases and the peak location slightly moves downstream. The effect of the perturbation amplitude is therefore negligible. For higher frequencies, the similar conclusion can be drawn since the spanwise evolution of the pressure amplification distribution is similar to the one with a lower amplitude of perturbation.

Evolution of unsteady pressure phase-angle

Similarly, Figures N.18-N.24 (sub-figures b,d,f,h,j) present the spanwise evolution of the unsteady pressure phase-angle for different perturbation frequencies. For each respective perturbation frequency, the phase-angle distribution is nearly identical throughout the width of the channel. The perturbation amplitude has therefore almost no influence on the phase-angle distribution.

7.2.1.3.5 Concluding remarks

This sub-section was dedicated to the investigation of the spanwise location and side walls influence on both the unsteady pressure amplification and phase-angle distributions.

It should be noted that the spanwise evolution of the unsteady pressure distribution is significantly affected by the interaction between the side wall BLs and the shock. This interaction can clearly modify the pressure amplification or phase-angle distribution depending on the strength of the shock or the intensity of the BL thickening. It was for instance observed that non symmetrical BLs give completely different phase-angle distributions close to the side walls (comparison Figure N.2, sub-figures b and j).

The perturbation frequency does not have any particular influence on the spanwise evolution of the unsteady pressure distribution but rather seems to accentuate the already existing trends due to the interaction between the shock and the side wall boundary layer.

Finally, the amplitude of perturbations does not seem to have any influence on the shape of the pressure amplification or phase-angle distributions. A minor influence was however observed regarding the level of pressure amplification downstream of the shock.
7.2.1.4 Analysis of unsteady shock motion

7.2.1.4.1 Influence of mean flow gradients

Evolution of shock motion amplification

Figure N.1(i) presents the amplitude of the unsteady shock motion at low frequency (50Hz), for several channel height locations. As described in the chapter on steady state results, the curvatures create a local acceleration over the bump and therefore a velocity gradient between the lower and upper walls. This results in analyzing the amplitude of the shock motion versus the mean flow gradients in the vertical direction. As observed in Figure N.1(i) the amplitude of the fundamental varies from ±5mm above the bump surface (z=12mm) to ±22mm at z=65mm. This increase in the shock motion amplitude is strictly due to the mean flow gradients. Low mean flow gradients tend to excite (or magnify) the shock motion whereas high mean flow gradients play a more stabilize role. At the top of the shock, around z=55mm, the amplitude suddenly strongly increases due to the presence of very low mean flow gradients. Indeed at this location the upstream sonic line and the shock wave meet and the mean flow gradient is ideally zero. The amplitude of the first harmonic also increase from ±1mm to ±8mm at z=65mm. It is noteworthy that the first harmonic contribution represents 20% of the linear (fundamental) motion at z=50mm for instance.

Evolution of shock motion phase-angle

Figure N.1(j) presents the phase-angle distribution of the shock motion throughout the channel width. It is interesting to note that, for the fundamental, the phase-angle varies from 120° above the bump surface to 100° at z=60mm. Thus, a variation of only 20° over the full length of the shock wave. This corresponds to a rigid body shock motion. The shock oscillates with a very small phase-lag between the foot and the top of the shock. It is also interesting to note that the this phase lag corresponds to a small delay of the top of the shock compared to the foot part of the shock as the phase-angle decreases with the height of the channel. A similar analysis can be made as well for the first harmonics with the particularity however that the phase-angle seems to further decrease towards the top of the shock. On the contrary, for the second harmonic, the phase-angle increases towards the top of the shock which corresponds to an advance of phase motion at the top of the shock. This result has however very little effect since the amplitude of the second harmonic is very small.

7.2.1.4.2 Influence of perturbation frequency

Evolution of shock motion amplification

Figures N.1-N.7 (sub-figure i) present for the strong shock configuration the amplitude of the unsteady shock motion at several perturbation frequencies. Clearly the amplitude of the shock motion decreases with the frequency. Figure 7.1 presents the amplitude and phase-angle of the fundamental component of the shock motion, for the strong shock configuration, as a function of the perturbation frequency. As observed for the two locations in the channel height, the amplitude does not exactly decrease linearly
with the frequency. Unfortunately more measurements at other perturbation frequencies would be required in order to draw a clear tendency or enlighten frequencies for which the amplitude of motion is excited or damped.

![Experimental shock motion for strong shock configuration and high perturbation amplitude (\(A_p = \pm 2.12\text{kPa}\))](image)

**Evolution of shock motion phase-angle**

Figures N.1-N.7 (sub-figure j) present the phase-angle of the unsteady shock motion for the same shock configuration and several perturbation frequencies. As the perturbation frequency increases, the phase-lag between the foot and top of the shock also increases which corresponds to some sort of wavy motion, the foot of motion being in advance compared to the top of the shock. This result can be explained by the fact that pressure perturbations propagate faster within the BL and reach the shock with a slight advance at the foot of the shock. It is important to mention that this phase-lag could be sensitive regarding the resulting moment coefficient on airfoils if a similar flow configuration would occur in an inter-blade passage and if the shock motion would oscillate with a 180° phase-angle between the suction side and the pressure side. Furthermore, Figure 7.1 shows that the shock motion phase-angle varies linearly with the perturbation frequency. This result can be explained by the fact that the wavelength of the upstream propagating perturbations is directly proportional to the perturbation frequency (since the propagation velocity is constant). The perturbations therefore reach the shock location with phase-angle value which is proportional to the ratio between the wavelength and the distance between the outlet plane and the shock location.

**7.2.1.4.3 Influence of shock location**

**Evolution of shock motion amplification**

Figures N.9-N.15 (sub-figure i) present, for the weak shock configuration, the amplitude of the unsteady shock motion at several perturbation frequencies. At this OP the shock wave is located close to the throat location, and only extends through one third of the channel height. At low frequency the shock motion amplitude of the fundamental varies from \(\pm 2.2\text{mm}\) above the bump (\(z=12\text{mm}\)) to \(\pm 3.8\text{mm}\) higher up at \(z=25\text{mm}\), and suddenly increases up to \(\pm 11.6\text{mm}\) at \(z=35\text{mm}\). These large amplitude oscillations at the top of the shock correspond due to the very low mean flow gradients. Furthermore, the non-linear contribution from the first harmonic represents around 21% of the fundamental
all along the shock length.

Although results at low frequency predict a smaller amplitude of motion for the weak shock configuration, it is interesting to compare the shock motion amplitude at higher frequencies as well. Figure 7.2(left) presents the shock motion amplitude as a function of the perturbation frequencies at two different vertical locations, for the weak shock configuration. It is striking that, for the location just above the bump surface, the amplitude of motion actually increases with the frequency until a maximum located at 250Hz and decreases beyond that frequency. On the other hand, at the top of the shock, the amplitude of motion presents a completely different behaviour and decreases with the perturbation frequency. It is noteworthy that the shock motion amplitude at 250Hz (Figure N.13i) is constant over the entire shock length whereas it is increasing, from the foot to the top, for all other perturbation frequencies. Although the above observations are not yet clearly understood, they illustrate the importance of the shock location (and SBLI intensity) regarding the shock motion amplitude.

![Shock Motion Amplitude](image1)

**Figure 7.2: Experimental shock motion for strong shock configuration and high perturbation amplitude ($A_p = \pm 2.12kPa$)**

**Evolution of shock motion phase-angle**

Figures N.9-N.15 (sub-figure j) present the unsteady shock motion phase-angle for the weak shock configuration and several perturbation frequencies. For the lowest frequency, namely 50Hz, the shock motion phase-angle is nearly constant throughout the channel height (slightly decreases close to the top of the shock). This observation is similar to the result obtained with the strong shock configuration and typical of a quasi-steady behaviour. The shock is oscillating as a rigid body without phase-lag between the foot and the top of the shock. It is interesting to note that the first harmonic phase-angle is also quite constant through the channel height except, again, close to the top of the shock. Beside the fundamental and first harmonic are nearly in opposite phase which means that the respective motions partly counter act each others.

Figure 7.2(right) presents the shock motion amplitude as a function of the perturbation frequencies at two different vertical locations, for the weak shock configuration. As observed, the phase-angle of the fundamental is the same for both vertical locations, meaning a rigid body shock motion even a higher perturbation frequencies. This observation is in contradiction with results obtained in the strong shock configuration where the phase-lag between the foot and the top of the shock was increasing with the perturbation frequency. A possible explanation involves the shape of the shock wave. Indeed,
for the weak shock configuration, the shock stands straight in the nozzle whereas it is slightly bent in the strong shock configuration. Besides, the shock motion phase-angle value decreases with the perturbation frequency similarly to the case with strong shock configuration. It is noteworthy that the phase-angle distribution, immediately above the bump (at z=12mm), is identical for both shock configuration (the small difference for increasing frequency is believed to originate from the 10mm difference in the mean shock location).

7.2.1.4.4 Influence of perturbation amplitude

Evolution of shock motion amplification

Figures N.17-N.23 (sub-figure i) present, for different perturbation frequencies, the unsteady shock motion amplitude for a strong shock configuration with a low perturbation amplitude (±1.14kPa instead of ±2.12kPa previously). Globally the trends are similar to the ones observed previously for a strong shock configuration with large perturbation amplitude. The shock motion amplitude increases with the vertical position in the channel. The only difference simply concerns the level of amplitude of the shock motion. A lower amplitude level of the outlet perturbations corresponds naturally to a lower shock motion amplitude. This result is in agreement with the unsteady pressure analysis which also showed a linear influence of the perturbation amplitude on the pressure amplification distribution.

Figure 7.3(left) presents the evolution of the shock motion amplitude for two locations in the channel height (z=12mm and z=50mm) as the perturbation frequency increases. The general trends are absolutely similar to the case with large perturbation amplitude. The shock motion amplitude decreases with the frequency and also exhibits a slight increase at 250Hz.

![Figure 7.3: Experimental shock motion for strong shock configuration and low perturbation amplitude (A_p = ±1.14kPa)](image)

Evolution of shock motion phase-angle

Figures N.17-N.23 (sub-figure j) present the shock motion phase-angle for different perturbation frequencies. The results are similar to the phase-angle behaviour obtained with a larger perturbation amplitude. The phase-angle decreases linearly with the perturbation frequency and the phase-lag between the foot and top of the shock linearly
increases with the frequency. Figure 7.3(right) presents the shock motion phase-angle evolution as a function of the perturbation frequency. The phase-angle distribution does not seem to be affected by the perturbation amplitude.

7.2.1.4.5 Concluding remarks

This sub-section was dedicated to the description of the unsteady shock motion throughout the channel height. Both amplitude and phase-angle distributions were described and a parametric study was conducted to establish the respective influence of the mean shock location, the perturbation amplitude and frequency.

The perturbation frequency has an influence both on the amplitude and phase-angle distributions of the shock motion. Indeed, the shock motion amplitude decreases, for any location in the channel height, as the perturbation frequency increases. The phase-angle also decreases with the frequency but, more interesting, the phase-lag between the foot and the top of the shock increases with the perturbation frequency. For low frequency, the shock oscillates in a rigid body motion whereas the foot of the shock oscillates in advance compared to the top of the shock at high perturbation frequency (in the case of a strong shock configuration).

The shock location was found to have a quite important influence both on the amplitude and phase-angle of the shock motion. As a matter of fact, the influence of the perturbation frequency is directly dependent on the mean shock location. For a weak shock configuration, the amplitude increases with the perturbation frequency until a certain frequency value and thereafter decreases. It was, however, not clear why the behaviour was only observed for the weak shock configuration and why the frequency value of 250Hz corresponded to a maximum amplitude of the shock motion. Besides, the influence of the shock location on the phase-angle distribution concerned the shock motion phase-lag between the foot and the top of the shock, which was found constant and independent of the perturbation frequency for the weak shock configuration only.

Finally, the perturbation amplitude does not seem to have any influence except on the level of pressure amplification, which seems to be directly proportional to the amplitude of the outlet pressure perturbations. No other influence of the perturbation amplitude was however observed.

7.2.1.5 Analysis of shock motion and unsteady pressure perturbation correlation

Figure 7.4 presents the phase-lag between the unsteady shock motion above the bump surface (z=12mm) and the unsteady pressure phase-angle (fundamental contribution) downstream of the shock as a function of the perturbation frequency and for the different OPs. The value of 180° was taken out of the difference of the respective phase-angle values in order to account for the quasi-steady phase-lag between the shock motion and the pressure perturbations. Indeed, when the downstream pressure of the shock increases, the shock moves upstream thus with a 180° phase-lag with the incoming pressure per-
turbations. As a result, a negative phase-lag value corresponds to a delay in the shock response to the incoming pressure perturbations.

![Phase-Lag Between Shock Motion and Unsteady Pressure Perturbations](image)

Figure 7.4: Experimental phase-lag between unsteady pressure perturbation downstream of shock and shock motion for different operating conditions

At low frequency the phase-lag increases almost linearly with the perturbation frequency. The time delay for the shock wave to "react" to the incoming pressure perturbations slightly increases with the frequency. Although the different operating conditions show different evolution of the phase-lag with the frequency, it is difficult to make any correlation between the observed phase-lag distribution and the operating conditions. There is for instance no direct explanation to justify the 30° phase difference observed at 100Hz for the strong shock configuration between the low and high perturbation amplitudes. Indeed, the previous analysis on the influence of the perturbation amplitude, performed for both the unsteady shock motion and pressure distribution. On the other hand, it should be noted that the presented data is issued from visual reading of the pressure and shock motion phase-angle and are entangled of a certain (quite large) error especially related to the upstream location of the shock. The trends however should be correct.

At higher frequency, the phase-lag between the shock motion and the unsteady pressure perturbations downstream of the shock decreases and even becomes positive for the weak shock configuration. Such positive phase-lag corresponds to a shock motion in advance of phase compared to the upstream propagating pressure disturbances, which is not really logical. However, as mentioned above, the current results are subject to a certain error and this low positive value might just correspond to a zero phase-lag behaviour between the incoming pressure perturbations and the resulting shock motion. However, even in such a case the decrease of the phase-lag at the perturbation frequency of 250Hz is in contradiction with previous trends at low frequency. It is unfortunately unclear why this behaviour happens and especially why at this particular perturbation frequency. It is although interesting to note that in the case of the weak shock configuration, a maximum of amplitude of the shock motion was also found at this particular frequency (250Hz). For even higher frequency (at 500Hz), the phase-lag increases anew, following the low frequency tendency, except for the OP with the strong shock configuration and low amplitude. Again, no obvious reason was determined which could explain such a behaviour.
7.2.1.6 Analysis of unsteady pressure phase shift underneath shock

As mentioned during the description of the phase-angle distributions for the different
OPs, a phase shift could be observed on the unsteady pressure phase-angle distribution
on the bump, underneath the shock location. It should be noted that this phase shift
is extremely important regarding aeroelastic stability prediction. Indeed, in the case of
an oscillating airfoil, this phase shift would directly contribute to the modification of the
resulting moment coefficient and possibly affect the stability of the airfoil. The phase shift
was therefore investigated for different operating conditions and results are presented in
Figure 7.5 below.

![Figure 7.5: Experimental phase shift underneath shock location for different operating
conditions](image)

Influence of perturbation frequency

Clearly, for all OPs, the phase shift underneath the shock increases (in absolute
value) linearly with the perturbation frequency. The fact that the actual phase shift val-
ues are negative is due to the fact that the pressure phase-angle is higher downstream
of the shock, which corresponds to a delay of the upstream propagating pressure waves
underneath the shock location. This result is quite surprising since the unsteady pressure
distribution results from a complex interaction between upstream propagating pressure waves
and the oscillating shock, and exhibited phase-angle jumps and “slope inversions”, especially
at high frequencies.

Influence of shock location

According to the results shown in Figure 7.5, the phase shift seems to increase faster
for strong shock configuration. The sensitivity (versus the perturbation frequency) seems
therefore directly correlated to the pressure jump conditions through the shock. The
higher the pressure jump through a shock discontinuity is, the more the phase shift in-
creases with the frequency.
Influence of perturbation amplitude

The perturbation amplitude, as well, seems to have an influence on the unsteady pressure phase shift underneath the shock location. For the same steady state shock configuration, i.e. the same shock strength, and a given perturbation frequency, the lower perturbation amplitude produces a lower unsteady pressure phase shift on the surface underneath the shock.

The above observations about the parametric influence on the phase shift value are quite interesting to predict which perturbation frequency, for a given shock location and perturbation amplitude, would produce a $180^\circ$ phase shift underneath the shock and banefully contribute to the imaginary part of the resultant aerodynamical force acting on the surface. As result, for the different operating conditions presented above, the perturbation frequencies of 300Hz, 350Hz, and 650Hz would be considered as problematic regarding the overall moment coefficient in case of an oscillating airfoil.

7.2.1.7 Analysis of time-averaged versus steady state pressure distribution

Figures 7.1-7.3 (sub-figures b) present the unsteady time-averaged versus the steady state pressure distributions in the middle of the channel ($y=50\text{mm}$) for each OP. The general tendency seems to place the shock from the unsteady time-averaged data slightly upstream of the steady state pressure distribution. This can be explained by considering that the unsteady oscillations of the shock wave create higher losses than in the steady state case. This consideration seems actually quite logical since the shock periodically moves downstream of its steady state location and generates higher losses. Since the loss generation is not a linear phenomenon with the shock location and therefore nonlinearly increases as the shock moves downstream and gets stronger, the resulting time averaged losses over one unsteady cycle is higher than the steady state value during the same time. As a result the pressure gradient between the shock location and the outlet plane is higher, and since the back pressure is fixed the pressure immediately downstream of the shock is increased and the time-averaged shock position is slightly moved upstream.

It is however difficult to establish a clear parametric influence due to a "too coarse" spacing of the measurement points underneath the shock location.

7.2.2 Numerical results and comparison with experiments

7.2.2.1 Introduction

From the 2D steady state RANS calculations presented in chapter 6, four flow configurations with different mean shock locations were selected for unsteady computations. Table 7.1 presents the unsteady operating conditions for all numerical simulations of the 2D nozzle. First, a subsonic OP was chosen to determine the influence of the presence of a shock on the pressure amplification phase-angle distribution. Then three other configurations with different shock structures, from weak to a large lambda shock system with choked nozzle flow, were also simulated. Unsteady computation were performed, for each of the
steady state OPs, at three different perturbation frequencies but only one perturbation amplitude. As a result the two intermediate steady state flow configurations can directly be compared with experimental results for two perturbation frequencies, namely at 100Hz and 500Hz.

<table>
<thead>
<tr>
<th>Steady state OP</th>
<th>Unsteady Back pressure perturbations</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_{inlet}$ mPa</td>
<td>$T_{inlet}$ K</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
</tr>
<tr>
<td>160</td>
<td>303</td>
</tr>
</tbody>
</table>

Table 7.1: Unsteady operating conditions for 2D RANS calculations in 2D nozzle

The following sections therefore first present the unsteady numerical results with the emphasis on the parametric influence of the perturbation frequency and mean shock location, and thereafter focus on the comparison with the experimental results. More precisely, the analysis concerns the amplitude and phase-angle distribution of the unsteady pressure both on the 2D bump surface and within the channel, the unsteady shock motion, the separated region motion and acoustic flow composition.

### 7.2.2.2 Analysis of unsteady pressure distribution on bump

#### 7.2.2.2.1 Influence of shock location

Evolution of unsteady pressure amplification

Figures N.25, N.28, N.31, and N.34 (sub-figure g) present the pressure amplification distribution obtained from unsteady calculations performed at a perturbation frequency of 100Hz for the different steady state OPs listed in Table 7.1. The Time-Averaged pressure distribution is plotted together with the fundamental and first harmonic contributions in order to correlate the pressure amplification to the shock location and strength.

The observed peak underneath the shock location basically reflects how the pressure discontinuity through the shock is diffused and damped through the BL. The strength of the shock (consequently its location) as well as the BL thickness and velocity distribution (though the shape factor parameter) are therefore directly responsible for the amplification level of this peak. For the subsonic unsteady flow calculation ($P_{out}=120kPa$ in Figure N.25(g)) the amplification is about 3.1 and is actually due to the fact that a weak shock appears during a part of the unsteady cycle. For further downstream steady state shock positions, the pressure amplification peak level increases due to a higher pressure discontinuity through the shock and its presence during the entire unsteady cycle. It is interesting to note however that, for the strongest shock configuration (with $P_{out}=102kPa$ in Figure N.34(g)), the peak level is actually lower due to the large and thick lambda shock system which features a smoother repartition of the pressure discontinuity and a
larger BL thickness.

Furthermore, it is interesting to note that, for any unsteady calculation, the pressure amplification outside the shock motion region is about twice the amplitude of the outlet disturbances downstream of the shock location, but outside the shock motion region. This amplification of the downstream perturbations is in agreement with the acoustic blockage theory (Atassi et al., 1994a), which corresponds to the stagnation and amplification of upstream propagating acoustic waves in near sonic flow regions. It is noteworthy that the pressure amplification distribution for the strong shock but not choked configuration (with $P_{\text{out}}^s = 110\text{kPa}$ in Figure N.31(g)) is both longer (in the streamwise direction) and higher than for any other OPs. It should be noted that a 30mm long shock induced separation has appeared at this OP and is synonym of a large BL thickening. Although it is tempting to relate the shape and level of the pressure amplification distribution to this flow characteristic, it is not yet clear whether it is the origin of the observed unsteady pressure distribution.

It should finally be mentioned that, for the subsonic and weak shock configurations (with $P_{\text{out}}^s = 120\text{kPa}$ and $116\text{kPa}$ in Figures N.25(g) and N.28(g) respectively), the travelling pressure perturbations nearly recover their downstream amplitude levels upstream of the nozzle contraction. This is naturally due to the fact that the shock does not spread over the entire channel height and therefore does not block the pressure perturbations from propagating upstream. Indeed for the two strong shock configurations (with $P_{\text{out}}^s = 110\text{kPa}$ and $102\text{kPa}$ in Figures N.31(g) and N.34(g) respectively) the unsteady pressure amplitude level is zero upstream of the shock location.

**Evolution of unsteady pressure phase-angle**

Figures N.25, N.28, N.31, and N.34 (sub-figure h) present the phase-angle distribution computed at a perturbation frequency of 100Hz for the different steady state OPs listed in Table 7.1. Similarly to the experiments, the phase reference was chosen at $x=290\text{mm}$, slightly upstream of the numerical outlet plane. The value of the phase-angle is therefore zero at this location and decreases towards the shock location as a single plane pressure wave (due to a uniform fluctuating back pressure at the outlet) is travelling upstream.

For the subsonic OP (with $P_{\text{out}}^s = 120\text{kPa}$ in Figure N.25(h)) the phase-angle is linearly decreasing while propagating upstream. This is a typical phase-angle behaviour for a single plane wave travelling at a constant relative velocity of $c-U$. It should be noted that, at this frequency and velocity, the wavelength is so long that there can only be plane waves travelling within the channel. In such case the phase-angle distribution is directly proportional the the travelling speed which, itself, depends on the main flow velocity. It is therefore interesting to note the fairly regular phase-angle distribution even though the flow accelerates and then decelerates in the contraction and diffusor part of the nozzle respectively as seen in Figure N.25(a). As a matter of fact, the phase-angle slightly varies as observed by the deformation of the wave front in the channel over the bump in Figure N.25(d), but the wavelength is so long that the phase-angle variations are barely noticeable. Furthermore, a small phase shift is observed at the throat location and is believed to originate from the appearance of a weak shock during part of the unsteady cycle. It is also
interesting to note the phase-angle behaviour of the first harmonic, especially the strong increase between $x=35\text{mm}$ and $x=50\text{mm}$, and the sudden drop at $x=50\text{mm}$. Although no clear understanding of this behaviour was reached, it is believed to originate, as well, from the intermittent appearance of the shock wave at the throat location.

For the weak shock configuration OP (with $P_{out}^{\text{op}}=116\text{kPa}$ in Figure N.28(h)) the phase-angle distribution is very similar to the previous description. A few differences can however be observed. The phase-angle distribution stops decreases linearly around $x=100\text{mm}$ and even slightly increases downstream of the shock. Furthermore, the phase shift underneath the shock location has increased compared to the subsonic OP previously described. It is believed that the strength of the shock and its presence during the entire unsteady cycle are responsible for the observed phase jump. The slight phase-angle distribution change is more problematic to understand since it basically corresponds, within the assumption of single plane travelling waves, to a downstream propagating wave.

For the strong shock configuration OP (with $P_{out}^{\text{op}}=110\text{kPa}$ in Figure N.31(h)) the few particularities noticed during the description of the previous OP are now clearly marked. The phase-angle distribution downstream of the shock first changes towards a more rapid decrease (at $x=160\text{mm}$), meaning a shorter wavelength and thus a lower wave propagation velocity, and thereafter increases (around $x=110\text{mm}$) synonym, at this frequency, of a downstream travelling wave. Furthermore the phase shift underneath the shock location has also increased, therefore changing the phase-angle distribution over the first half of the channel (compared to previous distributions with weaker shock configurations). As mentioned above, in the description of the experimental unsteady pressure distribution, this could give a significant contribution to the overall moment coefficient and consequently to the stability prediction if the results would be transposed, for instance, to the case of an oscillating airfoil. Although there is no clear understanding of the phase-angle distribution behaviour downstream of the shock, the observed tendency seems related to the shock size (increasing height while moving downstream), the shock strength (i.e. the intensity of the pressure jump through the shock), the SBLI or the shock induced separation and therefore large BL thickening.

For the strong shock configuration with choked nozzle flow (with $P_{out}^{\text{op}}=102\text{kPa}$ in Figure N.34(h)) the phase-angle distribution also shows a slight inclination of the slope (towards a lower wavelength thus lower propagation velocity) but no increase as in the previous case. It should be noted that, at this OP, the shock has evolved into a large lambda shock system which impacts on the concave part of the diffusor. The pressure discontinuity is thus much more diffused due to the set of oblique compression waves and also damped through the BL. This probably has an important effect on the dissipation, diffusion and reflection of the incident pressure waves. It is, by the way, interesting to note that the phase-angle distribution decreases until the location of the first oblique shock at $x=100\text{mm}$. The upstream propagating waves cannot progress beyond that location. The amplitude level of the unsteady pressure distribution is therefore zero and the phase-angle distribution does not correspond to anything physical and simply show random values.
7.2.2.2 Influence of perturbation frequency

Evolution of unsteady pressure amplification

Figures N.25-N.27 (sub-figure g) present, for the subsonic OP (with $P_{out}^{s} = 120\text{kPa}$), the evolution of the pressure amplification distribution for different perturbation frequencies. As mentioned in the previous paragraph, the pressure amplification peak observed at 100Hz around the throat location actually corresponds to the emergence of a weak shock during part of the unsteady cycle. At 500Hz the pressure amplification distribution for the fundamental is very similar to the previous case at 100Hz, with a level above 1 outside the shock motion region both upstream and downstream of the throat location. The main difference at 500Hz concerns the pressure amplification peak of the first harmonic, exhibiting a level of 1.8 at the throat location, which is certainly due to the intermittency of the shock appearance. It is noteworthy that the amplification level of the first harmonic also increases well beyond the extent of the shock motion, reaching for example 80% of the outlet perturbation amplitude and 40% of the linear (Fundamental) contribution at $x=55\text{mm}$. Finally at 1000Hz, the pressure amplification peak levels of both the fundamental and first harmonic have again increased due to the appearance of the shock over a longer part of the unsteady cycle as the perturbation frequency increases. An attenuation of the unsteady pressure distribution can be observed at $x=100\text{mm}$ but no direct explanation could be reached.

For the weak shock configuration (with $P_{out}^{s} = 116\text{kPa}$ in Figures N.28-N.30, sub-figures g) the influence of the perturbation frequency on the pressure amplification is characterized, on one hand, by a decrease of the peak level and secondly by the extension of the region of high level pressure amplification downstream of the shock. At high frequency (1000Hz) however, a pressure attenuation can be observed similarly to the previous case with subsonic steady state OP.

For the strong shock configuration (with $P_{out}^{s} = 110\text{kPa}$ in Figures N.31-N.33, sub-figures g) the evolution of the amplification peak level decreases with the perturbation frequency, similarly to the case described just above. A quite different behaviour of the pressure amplification distribution however occurs downstream of the shock location. Indeed, whereas the computation at 100Hz exhibited a long and high pressure amplification downstream of the shock, the distribution at 500Hz shows an attenuation similar to the ones observed at 1000Hz for weaker shock configuration. It is believed that such an attenuation is the result of a complex interaction between upstream and downstream propagating waves with varying amplitude and phase distributions. The pressure amplification distribution is indeed very similar to sinusoidal signals with opposite phase-angle and partly cancelling each others out. The decomposition and analysis of the acoustic field is presented further below in this subsection.

Finally, for the strong shock configuration and choked nozzle flow (with $P_{out}^{s} = 110\text{kPa}$ in Figures N.34-N.36, sub-figures g) the evolution of the pressure amplification peak level follows the general trend and decreases with the perturbation frequency. The pressure amplification distribution, as well, seems to decrease downstream of the shock as the frequency increases. This probably results from the damping of the pressure fluctuations through the BL.
7.2. TWO-DIMENSIONAL NOZZLE

Evolution of unsteady pressure phase-angle

Figures N.25-N.27 (sub-figure h) present, for the subsonic OP (with $P_{out}^{s}=120\text{kPa}$), the evolution of the phase-angle distribution for different perturbation frequencies. Globally, the perturbation frequency has an influence on the phase shift underneath the shock location, which looks more like an increase of the phase-angle for this OP.

For the weak shock configuration (with $P_{out}^{s}=116\text{kPa}$ in Figures N.28-N.30, sub-figures h) the influence of the perturbation frequency on the phase-lag distribution is similar than for the previous subsonic OP. The phase shift underneath the shock increases with the frequency. No other influence could be observed for this OP.

For the strong shock configuration (with $P_{out}^{s}=110\text{kPa}$ in Figures N.31-N.33, sub-figures h) the influence of the perturbation frequency on the phase-lag distribution is much more complex than in the precedent cases with weaker shock configurations. Indeed, at 100Hz already the phase-angle distribution features a change of slope and a phase shift respectively downstream and underneath the shock location. As the perturbation frequency increases, at 500Hz, another phase shift has appeared outside the shock motion extent. Furthermore at 1000Hz, the phase-angle distribution has again changed but still feature the phase shift underneath the shock. It is believed that, at this particular OP, the separated region motion, the upstream propagating perturbations and the shock oscillations interact together and generate the observed amplitude and phase-angle distributions.

For the strong shock with choked nozzle configuration (with $P_{out}^{s}=102\text{kPa}$ in Figures N.34-N.36, sub-figures h) the influence of the perturbation frequency on the phase-lag distribution is surprisingly almost non existent. Indeed the only observable differences concern the slope of the phase-angle distribution which logically changes with the perturbation frequency.

Finally, Figure 7.6 presents the phase shift observed underneath the shock as a function of the perturbation frequency for the different OPs. Similarly to the experiments, the increase of the phase shift is fairly linear with the frequency, which, as discussed previously, can give a significant contribution to the resultant aerodynamic force acting on the surface.

7.2.2.2.3 Comparison with experiments

Evolution of unsteady pressure amplification

Figure 7.7 presents the unsteady pressure amplification distribution obtained on the weak shock configuration for the perturbation frequencies of 100Hz and 500Hz. The agreement between experimental and numerical results is fairly good for both perturbation frequencies. The 2D Euler calculation seems to predict a slightly larger shock motion extent and a higher amplification peak level, especially at high frequency. On the other hand, the 2D RANS calculation and experimental results are very similar and collapse almost perfectly. The main result from this comparison is that the pressure amplification
which occur at 500Hz downstream of the location originates from a non-viscous phenomenon, most probably the acoustic blockage as describe by Ferrand et al. (1995) as an inviscid interaction.

It is noteworthy that although the comparison on the pressure amplification between 2D RANS calculations and experiments showed a fairly good agreement the amplitude of perturbation was different. Indeed the same elliptical rod was found to create a pressure perturbation amplitude varying with the operating conditions whereas the amplitude of perturbation was systematically set to 2% of the outlet static pressure for the numerical calculations. As a result, the amplitude of perturbation between simulations and experiments are similar for the strong shock configuration but slightly different for the weak shock configuration. This result seems to indicate a low influence of the perturbation amplitude on the pressure amplification distribution. However, anticipating forthcoming observations, this difference in the perturbation amplitude between numerical simulations and experiments for the weak shock configuration has a clear influence on the amplitude of the shock motion.

Similarly, Figure 7.8 presents the unsteady pressure amplification distribution for the strong shock configuration. It is interesting to note that, for a perturbation frequency of 100Hz, both the 2D RANS calculation and the experiments predict the pressure amplification downstream of the shock. Although the curves do not collapse perfectly (especially,
the slight attenuation immediately downstream of the shock observed on experimental results at \(x=72\) mm is not predicted by the 2D RANS calculation) the main tendencies and amplitude level are however correctly predicted. On the contrary, the 2D Euler simulation does not "capture" the pressure amplification at all, which therefore must originate from a viscous or turbulent phenomenon or interaction. Interestingly, the OP features a shock induced separation both experimentally visualized and numerically predicted. It is therefore quite possible that the separation might play an important role in the SBLI which presumably results in the observed pressure amplification distribution. Furthermore, at 500Hz, the very same observations and conclusions can be drawn although the amplification distribution is completely different and actually features a pressure attenuation. An attempt to further understand the basic mechanisms of the interaction by decomposing the unsteady pressure distribution into upstream and downstream propagating waves at specific respective velocities is presented further below.

![Graph](image_url)

**Figure 7.8:** Experimental-Numerical comparison of unsteady pressure amplification for the strong shock configuration at 100Hz and 500Hz

**Evolution of unsteady pressure phase-angle**

Figure 7.9 presents the unsteady pressure phase-angle distribution obtained on the weak shock configuration for the perturbation frequencies of 100Hz and 500Hz.

At 100Hz, the agreement between numerical simulation and experiments concerning the phase-angle distribution is also fairly good. Although the curves collapse almost perfectly, it should be mentioned that there should actually be a small offset between numerical and experimental results. Indeed, as mentioned during the description of experimental results, the experimental reference pressure was located on the side wall whereas the unsteady pressure measurements were performed on the lower wall. Due to the local curvatures, the outflow features a non-uniformity between the lower and upper walls. As the pressure perturbations generated downstream of the test section by the rotating elliptical rod propagate upstream at the relative velocity \(c-U\), there logically exists a continuous phase-lag between the lower and upper walls. Consequently the outlet pressure measurement does not exactly correspond to the reference for the unsteady pressure measurements performed on the bumps. However, for each OP, this phase-angle "offset" is directly proportional to the perturbation frequency and results can still be compared relatively to each others. Nevertheless the phase-lag between the lower and upper walls should definitely be checked in future measurements in order to confirm the assumption above.
The comparison between 2D RANS calculation and experiments is fairly good. Especially, the phase-angle distribution features the same trend concerning the slight increase observed downstream of the shock. On the other hand the 2D Euler calculation obviously overestimate the "increase" of the phase-angle, which leads to the conclusion that this effect results essentially from an inviscid phenomenon and is somehow damped by viscous or turbulent effects. Nonetheless, all results predict the small phase shift underneath the shock location, which therefore seem to originate from the shock motion.

At 500Hz, the offset mentioned above between experimental and numerical results is clearly seen and approximately equals to 90°. It is noteworthy that the 2D RANS calculation and experiments feature the same phase-angle distribution, shifted because of the offset mentioned above. It is also interesting to note that 2D Euler and 2D RANS calculations have a slightly different slope of the phase-angle distribution curve. The is a logical consequence of BL thickening effect. For viscous calculations, the slight acceleration provoked by the reduction of the effective section area results in a lower propagation velocity for the pressure perturbations and therefore a shorter wavelength at a similar frequency, thus a more "inclined" phase-angle distribution curve.

Figure 7.9: Experimental-Numerical comparison of unsteady pressure phase-angle for the weak shock configuration at 100Hz and 500Hz

Figure 7.10 presents the unsteady pressure phase-angle distribution obtained on the weak shock configuration for the perturbation frequencies of 100Hz and 500Hz. At 100Hz, apart from the phase-angle offset already discussed in the previous paragraphs, the 2D RANS and experimental results show fairly similar trends concerning the phase-angle "increase" downstream of the shock location. Similarly to the comparison on pressure amplification at the same OP, the 2D Euler calculation clearly underestimates this effect, which therefore ought to originate from a viscous interaction.

At 500Hz, similar observations can be made. Apart from the difference due to the phase-angle offset, the 2D RANS and experimental results present the same trends and in particular the same phase shift at x=95mm, outside the shock motion extent. On the other hand, the 2D Euler calculation does not predict this phase shift nor the "increasing" phase-angle distribution immediately downstream of the shock.

Evolution of unsteady pressure phase shift underneath shock

Figure 7.11 presents, for both shock configurations, the unsteady pressure phase shift...
7.2. TWO-DIMENSIONAL NOZZLE

Figure 7.10: Experimental-Numerical comparison of unsteady pressure phase-angle for the strong shock configuration at 100Hz and 500Hz

underneath the shock location obtained from 2D RANS calculations and experiments for different perturbation frequencies. For both shock configurations the agreement between calculations and experiments is fairly good at all perturbation frequencies. As observed on experimental results and confirmed by numerical calculations, this phase shift is dependent on the shock location and both the perturbation frequency and amplitude.

Figure 7.11: Experimental-Numerical comparison of unsteady pressure phase-shift underneath shock location

7.2.2.2.4 Concluding remarks

Although the amplitude of the unsteady pressure distribution showed both amplifications and attenuations for different perturbation frequencies and different steady state shock locations, it was not possible to clearly formulate the relative influence of each parameter. On the other hand, the phase-angle distribution showed a more coherent behaviour related to the perturbation frequency and shock location. Among others, the phase shift underneath the shock location appeared to change depending on the shock location (i.e. the shock strength) and to linearly vary with the perturbation frequency.

Interestingly, it was observed, for a strong shock configuration, a rather different behaviour of both the pressure amplification and phase-angle distribution downstream of the shock. The comparison between experiments, inviscid and RANS calculations clearly showed that the observed behaviour resulted from a viscous interaction. It is furthermore noteworthy that for this particular OP, and this OP only, the flow field features a shock induced separation and therefore a quite important BL thickening. It is thus very
probable that the observed pressure amplification and phase-angle behaviour are directly related to this flow characteristic.

From a general perspective, the comparison between experimental and numerical (both Euler and RANS) results was fairly good. In most cases the pressure amplification (or attenuation) and phase-angle distributions were captured by the simulations. In particular, the phase shift underneath the shock was fairly well predicted.

### 7.2.2.3 Analysis of unsteady pressure distribution in channel

#### 7.2.2.3.1 Influence of shock location

**Evolution of unsteady pressure amplification**

Figures N.25(c), N.28(c), N.31(c) and N.34(c) present the fundamental contribution of the pressure amplification distribution at the lowest perturbation frequency (100Hz) for the all different OPs.

For the subsonic shock configuration (with $P_{out_s}=120kPa$), a general pressure amplification, i.e. with an amplitude of fluctuation above the outlet reference value, can be observed above the bump contraction (from $x=20mm$ to $x=100mm$) throughout the entire channel height. This is a direct effect of the acoustic blockage theory. As the flow velocity is higher in the contraction part of the nozzle, upstream propagating pressure perturbations "slow" down, which leads to an accumulation of acoustic energy in near sonic flow regions and produces the observed high amplitude pressure fluctuations. Interestingly for this case, the pressure amplification is not high at all around the throat location although a weak shock does appear during part of the unsteady cycle. The reason is probably because the shock is so weak that the extent of the sonic pocket is very small. Consequently, downstream pressure perturbations propagate upstream of the bump but with a slightly lower amplification factor value. Two reasons can explain this observation. First, the flow velocity is slightly lower at the inlet than at the outlet due to the BL thickening all along both upper and lower walls. Secondly, due to the higher flow velocity in the contraction part of the nozzle, the wavelength of the pressure perturbations considerably decreases which leads to numerical dissipation since the mesh cannot correctly describe the more or less sinusoidal pressure variation.

For the weak shock configuration (with $P_{out_s}=116kPa$), the pressure amplification in the contraction part of the nozzle is even higher than in the previous case since a sonic pocket has appeared over a part of the channel height and the flow velocity above it is therefore very close to sonic conditions. The pressure discontinuity through the shock is no longer small and leads to a strong pressure amplification over the shock motion area.

For the strong shock configuration (with $P_{out_s}=110Pa$), the shock reaches the upper wall (although the steady state shock does not spread over the entire channel height) and therefore choke the nozzle during a part of the unsteady cycle. As a result the back pressure perturbations do not propagate further upstream than the shock location. A pressure amplification can however be observed close to the upper wall, around the shock...
location but outside its motion area, and basically corresponds to the acoustic blockage effect while the nozzle is not choked by the shock. More interesting is the pressure amplification above the bump, immediately downstream of the shock location. It should be reminded that at this OP a shock induced separation leads to a 30mm long separated flow in the diffusor part of the nozzle. It is believed that the coupling between the sudden BL thickening, leading to the so called post shock expansion (see figure 2.12 in chapter 2), the shock oscillations (inducing the separation motion) and the acoustic blockage effect is responsible for the observation pressure amplification. Indeed no strong pressure gradient or discontinuity is present in this region of the flow field, but the shock oscillations naturally induce a fluctuation of the BL thickening (with the strongest flow deviation as the shock is in its most downstream position), which leads to periodical variations of the post shock expansion, both in size and velocity level. As a result upstream propagating pressure perturbations are amplified in this region which features a high flow velocity during part of the unsteady cycle.

Similarly, for the strongest shock configuration (with $P_{out}^{s}=102\text{Pa}$), the pressure amplification is the highest within the shock motion extent. Two others regions with significant pressure amplification can be seen on both the upper and lower walls, downstream of the shock. On the lower wall the pressure amplification is basically due to the large lambda shock system, which features a weak rear shock with a low amplitude pressure jump, therefore resulting in a high flow velocity region. Furthermore, as the shock impacts on a concave surface, the flow velocity starts decreasing already upstream of the shock, which reduces the influence of the APG and avoid a separation of the flow. The BL thickening is therefore not as strong as in the previous case. On the other hand, on the upper wall, the interaction between the strong normal shock and the BL does induce a separation. Besides, the similarity with the pressure amplification distribution in the previous case (with $P_{out}^{s}=110\text{Pa}$) is striking and confirm that the pressure amplification mechanism in this region is directly related to the presence of the separation.

Finally, it should be mentioned that the analysis of the pressure amplification distribution of the first harmonic contribution, shown in Figures N.25(e), N.28(e), N.31(e) and N.34(e), did not provide any new information about the mechanism of pressure amplification in the nozzle.

**Evolution of unsteady pressure phase-angle**

Figures N.25(d), N.28(d), N.31(d) and N.34(d) present the fundamental contribution of the phase-angle distribution at the lowest perturbation frequency (100Hz) for the all different OPs.

For the subsonic shock configuration (with $P_{out}^{s}=120\text{Pa}$), the phase-angle distribution throughout the entire channel corresponds to the single travelling plane wave theory. The phase-angle indeed decreases as the downstream pressure perturbations propagate upstream at the relative velocity $c-U$. A slight modification of the phase-angle distribution can be observed close to the lower wall, in the converging and diverging parts of the nozzle as the flow velocity (and therefore the wave propagation velocity) increases and decreases respectively. Besides, the presence of the shock during a part of the un-
steady cycle does not seem to influence the phase-angle distribution at the throat location.

For the weak shock configuration (with $P_{\text{out}}=116\text{kPa}$), the phase-angle distribution follows the trends described previously. The presence of the weak shock wave during the entire unsteady cycle has however a "retardation" effect on the phase-angle value. Indeed the phase-angle does not decrease any longer downstream of the shock location and seems to remain constant within the shock motion extent until it suddenly "drops" at the most extreme shock position. Although not well understood, this effect might simply come from the fact that acoustic waves are blocked by the shock discontinuity, reflected and bounced back or are absorbed by the shock.

For the strong shock configuration (with $P_{\text{out}}=110\text{kPa}$), the phase-angle distribution clearly revealed the pressure amplification region downstream of the shock described above. The similarity between the observed "modified" phase-angle zone and the representation of the BL thickening (and shear flow) downstream of the shock is truly striking. It seems clear that the pressure amplification and phase-angle behaviour (retardation) are closely related to the interaction between upstream propagation of back pressure perturbations and the strong BL thickening (and possibly the separation). Interestingly the phase-angle distribution within the observed "BL thickening" region slightly decreases, similarly to the previous OP with the weak shock configuration, but somehow at a higher magnitude. It should finally be mentioned that the phase-angle distribution within the inlet part of the nozzle, down to the shock location, has a limited physical meaning since the shock mostly blocks the perturbations from propagating upstream and the pressure amplification is therefore nearly zero as observed in Figure N.31(c).

For the strongest shock configuration (with $P_{\text{out}}=102\text{kPa}$), the similar phase-angle retardation can be observed downstream of the shock on both the upper and lower walls. The absence of curvatures on the upper wall minimizes the APG effect to the shock influence only and therefore reduces the intensity of the BL thickening. This seems to result in a smaller extent of the pressure amplification area compared to the situation encountered in the previous OP where both the curvatures and the shock were strongly contributing to the BL thickening. The situation on the lower wall is slightly different since the shock impacts on concave curvatures where the flow is decelerating and the BL already thickening which induces a set of upstream oblique shock resulting in the observed lambda shock system. As a result, although the curvature induce an APG, they also lead in the present configuration to a more diffused pressure gradient with less severe consequences on the BL. Indeed the flow remains attached downstream of the shock which is probably the reason of the smaller extent of the pressure amplification region compared to the previous OP.

7.2.2.3.2 Influence of perturbation frequency

Evolution of unsteady pressure amplification

Figures N.25-N.27(sub-figures c) present, for the subsonic OP (with $P_{\text{out}}=120\text{kPa}$), the evolution of the pressure amplification distribution of the fundamental as the perturbation frequency increases. As discussed above, the pressure amplification distribution at low frequency corresponds to the propagating theory of a single travelling plane wave.
This seems to be valid as well at 500Hz, for which the intermittent appearances of the shock still does not have a huge influence on the pressure amplification around the throat location. At 1000Hz, however, there is a quite important modification of the pressure amplification distribution pattern. First, a large amplification occurs at the shock location. This contrasts with forthcoming observations on other OPs where the pressure amplification level decreases with the perturbation frequency. Secondly, the amplified pressure zone at the throat location actually extends quite a lot around the shock, outside the shock motion extent. Thirdly, a large pressure attenuation zone is located in the diffusor (at x=100mm) just downstream of the previously mentioned amplification region. And most interestingly, another pressure amplification region can be observed also in the diffusor but close to the upper wall, from x=50mm to x=150mm. Although the interpretation of these observations is fairly fastidious, the pressure amplification pattern suggests an interference pattern between upstream propagating and reflected (both on the surface or on the shock) or diffracted (at the "top" of the shock) waves. Indeed, at such high frequency, the pressure perturbation wavelength is small enough so that transversal standing waves can exist within the channel. As a result, depending on the flow characteristics (flow velocity, shock location), the geometry (curvatures) and the perturbations (frequency), a spatially steady pattern of interference between the upstream propagating pressure waves and reflected or diffracted waves may be observed in cases where contributing sources are coherent or phase related.

Figures N.28-N.30 (sub-figures c) present, for the weak shock configuration (with $P_{out} = 116kPa$), the evolution of the pressure amplification distribution of the fundamental as the perturbation frequency increases. At 100Hz, the pressure amplification pattern corresponds to the acoustic blockage theory as the flow is near sonic condition at the throat of the nozzle. The strong amplification observed at the throat corresponds to the shock motion extent. As the perturbation frequency increases the pressure amplification pattern changes and exhibits an important value above and downstream of the shock. Curiously the pressure amplification, at the shock location, fades down towards the upper wall, which is in agreement with the acoustic blockage theory. Besides, the amplitude level of the pressure perturbations is nearly zero upstream of the nozzle contraction although the shock is weak and does not block the channel at any time. It is, at the moment, uncertain why the pressure amplification distribution exhibits such behaviour. A possible explanation involves the lower shock motion amplitude and the resulting lower flow velocity as the shock is in its most downstream position, which would result in the observed lower pressure amplification. This assumption however does not explain why the pressure amplification level is nearly zero upstream of the throat location. At 1000Hz, a similar pattern than the one described for the subsonic OP appears for the weak shock configuration. However, anticipating forthcoming observations, the phase-angle distribution for the fundamental (Figure N.30(d)) does not correspond to the superposition of different travelling waves. The observed amplification must therefore have another origin.

Figures N.31-N.33 (sub-figures c) present, for the strong shock configuration (with $P_{out} = 110kPa$), the evolution of the pressure amplification distribution of the fundamental as the perturbation frequency increases. At 100Hz, the pressure amplification distribution features a high level region immediately downstream of the shock. It was showed that the observation amplification was related to the intense BL thickening origi-
nating from the strong APG due to the superposed effect of the shock and the curvatures. At 500Hz, the tendency is completely reversed and an attenuation zone is now observed downstream of the shock. This attenuation may result from the interference of upstream propagating and reflected waves on the shock discontinuity. Indeed, in particular cases where the incident and reflected waves present a fixed phase relationship, constructive ("additive") or destructive ("subtractive") interference can create steady spatial patterns of high and low amplitude resulting pressure fluctuations. Besides, anticipating forthcoming observations, the phase-lag between the shock oscillations and the incoming pressure perturbations or separated region motion might very well play a role in the fact that the downstream region of the shock becomes either amplifier or attenuator of pressure perturbations. Simultaneously the amplification level along the shock location is lower due to the reduce amplitude of motion at high frequency. Consequently the shock does not choke the nozzle during part of the unsteady cycle and a (large) pressure amplification region corresponding to the acoustic blockage effect can be observed close to upper wall. At 1000Hz, a pressure amplification is observed in the region of low mean pressure gradients, close to the upper wall and corresponds to the acoustic blockage theory.

Figures N.34-N.36 (sub-figures c) present, for the strongest shock configuration with choked nozzle (for \( P_{\text{out}} = 102 \text{kPa} \)), the evolution of the pressure amplification distribution of the fundamental as the perturbation frequency increases. At 100Hz, the pressure amplification distribution features two high level regions downstream of the shock on both the upper and lower walls. At 500Hz, it is noteworthy that the amplification region on the upper wall has now turned into an attenuation region similarly to the situation on the lower wall for the previous OP. It should be reminded that a shock induced separation occurs on the upper wall for the current OP (with \( P_{\text{out}} = 102 \text{kPa} \)). This observation is another confirmation that the amplification, or attenuation at higher frequency, is directly related to the presence of the separation or to the BL thickness. On the lower wall however, there seems to be no sign of either pressure amplification or attenuation. At 1000Hz, no pressure amplification can be observed close to the upper and walls but exclusively downstream of the shock, in the center of the channel. It is believed that this amplification actually results from an over estimation of the turbulent kinetic energy production through the shock by the two equations \( k - \omega \) turbulent model. Indeed, in Wilcox's model, the term of production is directly proportional to the mean pressure gradients and therefore reaches a peak through a high discontinuity like a shock wave.

From a general overview, the pressure amplification of the first harmonic also features some pressure amplified regions outside the the shock motion extend. Although it is not really clear the phenomena from which originate such amplifications, it seems related to wave reflections and interactions between the shock, the geometry and the upstream travelling waves.

**Evolution of unsteady pressure phase-angle**

Figures N.25-N.27 (sub-figures d) present, for the subsonic OP (with \( P_{\text{out}} = 120 \text{kPa} \)), the evolution of the phase-angle distribution of the fundamental as the perturbation frequency increases. At 100Hz the phase-angle throughout the nozzle was described as a single travelling plane wave behaviour with a regularly decrease phase-angle as the per-
turbations propagate upstream. As the perturbation frequency increases, the perturbation wavelength decreases but show the same behaviour as previously. It should be noted that due to the shorter wavelength, the variations of the flow velocity due to local curvatures have a higher influence on the phase-angle distribution. At 1000Hz the very same trends can be observed. It should however be reminded that the phase-angle upstream of the shock location does not reflect any physical meaning since the amplitude level is almost zero.

Figures N.28-N.30 (sub-figures d) present, for the weak shock configuration (with $P_{out} = 116$kPa), the evolution of the phase-angle distribution of the fundamental as the perturbation frequency increases. Similarly to the previous OP, the phase-angle distribution corresponds, for all frequency values, to the propagation of a single plane wave.

Figures N.31-N.33 (sub-figures d) present, for the strong shock configuration (with $P_{out} = 110$kPa), the evolution of the phase-angle distribution of the fundamental as the perturbation frequency increases. It should first be reminded that, at this OP, the analysis of the pressure amplification revealed a strong influence of the perturbation frequency and, in particular, the change from amplified to attenuated of a large region downstream of the shock. At 100Hz the phase-angle distribution exhibits a significant change in a large region close to the lower wall downstream of the shock which corresponds to the mentioned amplified region. It was stated that the BL thickening was directly related to the observed pressure amplification and therefore change in the phase-angle distribution. At 500Hz, the phase-angle distribution features a large phase shift in the BL thickening. Although the origin of the strong phase shift is still unclear, it seems obvious that it is related to the pressure attenuation observed at this frequency. Finally, at 1000Hz, only a small phase shift can be observed in the region where the BL thickens downstream of the shock.

Figures N.34-N.36 (sub-figures d) present, for the strongest shock configuration (with $P_{out} = 102$kPa), the evolution of the phase-angle distribution of the fundamental as the perturbation frequency increases. As the pressure perturbation increases, for 500Hz, the observed phase-angle decrease downstream of the shock and related to pressure amplification disappears on the lower wall but is still observable on the upper wall.

7.2.2.3.3 Concluding remarks

This sub-section was dedicated to the description of the unsteady pressure distribution within the channel. A parametric study was conducted to establish the respective influence of the mean shock location, the perturbation amplitude and frequency.

shock loc influence: pressure amplif mainly explained by amplif of upstream propag wave in high flow velo region, even when shock block nozzle flow during part of unsteady cycle. Strong amplif observed in case of separated flow downstream of shock location. The coupling between the sudden BL thickening (leading to the so called post shock expan- sion), the shock oscillations (inducing the separation motion) and the acoustic blockage effect is assumed to produce the observed strong pressure amplification. The fluctuations
of the BL thickening induced by the shock oscillations lead to periodical variations of the post shock expansion, both in size and velocity level, which results in the amplification of upstream propagating pressure perturbations in the region downstream of the shock. The phase-angle distribution globally corresponds to a single travelling plane wave behaviour. A singular phase "decrease" was however observed for the strong shock configuration and correlated to the pressure amplification mechanism described above.

Similarly to previous observations, the influence of the perturbation frequency seems coupled to the shock position due to different flow characteristics. For the weak shock configurations, for instance, the unsteady pressure distribution corresponds to the acoustic blockage theory and pressure amplification are observed in high flow velocity regions. However at high frequency, new amplified zone appears and are believed to originate acoustic interactions within the channel. However, the phase angle distribution could not clearly verify this assumption and further analysis is required to fully understand the pressure amplification patterns. For strong shock configurations, the evolution of the unsteady pressure distribution, and in particular the amplification observed around the separate region, is clearly influenced by the perturbation frequency, either directly or indirectly through unsteady phenomena like the separated region motion. Curiously the perturbation frequency does not seem to have a huge influence on the phase-angle distribution except in region where a particular phase behaviour was already observed at low frequency.

7.2.2.4 Analysis of unsteady shock motion

7.2.2.4.1 Influence of shock location

Evolution of unsteady pressure amplification

Figures N.25(i), N.28(i), N.31(i) and N.34(i) present the shock motion amplitude as a function of the vertical position, at 100Hz, for the different OPs.

For the subsonic shock configuration (with $P_{out}^s=120\text{kPa}$), the curves presented in Figure N.25(i) are absolutely not representative of reality due to the disappearance of the shock during part of the unsteady cycle.

For the weak shock configuration (with $P_{out}^s=116\text{kPa}$), the amplitude of the fundamental contribution of the shock motion is fairly constant around $\pm 7\text{mm}$ in the vertical direction. It should be noted that the amplitude actually increases for vertical positions above $z=18\text{mm}$ (as seen at higher frequencies) but the shock motion amplitude at 100Hz is such that the shock disappear during part of the cycle for high vertical location. The first harmonic contribution on the shock motion is also quite constant around $\pm 11\%$ ($\pm 0.8\text{mm}$) of the fundamental signal.

For the strong shock configuration (with $P_{out}^s=110\text{kPa}$) the fundamental, which seems to take part up to 90% of the full shock motion composition, increases from the lower wall ($\pm =4\text{mm}$) towards the upper wall ($\pm =11.8\text{mm}$). It was stated during the analysis of the experimental shock motion that this was an effect of the mean flow gradients. The
lower the mean flow gradients are, the larger the amplitude of the shock motion. The non-linear contribution, although including only the first and second harmonics, seems to be constant throughout the channel height of about 10% of the linear (fundamental) contribution of the shock motion.

Finally, for the strongest shock configuration with the choked nozzle \( P_{\text{out}} = 102 \text{kPa} \), the fundamental contribution seems to represent most of the shock motion since both the first and second harmonic contributions are negligible. It is interesting to note that amplitude of the shock motion (fully represented by the fundamental signal) is now constant about ±5mm throughout the channel height. There seems to be two possible explanations for the observed shock motion "stabilization". First, as the shock now chokes the entire channel, there is no low mean flow gradients region (previously located at the junction of the upstream sonic line and the shock). Secondly, the newly appeared lambda shock system might simply block the natural oscillations of the shock or add some inertia to it. It is also fairly possible that both phenomena have a join effect on the shock motion.

Evolution of unsteady pressure phase-angle

Figures N.25(j), N.28(j), N.31(j) and N.34(j) present the shock motion phase-angle as a function of the vertical position, at 100Hz, for the different OPs.

For the subsonic shock configuration (with \( P_{\text{out}} = 120 \text{kPa} \)), although the shock motion amplitude was over estimated, the phase-angle distribution presented in Figure N.25(j) is based on the signal obtained during part of the unsteady cycle while the shock is appearing in the channel. Although the steady state OP is subsonic, the upstream propagating pressure perturbations are large enough to create a transonic flow over the bump with a sonic pocket 10mm high. As observed, the phase-angle distribution for the fundamental is quite constant which is not really surprising considering the size of the sonic pocket and denotes a rigid body motion. Both the first and second harmonics are also constant and shifted with 180° compared to the fundamental with means that the linear and non-linear contribution of the shock motion partly cancel each others.

For the weak shock configuration (with \( P_{\text{out}} = 116 \text{kPa} \)), the fundamental contribution of the phase-angle on the shock motion is also constant. It should be noted that contrarily to the subsonic OP, the phase-angle is now only plotted for the locations where the shock is observed during the entire unsteady cycle. The first and second harmonics, on the other hand, feature a slight evolution which results from a "wavy" motion for their respective contribution to the shock wave motion.

For the strong configuration (with \( P_{\text{out}} = 110 \text{kPa} \)), the fundamental contribution of the phase-angle is no longer constant and varies through the channel height from 72° at the lower wall to 50° towards the upper wall. This reflects the "wavy" motion from the foot to the "top" of the shock. It should be mentioned that the foot of the shock oscillates with a 22° phase advance compared to the "top" of the shock, which is noteworthy considering the shape of the shock and the fact that the upstream propagating perturbations reach the top of the shock slightly before they reach the foot. The first harmonic contribution also decreases with the vertical location and features an approximately 90° phase-lag.
between the foot and the "top" of the shock. Besides, the phase-lag between the fundamental and the first harmonic is more or less 180° throughout the channel height, which reflects a counter action of the respective contributions on the shock motion. However, the amplitude of the first harmonic is so low that the fundamental still represents the main contribution to the shock motion.

For the strongest configuration (with $P_{out}=102\text{kPa}$), the fundamental contribution of the phase-angle also slightly varies through the channel eight from 36° at the lower wall to 50° at the upper wall. Similarly to the previous OP, it reflects the "wavy" motion from the foot to the "top" of the shock. However, in this case, the foot of the shock oscillates with a 15° phase retardation whereas a phase advance was found for the previous OP. This situation seems fairly logical according to the steady state shock configuration. The upstream propagating plane waves reach the shock first at the upper wall, while it takes a slight delay to reach it at the lower wall.

### 7.2.2.4.2 Influence of perturbation frequency

**Evolution of unsteady pressure amplification**

Figures N.25-N.27 (sub-figure i) present, for the subsonic shock configuration, the evolution of the shock motion amplitude for different perturbation frequencies. However, at this OP, as mentioned above, the shock is only present during a part of the unsteady cycle and the amplitude values calculated by the post treatment tools are erroneous.

Figures N.28-N.30 (sub-figure i) present, for the weak shock configuration, the evolution of the shock motion amplitude for different perturbation frequencies. From a general perspective, the shock motion amplitude directly above the 2D bump decreases with the frequency (from $\pm 7.4\text{mm}$ at 100Hz down to $\pm 0.8\text{mm}$ at 1000Hz). The vertical trends of the shock motion on the other hand do not seem to be affected by the perturbation frequency and still increases with the vertical location of the shock. This observation is in agreement with the fact that low mean pressure gradients are exciting regarding the shock motion. It should be noted that, at high frequencies, as the amplitude of the shock motion is reduced so that the shock remains present over a longer vertical extent whereas it used to disappear during part of the cycle at 18mm at low frequency. It should be noted that the non-linear effects of the first harmonic increases with the frequency and reaches 11%, 27% and 40% of the linear contribution for 100Hz, 500Hz and 1000Hz respectively.

Figures N.31-N.33 (sub-figure i) present, for the strong shock configuration, the evolution of the shock motion amplitude for different perturbation frequencies. Similarly to the previous OP, the shock motion amplitude directly above the 2D bump decreases with the frequency (from $\pm 4\text{mm}$ at 100Hz down to $\pm 0.2\text{mm}$ at 1000Hz). Following the general trend for non-choked nozzles, the shock motion amplitude increases with the vertical extent of the shock. Furthermore, similarly to the weak shock configuration, the non-linear contribution of the first harmonic also increases with the perturbation frequency but to a lower extend. Indeed the amplitude level of the first harmonic only represents 9%, 11% and 27% of the fundamental contribution at 100Hz, 500Hz and 1000Hz respectively.
7.2. TWO-DIMENSIONAL NOZZLE

Figures N.34-N.36 (sub-figure i) present, for the strongest shock configuration with choked nozzle, the evolution of the shock motion amplitude for different perturbation frequencies. According to the general tendency regarding the influence of the perturbation frequency on the shock motion, the amplitude directly above the 2D bump decreases with the frequency (from ±4.5mm at 100Hz down to ±0.3mm at 1000Hz). It is noteworthy that the amplitude of the fundamental is slightly higher for the configuration with choked nozzle than without (previous OP). The main difference lies in the fact that, for any perturbation frequency, the shock motion amplitude does not increase any longer in the vertical direction. It is believed that either the absence of low mean flow gradients or the presence of the second lambda shock system on the upper wall "limits" the amplitude of motion of the shock wave.

Evolution of unsteady pressure phase-angle

Figures N.25-N.27 (sub-figure j) present, for the subsonic shock configuration, the evolution of the shock motion phase-angle for different perturbation frequencies. Although the shock motion amplitude is incoherent at this OP, the phase-angle distribution could be calculated from only part of the signal while the shock appears in the nozzle. Results show a constant phase-angle distribution for all perturbation frequencies.

Figures N.28-N.30 (sub-figure j) present, for the weak shock configuration, the evolution of the shock motion phase-angle for different perturbation frequencies. Similarly to the subsonic OP, the phase-angle distribution is fairly constant throughout the height of the channel for all perturbation frequencies. The frequency therefore does not seem to have any influence on the shock motion phase-angle distribution for subsonic or weak shock configurations. On another hand, the first harmonic phase-angle distribution is slightly more fastidious to analyze since it exhibits different trends for each perturbation frequency.

Figures N.31-N.33 (sub-figure j) present, for the strong shock configuration, the evolution of the shock motion phase-angle for different perturbation frequencies. As mentioned during the analysis of the shock location influence, the phase-angle distribution at low perturbation frequency slightly decreases with the vertical extent of the shock and leads to a phase-lag between the foot and the top of the shock which reflects the wavy motion of the shock. As the perturbation frequency increases, this phase-lag tends to increase to reach 72° and 216° at 500Hz and 1000Hz respectively. It should also be mentioned that the phase-lag, at this OP, systematically evolves towards a phase retardation of the "top" of the shock (phase advance for the foot). This observation can partly be explained by the smaller perturbation wavelength as the frequency increases but the general tendency is somehow curious since the pressure perturbations reach the "top" of the shock before the foot. Interestingly, the first harmonic phase-angle distribution does not seem as affected as the fundamental by the perturbation frequency and equally increases for each frequency from the foot to the "top" of the shock.

Figures N.34-N.36 (sub-figure j) present, for the strongest shock configuration, the evolution of the shock motion phase-angle for different perturbation frequencies. For this shock configuration, the phase-angle distribution contrasts with the previous OP and
increases from the foot to the "top" of the shock corresponding to a phase advance of the "top" compared to the foot of the shock. According to the steady state shape of the shock, this situation seems logical since the downstream perturbations reach first the shock on the upper wall. Similarly to the previous OP, the phase lag is magnified as the perturbation frequency increases. This result is also believed to originate from the shorter perturbation wavelength.

7.2.2.4.3 Comparison with experiments

Evolution of unsteady pressure amplification

Figure 7.12 presents the comparison between experimental visualizations and numerical simulations regarding the shock motion amplitude for the weak shock configuration at two different frequencies.

At low frequency (left sub-figure), the amplitude of motion between experiments and numerical calculations (both inviscid and turbulent) is found to be quite different. Indeed, the 2D RANS calculation presents the largest shock motion with an amplitude of \( \pm 7.2 \text{mm} \) at \( z=12 \text{mm} \) compared to the Euler simulation (\( \pm 6.5 \text{mm} \)) and experimental visualization (\( \pm 3.8 \text{mm} \)). Although a high shock motion amplitude was expected for Euler calculations, it is very surprising to observe an even higher amplitude for RANS simulations. Indeed previous analysis on unsteady pressure amplification on the bump surface showed a fairly good agreement between 2D RANS calculation and experimental results. However, the comparison on the pressure amplification between numerical and experimental results also pointed out a difference in the amplitude of perturbation. Indeed, on the experimental side, it was found that the same elliptical rod would create different amplitude of pressure perturbations depending on the operating conditions. Consequently, although the experimental-numerical comparison on pressure amplification results was fairly good, the amplitude level of the perturbations was also quite different. It seems therefore that, although the perturbations amplitude does not have a huge influence on the pressure amplification, it does have a significant impact on the shock motion through the amplitude level of the upstream propagating perturbations.

It should be noted that the vertical extend of the shock motion decreases with the amplitude. The larger the amplitude of motion is, the shorter the vertical extent of the shock motion. This actually corresponds to the fact that, for a certain vertical position, the shock disappears during part of the unsteady cycle is the amplitude of motion is too large.

At higher frequency (500Hz) the shock motion amplitude for the 2D RANS (\( \pm 4 \text{mm} \) at \( z=12 \text{mm} \)) and the experiments (\( \pm 3.4 \text{mm} \)) do not feature such a large difference any more. This result illustrates the fact that the shock does not respond linearly to large amplitude pressure perturbations as the frequency increases. Besides, considering the evolution of the shock motion amplitude for Euler calculations, it seems that this non-linear behaviour is a consequence of viscous interaction therefore between the shock and the boundary layer. It should finally be noted that, for both experimental and numerical cases, the similar vertical tendency was observed. The amplitude of the shock motion
increases as the mean flow gradients decreases.

Figure 7.12: Experimental-Numerical comparison of the unsteady shock motion amplitude for the weak shock configuration at 100Hz and 500Hz

Similarly, the shock motion amplitude for experimental visualizations and numerical simulations is presented in Figure 7.12 for the strong shock configuration at two different frequencies.

It is interesting to note that, at 100Hz, the shock motion amplitude of the 2D RANS simulations agrees fairly well with the experiments. Indeed, for this shock configuration, the amplitude level of the upstream propagating pressure perturbations is similar (about ±2.32kPa) for both the experiments and the simulations. The Euler calculation however exhibits a slightly larger shock motion amplitude probably due to the lack of viscous dissipation in a quasi-steady case.

At 500Hz, the agreement between 2D Euler, 2D RANS and experiments is fairly good and exhibits a more "potential" behaviour of the shock motion compared to the case with weak shock configuration.

Figure 7.13: Experimental-Numerical comparison of the unsteady shock motion amplitude for the strong shock configuration at 100Hz and 500Hz

Evolution of unsteady pressure phase-angle

Figure 7.14 presents the comparison between experimental visualizations and numerical simulations regarding the shock motion phase-angle for the weak shock configuration at two different frequencies.
At low frequency (left hand side figure) the phase-angle distribution from Euler and viscous calculations features a 14° phase shift with the experiments, which can be considered as a fairly good agreement. Especially there is no phase-lag between the foot and the "top" of the shock, which is typical of a quasi-steady behaviour.

At higher frequency (right hand side figure) the phase-angle distributions differ for all cases and is probably due to a slightly different propagation velocity. Indeed the shock motion phase-angle is directly related to the upstream propagating perturbation wavelengths, which is itself a function of the flow velocity downstream of the shock location. As a result, the presence or not of BLs, the difference in BL thickening intensity or a slight difference in the shock location have an impact on the shock motion phase-angle distribution. The decrease of the perturbations wavelength magnifies the differences on the phase-angle values. It is interesting to note however that the vertical phase-angle distribution also differ between the different cases. In particular for the Euler calculation where the a ~30° phase-lag can be observed between the foot and the "top" of the shock.

![Figure 7.14: Experimental-Numerical comparison of the unsteady shock motion phase-angle for the weak shock configuration at 100Hz and 500Hz](image)

For the strong shock configuration presented in Figure 7.15, the phase-angle distributions for numerical simulations and experiments at low frequency (100Hz) collapse almost perfectly. It should be noted that the influence of the outflow velocity on the phase-angle distribution is minimized by the long wavelength at low frequency. Similarly to the weak shock configuration, the phase-angle distribution is constant, reflecting a quasi-steady shock motion behaviour at low frequency.

At higher perturbation frequency, the phase-angle distributions for both the 2D RANS simulation and experimental visualization exhibit the same trend and decrease with the vertical shock position, whereas it remains constant throughout the channel height for the Euler calculation. As previously discussed, this observed phase-lag (of about 75°) reflects a phase advance of the foot of the shock compared to its motion at the "top", which therefore seems to result from viscous effects.

### 7.2.2.4.4 Concluding remarks

This sub-section was dedicated to the description of the unsteady shock motion throughout the channel height. Both amplitude and phase-angle numerical distributions were described and compared to experimental results. A small parametric study was conducted.
Figure 7.15: Experimental-Numerical comparison of the unsteady shock motion phase-angle for the strong shock configuration at 100Hz and 500Hz

to establish the respective influence of the mean shock location and the perturbation frequency.

From a general perspective, the mean shock location was found to have, through the mean flow gradients distribution, quite an important influence on the shock motion amplitude. Indeed, large amplitude motion was observed for low mean pressure gradients, observed either in weak shock configuration or at the junction point between the sonic line and the shock in stronger shock configurations. In particular, the absence of mean flow gradients in choked nozzle flow configurations (or possibly the double lambda shock system configuration) seems to inhibit large amplitude shock motion. Additionally, a decrease of the non-linear behaviour of the shock motion was observed as the shock moves downstream, gets stronger. This reflects a higher non-linear behaviour for weak shock configurations. The analysis of the phase-angle distribution revealed, on one hand, a rigid body motion for weak shock configurations and, on the other hand, a "wavy" shock motion for strong shock configurations illustrating a phase-lag between the foot and the "top" of the shock.

Similarly, the influence analysis of the perturbation frequency on the shock motion revealed that the amplitude of motion globally decreases with the frequency, except for the OP with subsonic mean flow field for which the tendency is interestingly reversed. Additionally an increase of the non-linear behaviour of the shock motion was observed for all different shock configurations as the perturbation frequency increases. Furthermore, the analysis of the phase-angle distributions revealed that, for weak shock configurations, the perturbation frequency does not seem to have any influence on the observed constant phase-angle distribution. For strong shock configuration or choked nozzle flow however, the perturbation frequency seems to accentuate the phase-lag between the foot and the "top" of the shock.

Finally, the comparison with experimental results, 2D RANS and inviscid calculations showed, for weak shock configurations, a non-linear response of the shock to large amplitude perturbations (due to viscous effects) as the frequency increases. For the strong shock configuration, a much more diffused influence of viscous effects was observed with an increasing perturbation frequency. The comparison of the phase-angle distributions revealed, for both shock configurations, a quasi-steady behaviour illustrated by a rigid body motion. For higher frequency however, the weak shock still exhibited a rigid body motion whereas the strong shock configuration clearly featured a phase-lag between foot
and "top" of the shock, for both experimental results and viscous simulations. The different behaviour of the inviscid calculation suggested that this phase-lag as a consequence of viscous interaction between the shock, the boundary layer and the upstream propagating perturbations.

It should be mentioned that, in general, the shock motion phase-angle value cannot be directly analyzed without considering at the same time the phase-angle of the pressure perturbations. Indeed, the shock motion originating from the incoming pressure fluctuations, the phase-angle of the pressure perturbations downstream of the shock therefore depends primarily on the shock location and the perturbations wavelength. However, anticipating forthcoming results, it is sometime observed that the incoming pressure perturbations and the shock motion feature, for some reasons, a phase-lag which makes the full prediction of the shock motion, and therefore of the overall aerodynamic force, rather difficult.

### 7.2.2.5 Analysis of shock motion and pressure perturbations correlation

Similarly to experiments, the phase-lag between the numerical shock motion above the bump surface (at $z=12\text{mm}$) and the unsteady pressure phase-angle downstream of the shock has been plotted (for the fundamental contribution) in Figure 7.4 as a function of the perturbation frequency. A value of $180^\circ$ was taken out from the calculated phase-lag to account for the steady state phase-lag between the the shock and the downstream pressure (the shock moves upstream when the downstream pressure increases). As a result, a negative phase-lag value corresponds to a delay in the shock response to the incoming pressure perturbations.

For the subsonic flow and weak shock configurations, the phase-lag is negative and increases in absolute value with the frequency. This result reflects the fact that the shock motion is more and more delayed compared to the incoming pressure perturbations as the frequency increases. A zero phase-lag is however observed at $1000\text{Hz}$ for the subsonic flow configuration and is at the moment unexplained. For stronger shock configurations, it is striking to note that the phase-lag is also increasing with the frequency, but features positive values which correspond to an advance of the shock motion compared to the downstream pressure perturbations. Although those results are completely not understood, a possible explanation might be that the phase-angle downstream of the shock results from the superposition of upstream propagating pressure waves and reflected waves on the shock discontinuity which would alter the "total" phase-angle value of the fundamental contribution of the unsteady pressure distribution.

A comparison between experimental and 2D RANS numerical results has also been plotted in Figure 7.16 (right hand side sub-figure). Although the results do not collapse nicely, they however show some similarity trend for the weak shock configuration. The tendency showed on the numerical results for the strong shock configuration is rather difficult to interpret and would require further investigation. It should however be noted that the presented data is issued from visual reading of the pressure and shock motion phase-angle and therefore are entailed of a certain error. Even with this visual method
7.2. TWO-DIMENSIONAL NOZZLE

Figure 7.16: Numerical phase-lag between shock motion and unsteady pressure perturbations downstream of shock location for different operating conditions

however, the trends should be correct. The observed results suggests a more complex interaction.

7.2.2.6 Analysis of the Time-Averaged Versus steady state pressure distribution on bump

Figures N.25-N.36 (sub-figures b) present the unsteady time-averaged versus the steady state pressure distributions on the bump in the center of the channel (at y=50mm) for each OP. Similarly to experimental observations, the general tendency seems to place the "time-averaged" shock upstream of its steady state location. Confirmed by experimental measurements, this observation is explained by considering that the unsteady oscillations of the shock wave create higher losses than a stationary (steady state) shock. Indeed, as the shock periodically moves downstream of its steady state location, it generates a higher loss level over the entire cycle\(^2\). As a result the pressure gradient between the shock location and the outlet plane is higher, and since the back pressure is fixed the pressure immediately downstream of the shock is increased and the averaged shock position is slightly moved upstream.

This explanation is also in agreement with the observation of larger differences between time-averaged and steady state pressure distribution at low perturbation frequency at which higher shock motion amplitudes were observed.

7.2.2.7 Analysis of the separated region motion

The analysis of the unsteady pressure distribution on the bump outlined, for the strong shock configuration only, a very singular behaviour of both the pressure amplification and phase-angle distribution downstream of the shock. Besides, the comparison between experiments, inviscid and RANS calculations clearly showed that the observed behaviour resulted from a viscous interaction. It was furthermore observed that for this particular OP, and this OP only, the flow field features a shock induced separation and therefore a quite important BL thickening. It was finally postulated that the observed pressure am-

\(^2\)Since the loss generation is not a linear phenomenon with the shock location and non-linearly increases as the shock moves downstream and gets stronger, the resulting time averaged losses over one unsteady cycle is higher than the steady state value.
Table 7.2: Characteristics of the reattachment point motion for the strong shock configuration

<table>
<thead>
<tr>
<th>$P_p$</th>
<th>100Hz</th>
<th>500Hz</th>
<th>1000Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shock motion amplitude</td>
<td>±3.85mm</td>
<td>±1.22mm</td>
<td>±0.15mm</td>
</tr>
<tr>
<td>Reattachment point motion amplitude</td>
<td>±19.5mm</td>
<td>±8mm</td>
<td>±1.34mm</td>
</tr>
<tr>
<td>Reattachment point Amplification factor</td>
<td>5</td>
<td>6.5</td>
<td>8.9</td>
</tr>
<tr>
<td>Reattachment point motion phase-lag</td>
<td>$18^\circ$</td>
<td>$88^\circ$</td>
<td>$90^\circ$</td>
</tr>
</tbody>
</table>

In order to investigate the unsteady motion of the separated zone, the motion of the separation and reattachment points was plotted, in Figures 7.17-7.18 for each perturbation frequency, as a function of the shock motion for the 2D RANS numerical simulation. The advantage to display the separation versus the shock motion is to be able to observe both the amplitude of motion and the phase-lag between the separation/reattachment points and the shock oscillations.

At 100Hz in Figure 7.17, the separation point clearly oscillates with the same amplitude and nearly in-phase (more or less a few degrees) with the shock wave. This behaviour does not seem to change with the frequency and confirms the idea of a shock induced separation. On the other hand, the reattachment seems to oscillate with a much larger amplitude and a certain (low) phase-lag with the shock. As the perturbation frequency increases the shock motion decreases but, comparatively, the amplitude of oscillation of the reattachment point is magnified as reported in Table 7.2.

Indeed, as the perturbation frequency increases, although both the shock and reattachment point motion amplitudes decrease, the ratio between them actually increases. Furthermore, the phase-lag between the reattachment point and the shock motion also increases with the frequency but seems to reach a limit at $90^\circ$ for high frequencies.

Although an interesting behaviour of the separation region could be observed as a function of the perturbation frequency, no clear relation between the separated region motion and the unsteady pressure distribution on the bum could be established. Besides,
7.2. TWO-DIMENSIONAL NOZZLE

the experimental determination of the separation motion would be required in order to confirm or deny the observed numerical tendencies. Finally, further RANS calculation would also be required in order to draw a clear tendency regarding the phase lag between the reattachment point and the shock motion and determine if a phase "locking" phenomenon occur at a high perturbation frequency.

7.2.2.8 Analysis of acoustic propagation in channel

It was suggested, at several occasions during the analysis of the unsteady pressure distribution on the bump for both experimental and numerical results, that the phase-angle and pressure amplification distributions downstream of the shock location was the result of a superposition of upstream and downstream propagating waves with different amplitude and phase-angle distributions. The suggestion basically resulted from the observation of a pressure amplification and phase-angle "retardation" or increase immediately downstream of the shock location. This observation was made on both experimental and numerical results at low frequency and for all OPs, but with a varying intensity depending on the strength of the shock wave. In particular, for the strong shock configuration\(^3\), the pressure amplification and phase-angle distributions suggested, at further higher perturbation frequency, a complex signal composition of the unsteady pressure distribution downstream of the shock with phase shift and pressure amplification factor varying with the frequency (amplification or attenuation). The presence, at this particular OP, of a shock induced separation with a large BL thickening suggested a strong interaction between upstream propagating pressure disturbances, the BL thickening and downstream propagating waves issued from the reflection of incident acoustic waves on the shock.

At the lowest presented perturbation frequency, the typical wavelength is about 2.4m and only plane waves can propagate within the channel. For higher perturbation frequencies the wavelength is smaller and multiple waves can propagate in the vertical or spanwise directions, making the physic of the interaction and the analysis much more complex. As a result, with the objective to deepen the understanding of the unsteady pressure distribution on the bump, a one dimensional acoustic decomposition was performed on the 2D RANS numerical results using the flow and sound velocity values from

\(^3\)with \(P_{\text{out}}^{\text{s}}=110\text{kPa}\) for RANS calculations, and \(P_{\text{out}}^{\text{s}}=116\text{kPa}\) for experiments
Figure 7.19: Acoustic decomposition of the unsteady pressure distribution on 2D bump surface for the subsonic flow configuration at 100Hz within the channel. Performed between a reference location situated at the outlet plane and different locations within the channel, the decomposition only concerns the first mode (fundamental) of the DFSD on unsteady pressure distribution on the bump. According to Equation 7.1, the amplitude $A_{up}$ and phase angle $\Phi_{up}$ of a single upstream travelling wave should be constant if the theory of single travelling plane waves is correct (since time and spatial fluctuations are correlated in travelling waves).

$$P(x, t) = A_{up} e^{i(\omega t + \frac{\omega (x-x_{out})}{c_U}) + \Phi_{up}} + A_{dwn} e^{i(\omega t - \frac{\omega (x-x_{out})}{c_U}) + \Phi_{dwn}} \text{ with } x < x_0 \quad (7.1)$$

Figure 7.20: Acoustic decomposition of the unsteady pressure distribution on 2D bump surface for the weak shock configuration at 100Hz

The calculated amplitude and phase-angle distributions of the respective upstream and downstream single travelling plane waves have been plotted, for each shock configuration, in figures 7.19-7.22. At the outlet, and for all different OPs, the acoustic field is mainly (at 90%-95%) composed by upstream propagating waves. It seems however that the proportion of downstream propagating waves slightly increases for the stronger shock configurations (Figures 7.21 and 7.22 on the left hand side). It is noteworthy that both the upstream and downstream propagating wave amplitudes are fairly constant between the shock location and the outlet plane. It is also interesting to note the increase of both amplitude levels over a long area downstream of the shock location for the strong shock configuration (with $P_{out} = 110kPa$ in Figure 7.21). For all other OPs however, the amplitude of the upstream and downstream propagating waves is fairly constant until very near the shock location. Especially, the downstream propagating waves seem to fade down rather quickly.
The analysis of the phase-angle distributions is more delicate but some parts can however be understood without ambiguity. At the outlet for instance, the phase-angle distributions for both the upstream and downstream propagating waves are quite constant which confirms, at this location, the assumption that the outlet acoustic field is composed by single plane waves. It should be mentioned that the small deviation very close to the outlet plane originates from inaccuracies in the decomposition method since the amplitude and phase of a long wavelength are being estimated over a very short distance. The phase-angle behaviour is much more ambiguous and difficult to interpret closer to the shock location, especially for downstream propagating plane waves. First of all, it is noteworthy that for the subsonic flow, weak shock and choked nozzle flow configurations (Figures 7.19 and 7.20 respectively), the phase-angle distribution of the upstream propagating waves is fairly constant between the outlet and the shock location. For the stronger shock configuration (Figures 7.21 and 7.22 respectively), the phase-angle distribution presents some slight variations in the vicinity of the shock. Globally, when looking in the propagation direction, i.e. towards the inlet (respectively the outlet) for upstream (respectively downstream) propagating waves, a decreasing phase-angle value corresponds to a shortening of the wavelength and therefore to a decrease of the propagation velocity. Consequently, depending on the mean flow velocity and the propagation velocity (equal to $c - U$ and $c + U$ for upstream and downstream propagating perturbations respectively) the phase-angle will change accordingly. However for downstream propagating perturbations the velocity distribution cannot explain the phase-angle variations observed in the acoustic decomposition and therefore suggests either that the acoustic composition model does not take into account higher modes or that other types of waves also take part in the total acoustic field.
7.3 Three-dimensional Nozzle

7.3.1 Experimental results

7.3.1.1 Introduction

The following section presents the unsteady results related to the experimental investigation of the 3D nozzle. The different unsteady operating conditions are presented in Table 5.9 in the experimental model (chapter 5).

Figures N.37-N.44 present the unsteady pressure measurement performed on the 3D bump at several spanwise locations and different perturbation frequencies. Since only one steady state OP was investigated, the analysis simply presents the pressure amplification and phase-angle distributions as a function of the spanwise location, with a small parametric study on the influence of the perturbation frequency.

7.3.1.2 Analysis of unsteady pressure distribution through the channel width

Evolution of unsteady pressure amplification

Figures N.38(left hand side) present the pressure amplification at different spanwise locations, namely y=20mm, 35mm, 50mm, 65mm and 80mm, for the lowest perturbation frequency (50Hz).

Clearly, the upstream propagating perturbations are not strongly amplified (up to 1.2 only) at low frequency, for any spanwise location. Although, this is in agreement with the experimental unsteady results observed on the 2D bump for the weak shock configuration (with $P_{out}^{s}=112$kPa in Figure N.9), it is surprising not to find a stronger influence of the spanwise location. Indeed, experimental pressure measurements on the 2D bump showed a quite important amplification of the perturbations for a strong shock configuration (with $P_{out}^{s}=106$kPa in Figure N.1), and it would therefore have been logic to find a smooth evolution of the pressure amplification with the spanwise location as the shock is stronger on the high curvatures side of the 3D bump. A possible reason, however, might be that the shock is still not strong enough, the pressure amplification being maximum when the flow features a lambda shock system and a strong BL thickening (resulting from a shock induced separation as observed on numerical results on the 2D bump). On the other hand, the amplification level underneath the shock does vary in the spanwise direction. It is interesting to note that the amplitude of the first harmonic represents around 40% of the fundamental contribution, and therefore 40% of the outlet perturbations amplitude\(^4\) since the fundamental amplification factor is about 1. The reason for such high level non-linear contribution of the unsteady pressure distribution on the bump, for a quasi-steady case, is however unclear.

At 100Hz, in Figures N.40(left hand side), a higher pressure amplification (up to 1.5) can be observed downstream of the shock location, as well as a slight spanwise evolution

\(^4\)Indeed, all harmonic contributions of the unsteady pressure signal were respectively divided by the amplitude of the fundamental mode of the outlet static pressure signal
of the shape of the distribution (the level being constant). Indeed, whereas the pressure amplification regularly increases to a level of 1.5 downstream of the shock location on the low curvature side on the bump (y=20mm), a slight decrease of the amplification can be observed immediately downstream of the shock on the high curvature side (y=80mm). This behaviour was also observed on the 2D bump, for the strong shock configuration (with \(P_{\text{out}}^{\text{ss}}=106\text{kPa}\)), at low frequency (Figure N.2), and seems therefore related to the strength of the shock or possibly to a consequence of a stronger pressure rise (larger BL thickening, formation of upstream oblique shocks, etc.). Besides, for this perturbation frequency, the amplitude of the first harmonic has decreased to 10\% of the fundamental contribution compared to a level of 40\% at 50Hz.

At 250Hz, in Figures N.42(left hand side), a higher pressure amplification can be observed downstream of the shock (from 2 at y=20\%, up to 2.5 for y=80\%). It is interesting to note the spanwise evolution of both the level and length of the amplification region downstream of the shock. Indeed, the extent and level of the pressure amplification zone get shorter but higher towards the high curvature side of the 3D bump (as the bump becomes smaller and thicker). This could be interpreted as an effect of the acoustic blockage theory for which upstream propagating pressure perturbations are gradually amplified as they travel through continuously higher flow velocity regions until they reach the shock location. Furthermore, the amplitude of the first harmonic also increases downstream of the shock location up to 45\% of the fundamental contribution level.

A similar amplification behaviour of the fundamental can be observed at 500Hz in Figure N.43(left hand side). The amplification level is even higher (from 2.5 at y=20\%, up to 3 for y=80\%) than for the previous perturbation frequency. The extent and level of the amplification region related to the spanwise variation of the geometry curvatures seems also to correspond to the description presented above. On the low curvature side of the 3D bump (at y=20mm), and thus for low mean flow gradients, the amplification zone is longer (from x=55mm to 115mm) but features a lower level (A=2.2 at x=55mm), whereas on the high curvature side, the amplification zone is smaller (from x=50mm to 85mm) but features a higher level (A=3.4 at x=50mm). This result seems to be related to the amplification of upstream propagating pressure waves in high flow velocity regions. Indeed for this OP, the shock configuration is weak and the pressure jump through the shock is relatively small. This results in a high flow velocity region downstream of the shock location, which logically extents farther on the low curvature side of the bump. Furthermore a strong non-linear behaviour of the unsteady pressure distribution is observed in previously described amplification region downstream of the shock. Anticipating forthcoming observation on numerical simulation at the same perturbation frequency, this behaviour might originate the formation of a second sonic pocket during the unsteady cycle, while the shock is at its most upstream position and starts to move downstream again. No experimental observation or measurement could however verify this theory.

From a general perspective, the pressure amplification was observed to increase with the perturbation frequency and feature an important non-linear behaviour.

Evolution of unsteady pressure phase-angle
Figures N.37(right hand side) present the pressure phase-angle distribution at different spanwise locations for the lowest perturbation frequency (50Hz). On the low curvature side of the 3D bump, the phase-angle distribution varies linearly with the streamwise position outside the shock motion extent. Underneath the shock location, a small phase shift can be observed. It is interesting to note the spanwise evolution of both the phase-angle distribution and phase shift though the shock. Indeed, both the slope of the phase-angle distribution and the phase shift slightly increase towards the higher curvature side of the bump. A rough estimation of the slope of the phase-angle distribution allowed the calculation of the perturbation wavelength of about 2m to 2.25m corresponding to a relative propagation velocity \((c-U)\) of about 100m/s, which seems to be a fairly good approximation downstream of the shock location. The increasing phase shift underneath the shock with the spanwise location seems also logical and in agreement with previous experiments on the 2D bump. Indeed, it was found that the phase shift underneath the shock location was increasing with both the perturbation frequency and the strength of the shock (steady state shock location).

At 100Hz, in Figures N.39(right hand side), the same streamwise and spanwise evolution trends of the phase-angle distribution can be observed, with the following slight different however. The phase shift seems to have increased with the perturbation frequency and the phase-angle distribution feature a slight "retardation" on the high curvature side of the 3D bump \((y=80\text{mm})\). This change in the slope of the phase-angle was also observed at low perturbation frequency, for both weak and strong shock configurations on the 2D bump. It was besides suggested that this phase-angle distribution might result from the superposition of impinging and reflected pressure waves on the shock.

At 250Hz, in Figures N.41(right hand side), the phase-angle distribution downstream of the shock location is fairly linear which denotes a regular wave propagation. Similarly to previous results, the phase shift underneath the shock increases in the spanwise direction. The main difference at this frequency is the presence, upstream of the shock, of a phase variation which level seems to increase with the spanwise location. This behaviour is believed to originate from the supersonic flow on the converging part of the nozzle. Indeed, only downstream propagating waves are convected in this area. However, the phenomenon was not observed on the 2D bump although a similar sonic pocket was developing upstream of the shock shock location.

For 500Hz, in Figures N.43(right hand side), the very same trends as at 250Hz can be observed. First, a slope increase of the phase-angle distribution (simply due to the higher perturbation frequency), an increasing phase shift underneath the shock in the spanwise direction, and a phase variation upstream of the shock probably due to the extent of the sonic pocket on the converging part of the nozzle.

For all different perturbation frequencies, it is interesting to note that the perturbations do propagate upstream, at a slightly lower amplitude though, since the nozzle is not choked.

---

\(^5\text{This effect is even more pronounced at higher perturbation frequency}\)
7.3. THREE-DIMENSIONAL NOZZLE

7.3.1.3 Analysis of the unsteady pressure phase shift underneath shock

Figure 7.23 presents an overview, for different spanwise locations, of the phase shift underneath the shock location as a function of the perturbation frequency.

![Figure 7.23: Experimental phase shift underneath shock location for different spanwise positions in 3D nozzle](image)

It is noteworthy that the phase shift underneath the shock varies "quite" linearly with the perturbation frequency. This result is in complete agreement with the observations performed on the 2D bump. Furthermore, the phase shift seems to increase with the spanwise location, which is also in agreement with results on the 2D bump obtained for different mean shock location (and therefore shock strength). This phenomenon is maybe better observed in Figure 7.24 further below presenting the comparison with numerical results. It should be noted, however, that a slightly different tendency can be observed between the phase shift distribution at the spanwise locations \(y=50\text{mm}\) and \(y=65\text{mm}\). Indeed, it seems that the phase shift features higher values in the middle of the channel (\(y=50\text{mm}\)) than at \(y=65\text{mm}\). It is indeed possible that the unsteady phase-angle distribution might be modified in such way in the case of a second shock wave appearing during the unsteady cycle, as mentioned previously. It is however uncertain whether such non-linear flow behaviour really happens experimentally.

7.3.1.4 Concluding remarks

The pressure amplification distribution on the 3D bump was found to mainly correspond to the characteristic behaviour of the weak shock configuration as analyzed on the 2D bump. In similarity with a quasi-steady flow behaviour, no large amplification could be observed downstream of the shock location at low perturbation frequency, whereas a moderate amplification could be observed at higher frequency. There results are in agreement with the analysis on the 2D bump where limited pressure amplification levels were observed in weak shock configurations. Also observed during the flow investigation within the 2D nozzle, the phase-angle distribution features a very slight "retardation" downstream of the shock believed to originate the superposition of impinging and reflected pressure waves on the shock wave front. Additionally, it was observed an increasing phase
shift on the surface underneath the shock location with both the perturbation frequency and the spanwise location as the shock strength varies, as a function of the curvatures, in the spanwise direction.

Considering the similarities with the weak shock configuration case on the 2D bump, it would be interesting to also compare the unsteady pressure distribution for a strong shock configuration, like the one obtained for a back pressure value of 112kPa for instance. Indeed, although this OP does not feature a middle channel separation, the extreme shock configuration (while putting the back pressure into oscillation) would certainly (according to the steady results obtained for $P_{\text{out}}=110\text{kPa}$) feature a large separation and therefore a large BL thickening with a progressive evolution in the spanwise direction. This case would therefore be interesting to verify whether the BL thickening is a major parameter in the pressure amplification phenomenon in unsteady transonic flows.

Finally, it should be mentioned that due to physical constrains the spatial resolution of the unsteady pressure measurements could not be better than 5mm between each pressure taps. This coarse resolution therefore limits the analysis especially in regions underneath or downstream of the shock wave. The solution to such inconvenient would be, in future experimental investigations, to use a newly developed pressure sensitive paint with response time down to a few nano-seconds. Although this technique is still at the moment under development and only features single point measurement, it could soon develop and offer the possibility to map the entire surface in a few seconds, allowing real time processing and a much more interacting research possibility.

### 7.3.2 Numerical results and comparison with experiments

#### 7.3.2.1 Introduction

Among the various 3D steady state RANS calculations presented in chapter 6, one single configuration was selected due to flow similarities with the design objectives presented in Figure 5.4 in chapter 5. Similarly to 2D RANS calculations, the shock wave was put into an oscillating motion by imposing a sinusoidal plane back pressure fluctuation. Table 5.9 presents the unsteady operating conditions for the selected numerical steady state configuration for the 3D nozzle.

<table>
<thead>
<tr>
<th>Steady state OP</th>
<th>Unsteady Back pressure perturbations</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_{\text{inlet}}$</td>
<td>$F_p$</td>
</tr>
<tr>
<td>[kPa]</td>
<td>[Hz]</td>
</tr>
<tr>
<td>160</td>
<td>500</td>
</tr>
</tbody>
</table>

Table 7.3: Unsteady operating conditions for 3D RANS calculations in 3D nozzle

The following paragraphs present the unsteady numerical results within the 3D nozzle with comparison to the unsteady pressure measurements performed on the 3D bump. A great advantage of numerical simulations is to be able to visualize aerodynamic quantities which would not be directly measurable with experimental techniques. The analysis of the unsteady pressure distribution as well as the shock motion within the channel are
therefore also presented in the following.

7.3.2.2 Analysis of the unsteady pressure distribution through channel width

Evolution of unsteady pressure amplification

Figures N.46(left hand side) present the pressure amplification distribution on the surface of the 3D bump for different spanwise locations. Those results can directly be compared with experimental data previously analyzed and illustrated in Figure N.44.

Globally for all spanwise positions, a slight pressure amplification occurs downstream of the shock location. Similarly to experimental results, the level and extent of the pressure amplification region depends on the spanwise position. On the low curvature side of the 3D bump (at y=20mm), and thus for low mean flow gradients, the amplification zone is longer (from x=55mm to 80mm) but features a higher amplification level (A=2.2 at x=55mm). Contrarily, on the high curvature side, the amplification zone is smaller (from x=45mm to 65mm) but features a lower level (A=3.5 at x=45mm). A gradual evolution can be observed between the two spanwise positions. This result can be explained by the fact that, at this OP, the shock is "quite" weak (compared to a lambda shock system) and therefore features a small pressure rise. As a result the downstream velocity is close to the sonic value, which creates an amplification of the upstream propagating pressure perturbations.

As mentioned, experimental results showed the same trends, but comparatively, the extent of the amplification region for 3D RANS calculation is much smaller than observed in experimental results (Figures N.44(left)). This difference can be simply explained by the difference with the steady state velocity distribution downstream on the shock location. Indeed, as observed in Figure M.32 (Appendix M) the pressure distribution at all spanwise location is higher for the 3D RANS simulation than in the experimental case. This reflects a lower flow velocity region downstream of the shock, which explains the observed lower extent of the pressure amplification area.

Evolution of unsteady pressure phase-angle

Figures N.46(right hand side) present the phase-angle distribution of the unsteady pressure on the surface of the 3D bump for different spanwise locations. For all spanwise locations the phase-angle decreases between the outlet plane and the shock location with the same rate, which denotes the same propagation velocity and, more or less, the same flow velocity distribution at this OP. Thereafter, similarly to experimental results on the 3D bump, an increasing phase shift occurs with the spanwise position underneath the shock location. Finally, between the shock location and the inlet plane, the phase-angle further decreases. From a global perspective, the phase-angle distribution is characteristic of a single upstream travelling wave, which is in agreement with results obtained in 2D RANS calculations at the same perturbation frequency, for a similar weak shock configuration.
The comparison with experimental results presented in Figure N.44 (right hand side) shows however a major difference of the phase-angle distribution upstream of the shock. Indeed, the 3D RANS simulation does not predict at all the phase increase upstream of the shock, nor its variation with the spanwise location observed in experimental pressure measurements. The reasons of such a difference are unfortunately at the moment still unclear. Apart from this difference, the phase-angle distribution downstream of the shock is fairly similar to experimental results. It should be reminded that the phase difference observed at the outlet between experimental and numerical results probably originates from the fact that the experimental reference value was measured on the side walls whereas pressure measurements were performed on the lower wall. Due to the outflow non-uniformity, the back pressure perturbations generated using a rotating rod further downstream of the test section seems to propagate with a different velocity depending on the location in the channel.

**Evolution of phase shift underneath the shock**

Figure 7.24 presents the phase shift values observed underneath the shock location as a function of the spanwise location for both the 3D RANS calculation and experiments performed at different perturbation frequencies.

As mentioned in the previous subsection, the phase shift underneath the shock increases with the spanwise location, i.e. the shock strength. It is noteworthy that the comparison between experimental and numerical results gives a fairly good agreement. It is however unclear why experimental results show a lower (absolute) value at \( y = 65 \text{mm} \). As mentioned during the analysis of experimental pressure measurements on the 3D bump, this might be the effect of a flow phenomenon occurring at that particular location.

![Experimental-Numerical Comparison on 3D Bump Phase-Shift Underneath Shock Location](image)

Figure 7.24: Numerical and experimental phase shift underneath shock location for different spanwise positions in 3D nozzle

**7.3.2.3 Analysis of unsteady pressure distribution in channel**

**Evolution of unsteady pressure amplification**
7.3. THREE-DIMENSIONAL NOZZLE

Figures N.47 (left hand side) present the pressure amplification for different spanwise locations within the 3D nozzle channel. At all spanwise locations, a strong amplification zone (up to A=5 and higher) can be clearly seen at the throat of the 3D bump and simply corresponds to the presence or appearance of the shock wave during the unsteady cycle. A lower pressure amplification region (up to A=2 only) can also be observed outside this shock motion area and reaches approximately two third of the channel height. This observation, very similar to the results obtained with the 2D RANS calculation at 500Hz (Figure N.29(c)), is a direct effect of the acoustic blockage theory as the mean Mach number is about 0.8 in this area as observed in Figures M.32(b). Besides, as the 3D shock oscillates, the extent of this high flow velocity region increases during part of the unsteady cycle as observed in Figure M.34(b) in Appendix M. Finally, outside the contraction part of the 3D nozzle, the unsteady pressure amplitude is about 1, which corresponds to the propagation without amplification of the downstream perturbations. Additionally, the pressure perturbations propagate upstream of the shock location with a slightly damped amplitude (compared to the imposed amplitude at the outlet), which might be the result of numerical dissipation through the high flow velocity region.

Evolution of unsteady pressure phase-angle

Figures N.47 (right hand side) present the phase-angle distribution for different spanwise locations within the 3D nozzle channel. From a global perspective, the phase-angle distribution does not seem to change much throughout the spanwise direction in the channel, which is in agreement with the results observed previously on the pressure amplification distribution. This observation directly originate the fact that the spanwise distribution of the flow velocity is quite uniform, which is surprising over a 3D geometry but probably results from the weak shock configuration. The vertical phase-angle distribution is very similar to the results obtained with 2D RANS calculations (Figure N.29(d)) and exhibits, above the bump surface, a very slight variation to the regularly decreasing value observed on the upper wall.

Finally, it should be mentioned that the unsteady pressure distribution behaviour within the 3D bump is similarly to the analysis performed on the 2D RANS calculations of the 2D nozzle (Figure N.29) and corresponds, as described during the analysis of the 2D simulation, to a characteristic behaviour obtained with a weak shock configuration. It would have been interesting to verify if a stronger shock configuration would have produced the same amplification pattern over the 3D geometry as the one observed in the 2D nozzle and presented in Figures N.31-N.33.

7.3.2.4 Analysis of the unsteady shock motion

Global presentation

A time strip of the sonic pocket is presented in Figure 7.25 in order to obtain a direct and visual overview over the 3D unsteady shock motion. As observed, the extreme shock positions and shapes correspond rather closely to the steady state configurations presented in Figures 6.47(a) and 6.47(c) in chapter 6. The unsteady behaviour in between those two quasi-steady positions is however not linear at all. Indeed, as seen in
Figures 7.25(h)-7.25(j), a second sonic pocket appears during the unsteady cycle, while the shock starts moving downstream from its most upstream position. This second sonic pocket simply originates from the interaction between "relatively" high amplitude incoming perturbations in a region where the flow velocity is already close to sonic conditions. Nevertheless, this phenomenon creates a strong non-linear behaviour of the shock motion, which induces the high non linearity observed on the unsteady pressure distribution on the bump surface.
7.3. THREE-DIMENSIONAL NOZZLE

Evolution of unsteady pressure amplification

Figure N.48 (left hand side) present the amplitude of the unsteady shock motion at different spanwise locations and for the three first harmonic contributions. On the low curvature side of the 3D bump, for y=20mm and 35mm, the shock disappear during the unsteady cycle and the presented values does not have any physical meaning. From the mid-channel (y=50mm) to the high curvature side (y=80mm) the shock motion amplitude seems rather constant through the height of the channel and decreases with the spanwise position. This result is in agreement with previous observation regarding the amplifying effect of low mean flow gradient region on the shock motion. Besides, due to the non-linear shock motion behaviour of the shock motion non-linear behaviour observed in Figure 7.25, the amplitude of the first and second harmonics are quite important and represents respectively two and one third of the fundamental contribution. Similarly to the fundamental part of the shock motion, the distribution is fairly constant throughout the height of the channel.

Evolution of unsteady pressure phase-angle
Figure N.48 (right hand side) presents, for different spanwise locations, the phase-angle distribution of the unsteady shock motion as a function of the vertical direction. Although the amplitude distribution on the low curvature side of the 3D bump was erroneous, the phase-angle could be estimated from only part of the signal since a Fourier decomposition was performed for each harmonic contribution and thus at a specific frequency. As a result, it is interest to note that the phase-angle of the fundamental of the unsteady shock motion is nearly constant in the spanwise direction. Similarly, the first and second harmonic contributions are also constant in the spanwise direction and both feature a $180^\circ$ phase shift with the fundamental, which makes the amplitude of the different harmonic contributions partly cancelling each other. As mentioned, the vertical phase-angle distribution is fairly constant for each spanwise location. As a result, the shock wave oscillates in a rather solid body motion, which is in agreement with previous results observed both on experimental visualizations and 2D RANS calculations with a weak shock configuration.

### 7.3.2.5 Concluding remarks

This subsection was dedicated to the description of the 3D unsteady RANS calculation of upstream propagating pressure disturbances in a 3D transonic flow with a weak shock configuration.

The unsteady pressure distribution (amplitude and phase-angle) was investigated both on the 3D bump surface and within the channel as a function of the spanwise location. A similar behaviour (and fairly good agreement) with experimental results was found. In particular, an amplification region was observed downstream of the shock location to vary in streamwise extent and intensity level with the spanwise direction. The comparison with experimental results outlined a slight difference in the streamwise extent of this amplification zone and could be explained by a difference in the mean flow velocity downstream of the shock. The phase-angle distribution was found characteristic of single travelling plane wave, in agreement again with experimental measurements on the bump surface. Especially interesting, it was observed an increasing phase shift underneath the shock location with the spanwise direction.

Furthermore, the unsteady shock motion was analyzed for different spanwise location as a function of the vertical direction. It was found that a second sonic pocket appears while the shock starts to move downstream from its most upstream position due to large amplitude incoming pressure perturbations. This behaviour resulted in a raise in the non-linear contribution of both the shock motion amplitude and the unsteady pressure distribution on the bump. The analysis of the shock motion phase-angle distribution throughout the channel width indicated that the shock oscillates as a solid body motion with no phase at all.

As a final remark, it should be noted that the overall behaviour of the interaction between the upstream propagating pressure perturbations and the weak shock was found similar in many points to the analysis performed on the 2D nozzle over a weak shock configuration.
7.4 Conclusion

This chapter was dedicated to the investigation of unsteady interaction between upstream propagating pressure disturbances with a transonic flow in 2D and 3D nozzle geometries. Experimental results obtained from high frequency pressure measurements and high speed Schlieren visualizations were analyzed and compared with each others through a parametric study. Simultaneously, unsteady 2D RANS calculations were performed, on one hand, at the same configurations in order to enable a realistic comparison, and on the other hand, for other operating conditions in order to enlarge the physical understanding of the complex interaction and include further mean shock configurations and higher perturbing frequencies. The analysis was systematically performed on both the real and imaginary part of the unsteady results, including when possible a parametric study regarding the amplitude and frequency of the imposed perturbations, as well as on the effect of the steady state SBLI intensity, determined by the mean shock location.

Experiments on the 2D bump showed that the unsteady pressure distribution (amplification and phase-angle) is strongly affected by both the mean shock location and the perturbation frequency. Globally, for a weak shock configuration, the pressure amplification is nearly non-existent downstream of the shock, whereas an important amplification occurs for strong SBLI with almost choked nozzle flow and large shock induced separation. Furthermore, the influence of the perturbation frequency was found closely coupled to the mean shock location and could lead either to pressure amplification or attenuation downstream of the shock, depending on the mean shock location. Similar conclusions could be made for the phase-angle distribution. The amplitude of perturbation, on the other hand, does not seem to have any influence except on the level of pressure amplification.

Similarly, Schlieren visualizations showed that the unsteady shock motion (amplitude and phase-angle distributions) is also influenced by both the perturbation frequency and the mean shock location. Globally, for non-choked nozzle flow, the shock motion amplitude is magnified in low mean flow gradient regions but systematically decreases with the perturbation frequency. Furthermore, an evolution of the shock motion pattern from rigid body motion to a "wavy" oscillations as the shock grows in strength i.e. for stronger SBLI. For choked nozzle flow configuration on the other hand, the shock motion amplitude is rather attenuated due to the presence of a lambda shock system on both extremities of the discontinuity, and, hence, the absence of low mean flow gradient regions any longer. Moreover for such configuration, the shock motion pattern is back to a rigid body motion. In any case, the amplitude of motion systematically decreases with the perturbation frequency.

Two-dimensional simulations showed the very same tendencies regarding both the unsteady pressure behaviour (amplification and phase-angle distribution) and the shock motion. In particular, further comparisons with Euler computations outlined the importance role of viscous (or/and turbulent) effects regarding pressure amplification (or attenuation) in the case of shock induced separation. The strong BL thickening originat-
ing from the superposition of two adverse pressure gradient effects (curvatures and shock discontinuity) was believed to be closely related to the observed pressure amplification and phase-angle shift downstream of the shock location.

Furthermore, a phase shift was systematically observed on the surface underneath the shock location for both experimental measurements and numerical simulations to a fairly good degree of agreement. This phase shift was found to increase almost linearly with both the strength of the shock and the perturbation frequency. This phase shift basically gives a raise to the phase-angle distribution over half of the bump surface and therefore significantly contributes to the unsteady aerodynamic force acting on the surface.

The analysis of the correlation between the shock motion and the unsteady pressure perturbations immediately downstream of the shock showed a linear increase of the phase-lag (increasing delay of the shock response time) with the perturbation frequency for weak shock configurations. For strong configurations however, the results showed opposite trends corresponding to an advance of the shock motion compared to the incoming pressure perturbations.

Finally the analysis of the separated region motion downstream of the shock outlined the fact that the separation point oscillates linearly and in-phase with the shock wave, whereas the reattachment point motion features a strong non-linear behaviour and a clear phase-lag with the shock motion. Interestingly, both the amplification factor (related to the shock motion amplitude) and phase-lag of the reattachment point motion were found to increase with the perturbation frequency.

Unsteady results on the 3D bump showed, both for the experimental pressure measurements and 3D RANS calculation, an amplification region downstream of the shock location with a varying streamwise extent and intensity level with the spanwise direction. The comparison with experimental results outlined a slight difference in the streamwise extent of this amplification zone and could be explained by a difference in the mean flow velocity downstream of the shock. The analysis of the phase-angle distribution outlined the typical characteristics of single travelling plane wave. Furthermore, it was observed an increasing phase shift underneath the shock location with the spanwise direction.

The numerical analysis of the unsteady shock motion in the 3D nozzle revealed the appearance of a second sonic pocket while the shock starts to move downstream from its most upstream position. This non-linear behaviour was assumed to be related to large amplitude incoming pressure perturbations in a high flow velocity region. This resulted in high non-linear contributions of both the shock motion amplitude and the unsteady pressure distribution on the bump. The analysis of the shock motion phase-angle distribution throughout the channel width for the 3D RANS calculation indicated that the shock oscillates with a rigid body motion with no phase at all.

As a final remark, the overall behaviour of the interaction between the upstream propagating pressure perturbations and the weak shock configuration in the 3D nozzle geometry was found similar in many points to the analysis performed on the 2D nozzle over a weak shock configuration.
Chapter 8

Conclusion and perspectives

The present research work was dedicated to the investigation of the interaction between an oscillating shock wave and a turbulent boundary layer in a non-uniform flow. The objective, from a general point of view, was to deepen the fundamental understanding of unsteady flow phenomena and their interaction with steady flow in axial turbomachines in order to better predict the aerodynamic force acting on a turbomachine blade or an airfoil.

First, a general presentation of unsteady flow phenomena and aeroelasticity related problems in axial turbomachines was given. Thereafter, a detailed description of turbulent boundary layer flows, with and without adverse pressure gradient influence, was presented, followed by a phenomenological description of the complex SBLI on airfoils and wind tunnel applications. A non exhaustive literature survey on transonic viscous interactions, with and without separations, was also presented, with the emphasis on the main flow characteristics within the interaction region, and the assessment of influence parameters.

In order to focus the analysis on the essential flow features and avoid possible acoustic interactions in linear cascades, the study was performed on convergent-divergent 2D and 3D nozzles. Beside, with the aim at obtaining a coherent validation of the results, the investigation was conducted both experimentally and numerically. As a result, numerical simulations were performed using a CFD tool developed at ECL (France), which solves the fully unsteady 3D compressible RANS equations, using a finite volume formulation and a linear two-equations turbulent model. On the other hand, the experimental investigation was performed at KTH (Sweden). First, the test section of a supersonic wind tunnel was entirely redesigned in a modular way in order to feature easy accesses for instrumentation, flow visualizations, and test object exchange. Thereafter, the 2D and 3D test objects (also called "bumps") were designed and instrumented in order to facilitate the use of several measuring techniques and extensively investigate the unsteady SBLI. As a result, steady state and unsteady pressure measurements, thermal anemometry, laser-two-focus velocimetry, steady and high speed conventional Schlieren, as well as oil-paint visualizations were performed to fully describe the flow field within the 2D and 3D nozzles. Furthermore, the presentation of the experimental model systematically introduced the operating system, acquisition method and data reduction procedure, as
well as estimations of the respective measurement accuracy and errors.

An extensive steady state investigation was performed on both the 2D and 3D nozzle geometries for several operating conditions from subsonic to transonic choked nozzle flow. Experimental results obtained from the different measuring techniques cited above were analyzed and compared with each others. Simultaneously, 2D and 3D RANS calculations were performed on the same configurations and compared with experimental results. The evolution of different flow structures was described as a function of the shock position. Results on the 2D bump showed that the flow remains quasi two-dimensional over 80% of the channel width as long as the shock is weak. However when the shock reaches a certain streamwise position, the superposition of the strong adverse pressure gradient and the curvatures gives a rise the boundary layer thickness, which initiates the formation of a lambda shock system and simultaneously triggers a separation of the flow. The interaction between the shock and the side wall boundary layers is then strong enough to create large vortices which contribute to the thickening of the corner separation and narrow the two-dimensionality of the flow to a 20% large strip in the middle of the channel. Despite a mismatch of inlet boundary conditions, comparison with numerical simulations showed a fairly good agreement for weak shock configurations but also a clear under-estimation of the losses for strong SBLI. The comparison between 2D and 3D RANS calculations revealed that the thickening of the side wall boundary layers was essential in the shock position determination but numerically still largely under-predicted. Results on the 3D bump showed similar trends with even more pronounced consequences as the three-dimensional geometry induces higher mean flow gradients and larger separations. Although the flow structures were qualitatively very similar for weak shock configurations, as soon as the shock gets stronger, numerical and experimental results differ in such an order of magnitude that no direct comparison is possible any more. The under-estimation of the boundary layer thickening and extent of separated region is, again, believed to be the main origin of such difference.

The main contribution of this research work to the fundamental understanding of the unsteady SBLI basically consisted in the analysis of the interaction between upstream propagating pressure disturbances with an oscillating shock in 2D and 3D nozzle geometries. Similarly, both experimental and numerical investigation were performed. As a result, unsteady pressure measurements and high speed Schlieren visualizations were analyzed and compared with each others through a parametric study. Simultaneously, unsteady 2D RANS calculations were performed, on one hand, at the same configurations in order to enable a realistic comparison, and on the other hand, at other operating conditions in order to enlarge the physical understanding of the complex interaction and include further mean shock configuration and higher frequencies of the pressure perturbations. The analysis was systematically performed on both the real and imaginary part of the unsteady results, including when possible a parametric study regarding the amplitude and frequency of the imposed perturbations, as well as the influence of the mean shock location i.e. the steady state SBLI intensity.

Experiments on the 2D bump showed that the unsteady pressure distribution (amplification and phase-angle) is strongly affected by both the mean shock location and the perturbation frequency. Globally, for a weak shock configuration, the pressure amplifica-
tion is nearly non-existent downstream of the shock, whereas an important amplification occurs for strong SBLI with almost choked nozzle flow and large shock induced separation. Furthermore, the influence of the perturbation frequency was found closely coupled to the mean shock location and could lead either to pressure amplification or attenuation downstream of the shock, depending on the structure of the SBLI. Similar conclusions could be made for the phase-angle distribution. The amplitude of perturbation, on the other hand, does not seem to have any influence except on the level of pressure amplification.

Similarly, Schlieren visualizations showed that the unsteady shock motion (amplitude and phase-angle distributions) is also influenced by both the perturbation frequency and the mean shock location. Globally, for non-choked nozzle flow, the shock motion amplitude is magnified in low mean flow gradient regions but systematically decreases with the perturbation frequency. Furthermore, an evolution of the shock motion pattern from rigid body motion to a “wavy” oscillations as the shock grows in strength i.e. for stronger SBLI. For choked nozzle flow configuration on the other hand, the shock motion amplitude is rather attenuated due to the presence of a lambda shock system on both extremities of the discontinuity, and hence no low mean flow gradient regions any longer. Moreover for such configuration, the shock motion pattern is back to a rigid body motion. In any case, the amplitude of motion systematically decreases with the perturbation frequency.

Two-dimensional simulations showed the very same tendencies regarding both the unsteady pressure behaviour (amplification and phase-angle distribution) and the shock motion. In particular, further comparisons with Euler computations outlined the importance role of viscous (or/and turbulent) effects regarding pressure amplification (or attenuation) in the case of shock induced separation. The strong BL thickening originating from the superposition of two adverse pressure gradient effects (curvatures and shock discontinuity) was believed to be closely related to the observed pressure amplification and phase-angle shift downstream of the shock location.

Furthermore, a phase shift was systematically observed on the surface underneath the shock location for both experimental measurements and numerical simulations to a fairly good degree of agreement. This phase shift was found to increase almost linearly with both the strength of the shock and the perturbation frequency. This phase shift basically gives a raise to the phase-angle distribution over half of the bump surface and therefore significantly contributes to the unsteady aerodynamic force acting on the surface.

The analysis of the correlation between the shock motion and the unsteady pressure perturbations immediately downstream of the shock showed a linear increase of the phase-lag (increasing delay of the shock response time) with the perturbation frequency for weak shock configurations. For strong configurations however, the results showed opposite trends corresponding to an advance of the shock motion compared to the incoming pressure perturbations.

The numerical analysis of the separated region motion downstream of the shock outlined the fact that the separation point oscillates linearly and in-phase with the shock wave, whereas the reattachment point motion features a strong non-linear behavior and a clear phase-lag with the shock motion. Interestingly, both the amplification factor (re-
related to the shock motion amplitude) and phase-lag of the reattachment point motion were found to increase with the perturbation frequency. However, no clear relation between the unsteady separation and the observed pressure amplification (or attenuation) on the bump could be established. Besides, further RANS calculations would be required in order to draw a clear tendency regarding the phase lag between the reattachment point and the shock motion, and determine if a phase "locking" phenomenon occur at high perturbation frequency. Finally, it is would also be really interesting, in the future work, to experimentally confirm the above observations about the separated region motion.

Unsteady results on the 3D bump showed, both for the experimental pressure measurements and 3D RANS calculation, an amplification region downstream of the shock location with a varying streamwise extent and intensity level with the spanwise direction. The comparison with experimental results outlined a slight difference in the streamwise extent of this amplification zone which could be explained by a difference in the mean flow velocity field downstream of the shock. The analysis of the phase-angle distribution outlined the typical characteristics of single travelling plane wave. Furthermore, an increasing phase shift was observed underneath the shock location with the spanwise direction. This observation is in agreement with previous results on the 2D bump stating that the phase shift linearly increases with the shock strength.

The numerical analysis of the unsteady shock motion in the 3D nozzle revealed the appearance of a second sonic pocket while the shock starts to move downstream from its most upstream position. This non-linear behaviour was assumed to be related to large amplitude incoming pressure perturbations in a high flow velocity region. This resulted in high non-linear contributions of the shock motion amplitude but did not seem to have any influence on the unsteady pressure distribution on the bump. The analysis of the shock motion phase-angle distribution throughout the channel width for the 3D RANS calculation indicated that the shock oscillates with a solid body motion with no phase at all.

As a general remark, the overall behaviour of the interaction between the upstream propagating pressure perturbations and the weak shock configuration in the 3D nozzle geometry was found similar in many points to the analysis performed on the 2D nozzle over a weak shock configuration and it would therefore be extremely interesting, in future perspectives, to also compare the unsteady pressure distribution for a strong shock configuration, like the one obtained for a back pressure value of 112kPa for instance. Indeed, back pressure fluctuations around this OP would certainly trigger an important separation with a progressive evolution in the spanwise direction and would be ideal to confirm or deny whether the BL thickening is a major parameter in the pressure amplification phenomenon in unsteady transonic flows.

Finally, it should be mentioned that due to physical constrains the spatial resolution of the unsteady pressure measurements could not be better than 5mm between each pressure taps. This coarse resolution therefore limits the analysis especially in regions underneath or downstream of the shock wave. The solution to such inconvenient would be, in future experimental investigations, to use a newly developed pressure sensitive paint with response time down to a few nano-seconds. Although this technique is still at the moment
under development and only feature single point measurement, it could soon develop and offer the possibility to map the entire surface in a few seconds, allowing real time processing and a much more interacting research possibilities.
Appendix A

Numerical Model

A.1 Introduction

The aim of this present Appendix is to give the reader a detailed presentation of the numerical model used to simulate the shock boundary layer interaction in two and three-dimensional nozzles. The numerical code, so called PROUST, has been developed at Ecole Centrale de Lyon in order to simulate steady and unsteady, viscous and inviscid flows in turbomachines.

In the first part of this Appendix, the instantaneous Navier-Stokes equations are written in a conservative form, statistically averaged to account for turbulence effect, and finally made dimensionless to facilitate the numerical treatment.

The second part of the Appendix presents the different numerical methods used to discretize the partial differential equations system obtained previously. The finite volume numerical method will first be introduced, followed by the time and space discretization methods applied to the different terms of the equations. Finally, a description of the numerical treatment of the boundaries will be given.

Although no major development of the numerical methods has been performed in this thesis, the author wishes to present a thorough description of the numerical code. It is indeed important to understand the assumptions and limitations underlying any numerical computation. This is especially true when complex phenomena and interactions, which prediction directly depends on the numerical treatment, have to be simulated. The reader is also welcome to refer to further publications (Aubert, 1993; Smati, 1997) in order to get more detailed information on the numerical methods.

A.2 Governing equations

A.2.1 Instantaneous Navier-Stokes equations in conservative form

The motion of a compressible viscous Newtonian fluid is governed by the Navier-Stokes equations, which originate the fundamental principle of mechanics and thermodynamics.
Those equations are determined from the conservation laws for mass, momentum, and energy, and the thermodynamic relations for a perfect gas. The thermo-aero-dynamic flow field is then characterized by the so called \textit{conservative variables}, that is to say, the density $\rho$, the momentum $\rho \vec{V}$, and the total energy $\rho E$.

Assuming no external force, the Navier-Stokes equations can be written in differential conservative form for an Eulerian description:

\textbf{Mass conservation}

$$\frac{\partial \rho}{\partial t} + \nabla \cdot [\rho \vec{V}] = 0 \quad (A.1)$$

\textbf{Momentum conservation}

$$\frac{\partial \rho \vec{V}}{\partial t} + \nabla \cdot [\rho \vec{V} \otimes \vec{V} + p \vec{I} - \bar{\tau}] = 0 \quad (A.2)$$

\textbf{Energy conservation}

$$\frac{\partial \rho E}{\partial t} + \nabla \cdot \left[ (\rho E + p) \vec{V} + \Phi - \bar{\tau} \cdot \vec{V} \right] = 0 \quad (A.3)$$

The static pressure is determined by the equation of state for a thermally perfect gas:

$$p = \rho R T \quad (A.4)$$

Considering the equation relating the internal energy and the temperature for a calorically perfect gas, $e = C_v T = \left(E - \frac{1}{2} \vec{V}^2\right)$, the static pressure can be rewritten as follow:

$$p = \rho (\gamma - 1) \left[ E - \frac{1}{2} \vec{V}^2 \right] \quad (A.5)$$

The viscous shear stress tensor $\bar{\tau}$ for an isotropic Newtonian fluid can be expressed as follow:

$$\bar{\tau} = \lambda \nabla \vec{V} \otimes \vec{I} + \mu (\nabla \vec{V} + \nabla \vec{V}^t) \quad (A.6)$$

Although this is the most general form for a Newtonian viscous fluid, we will consider the range of fluid behaviour within local thermodynamic equilibrium for which the Stokes relation, $3\lambda + 2\mu = 0$, is valid.

The heat conduction flux determined by the Fourier law, can be expressed as function of the internal energy gradient:

$$\Phi = -\kappa \nabla T = -\frac{\kappa}{C_v} \nabla e = -\frac{\gamma \mu}{P_r} \nabla e \quad (A.7)$$
where \( P_r = \frac{\mu C_p \kappa}{\nu} \) is the Prandtl number and \( e \) the internal energy.

The dynamic viscosity of the fluid, \( \mu \), is only function of the temperature, and can be expressed by Sutherland’s law (Schlichting, 1979):

\[
\frac{\mu}{\mu_{\text{ref}}} = \left( \frac{T}{T_{\text{ref}}} \right)^{\frac{3}{2}} \frac{T_{\text{ref}} + S_1}{T + S_1}
\]

with \( S_1 = 110K \), \( T_{\text{ref}} = 293K \), \( \mu_{\text{ref}} = 1.81 \times 10^{-5} \text{kg.m}^{-1}.\text{s}^{-1} \) for air.

In practice, a constant dynamic viscosity is used when the Mach number is below 1.5.

\[\text{A.2.2 Reynolds-Averaged Navier-Stokes equations}\]

Most flows in engineering applications are turbulent. This property is inherent to the flow and has to be taken into account in any numerical modelisation. Turbulence is chaotic and originates instabilities in laminar flows. From a mathematical point of view, turbulence is a direct consequence of the non linearity of the Navier-Stokes equations. Physically, turbulent flows are characterized by widely fluctuating quantities and contain swirling structures (eddies) with characteristic length, velocity and time scales which are spread over very wide ranges with orders of magnitude larger than in laminar flows. A direct computation of the turbulent flow without approximations of the equations of motion would require a number of grid points that increases prohibitively fast with increasing Reynolds number and result in a thousandfold increase of the numerical resources required.

One other characteristic is the irregularity, or randomness, of all turbulent flows, which makes a deterministic approach impossible. One relies on statistical methods and tries to model the turbulence instead. The idea behind a statistical approach is to decompose the instantaneous flow field into a mean field and a fluctuating part. To account for compressibility effects and get ride of the density fluctuations, a Favre decomposition is used for the density weighted variables \((\bar{V}, E, e)\), whereas a Reynolds decomposition is used for \((p, \rho)\).

- **Reynolds average decomposition:**

\[
A(\bar{x}, t) = \bar{A}(\bar{x}, t) + A'(\bar{x}, t) \quad \text{with} \quad \frac{\bar{A}(\bar{x}, t)}{\tau'} = \frac{1}{\tau} \int_{\bar{x} - \tau}^{\bar{x} + \tau} A(\bar{x}, t + n)dn
\]

where \( \tau' \) represents the characteristic time of turbulence.

\( \tau \) represents the unsteady phenomenon characteristic time.

- **Favre averaging:**

\[
A(\bar{x}, t) = \bar{A}(\bar{x}, t) + \bar{A}'(\bar{x}, t) \quad \text{with} \quad \bar{A}(\bar{x}, t) = \frac{\rho \bar{A}(\bar{x}, t)}{\bar{\rho}(\bar{x}, t)}
\]

This decomposition is then introduced into the previous set of equations (eq. A.1 to A.3), which are thereafter ensemble averaged and finally describe the mean turbulent flow.
APPENDIX A. NUMERICAL MODEL

field evolution.

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot \left[ \rho \vec{\varepsilon} \right] &= 0 \\
\frac{\partial \rho \vec{\varepsilon}}{\partial t} + \vec{\nabla} \cdot \left[ \rho \vec{\varepsilon} \otimes \vec{\varepsilon} + p \vec{I} - \frac{\partial \rho \vec{\varepsilon}}{\partial t} - \rho \vec{\varepsilon} \otimes \vec{\varepsilon} \right] &= 0 \\
\frac{\partial \rho \vec{E}}{\partial t} + \vec{\nabla} \cdot \left[ (\rho \vec{E} + p) \vec{\varepsilon} - \{ \frac{\partial \rho \vec{E}}{\partial t} + \frac{\partial \rho \vec{E}}{\partial t} \cdot \vec{\varepsilon} - (\rho \vec{E} + p) \vec{\varepsilon} \} \right] &= 0 \\
\end{align*}
\]

(A.9)

with:

\[
\begin{align*}
p &= \rho (\gamma - 1) \\
\overline{\tau} &= \lambda \vec{\nabla} \cdot \vec{\varepsilon} + \mu \left( \vec{\nabla} \vec{\varepsilon} + \vec{\nabla} \vec{\varepsilon} \right) \\
\end{align*}
\]

The sign \( \otimes \) characterizes the product of two tensors of any order. Applied to two tensors of order one and two respectively, it becomes:

\[
\vec{V} \otimes \vec{\tau} = V_{i,a} a_{jk}
\]

The above equations have been written assuming that \( \lambda \) and \( \mu \) are proportional to the density \( \rho \), and to the kinematic viscosity \( \nu \), which is also supposed to be non-fluctuating:

\[
\overline{\mu} = \overline{p} \overline{V} = \overline{p} \overline{\nu}
\]

The error due to this approximation is of a lower order of magnitude than the one introduced by adding a turbulent viscosity calculated with a two equations turbulent model.

The system formed by the set of equations A.9 is not closed, and the double correlation terms have revealed new unknowns. The evolution equations of those variables would also reveal higher order correlation terms, and so on, ad infinitum. The closure of the above system thus requires the modelisation of those correlation terms, which is done by a turbulent model.

A.2.3 Turbulence closure

A.2.3.1 Introduction

As seen in the previous section, the average of the instantaneous Navier-Stokes equations has revealed different double correlation terms which have to be modelled in order to close the system from a mathematical point of view.

The first attempt in this task was made by Boussinesq (Boussinesq, 1897) who modelled turbulent flow simply by adding a so called eddy viscosity to the molecular viscosity. The idea behind the eddy viscosity is that it should take into account the enhanced momentum transport of the turbulent flow in the same way as molecular viscosity does for
A.2. GOVERNING EQUATIONS

laminar flow. Boussinesq thus made the hypothesis that the Reynolds stress tensor could be linearly related to the mean strain-rate tensor by a turbulent viscosity:

$$-\rho \overline{V'} \otimes \overline{V'} = \tau_t = \lambda_t \overline{\nabla \cdot \overline{V}} I + \mu_t (\overline{\nabla \overline{V}} + \overline{\nabla \overline{V}}^T) - \frac{1}{3} \rho \overline{V^n} \overline{V^n} \rightleftharpoons I \quad (A.10)$$

where $\lambda_t$ and $\mu_t$ are linked by Stokes hypothesis: $3\lambda_t + 2\mu_t = 0$.

This relation assumes that the main axis of the Reynolds stress tensor, $\tau_{ij}$, collapses together with the main axis of the main strain rate tensor, $S_{ij}$, at any location in the flow field. An important feature of turbulence is then neglected, its memory, especially in separated zones. Despite the non validity of this relation, it is still widely used nowadays because of its simplicity.

A simple dimensional analysis would show that the turbulent viscosity $\mu_t$ is a function of a velocity scale $u$ times a length scale $l$, and can therefore be obtained by the determination of two characteristic scales. Those two turbulent scales can be deduced from the resolution of their respective transport equation.

It is natural to relate the characteristic velocity scale to the velocity fluctuations, that is to say to the turbulent kinetic energy:

$$u \approx \sqrt{k} = \sqrt{\frac{1}{2} \overline{V^n}^2} \quad (A.11)$$

The choice for the second characteristic turbulent scale is however more arbitrary. It can be a length scale $\ell$, or any combination of type $\phi = u^m l^n$. The two equations models therefore consist of solving the transport equations for $k$ and $\phi$.

Among the different turbulent models available in the numerical model PROUST (Smati, 1997), the $k-\omega$ model developed by Wilcox (Wilcox and Traci, 1988; Wilcox, 1976) seemed to be the best choice. This model was first presented by Kolmogorov (Kolmogorov, 1942) and was a subject to further developments (Wilcox, 1991, 1992, 1993a,b) during the last decade. The second characteristic scale used is $\omega$, the specific dissipation rate, which physically represents the inverse of the time decay of large turbulent structures.

This model presents several advantages compared to other two equations models:

- Easier numerical resolution in near wall region since it requires less nodes.
- No damping function is needed in near wall region (Wilcox, 1993a).
- The same formulation for low or high Reynolds number regions.
- **Better prediction of separated flow** in general (Menter, 1992; Kandula and Wilcox, 1995).
- An appropriate choice of the second turbulent characteristic scale (Huang et al., 1994).

The disadvantages of this model are:

- A delicate numerical treatment of the theoretical infinite value of $\omega$ on the wall.
- A high sensitivity of the free stream values.
APPENDIX A. NUMERICAL MODEL

A.2.3.2 Simplification of the RANS

Turbulent kinetic energy

According to the definition of the turbulent kinetic energy $k = \frac{1}{2} \bar{V}_n^2$, the equation of state for a thermally perfect gas A.5 then becomes:

$$p = \bar{p} (\gamma - 1) \left[ \bar{E} - \frac{1}{2} \bar{V}_n^2 - \bar{k} \right]$$  \hspace{1cm} (A.12)

Reynolds stress tensor

The Reynolds stress tensor, $\rho \bar{V}_n \otimes \bar{V}_n$, determined using Boussinesq’s approximation becomes:

$$- \rho \bar{V}_n \otimes \bar{V}_n = \frac{\tau}{\tau_t} = \lambda_t \nabla \bar{V}_n \cdot \bar{I} + \mu_t (\nabla \bar{V}_n + \nabla \bar{V}_n^T) - \frac{2}{3} \rho \bar{k} \bar{I}$$  \hspace{1cm} (A.13)

with $3 \lambda_t + 2 \mu_t = 0$ (Stokes hypothesis).

Energy equation

The non-linear term, $(\rho \bar{E} + p) \bar{V}_n^T$, in the energy conservation equation A.3 can then be written:

$$\bar{V}_n^T \bar{M} = \rho \bar{H}_n \bar{V}_n + \rho \sum_{j=1}^{3} \bar{u}_j \bar{u}_j^T \bar{u}_n + \rho \sum_{j=1}^{3} \frac{1}{2} \bar{u}_j^T \bar{u}_j^T \bar{u}_n$$  \hspace{1cm} (A.14)

hence:

$$\bar{V}_n^T \bar{M} = \rho \bar{H}_n \bar{V}_n + \rho \sum_{j=1}^{3} \bar{u}_j^T \bar{u}_j^T \bar{u}_n + \rho \bar{k} \bar{V}_n$$  \hspace{1cm} (A.15)

The temperature-velocity correlations can be expressed as function of the mean temperature gradient using a turbulent Prandtl number:

$$\bar{p} T \bar{V}_n = - \frac{\mu_t}{P_{t_e}} \bar{\nabla} T \hspace{1cm} \text{with} \hspace{1cm} P_{t_e} = \frac{\mu_t C_p}{\kappa}$$  \hspace{1cm} (A.16)

A similar formulation can be used for the triple correlation terms $k \bar{V}_n$:

$$\bar{p} k \bar{V}_n = - \frac{\mu_t}{\sigma_k} \bar{\nabla} k$$  \hspace{1cm} (A.17)

where $\sigma_k$ is a constant of the $k - \omega$ model.

The development of the term $\bar{V}_n \bar{V}_n$ reveals, on one hand, the correlation term $\bar{V}_n \bar{V}_n$ which expresses the viscous strain mean work due to the displacement created by the
A.2. GOVERNING EQUATIONS

velocity fluctuations. In practice, this dissipation effect is very small and thus neglected in a first order turbulent model. On the other hand, the term $\overrightarrow{\tau_l} \overrightarrow{V}$ which appears in the previous development can be modelled by $\overrightarrow{\mu} \overrightarrow{\nabla} \widetilde{k}$. Thus:

$$\overrightarrow{\tau_l} \overrightarrow{V} = \overrightarrow{\tau_l} \overrightarrow{V} + \overrightarrow{\mu} \overrightarrow{\nabla} \widetilde{k}$$  \hspace{1cm} (A.18)

RANS equations

Finally, after the above simplifications, the Reynolds Averaged Navier-Stokes equations can be written as follow:

$$\begin{cases}
\frac{\partial \rho}{\partial t} + \overrightarrow{\nabla} \cdot [\rho \overrightarrow{V}] = 0 \\
\frac{\partial \rho \overrightarrow{V}}{\partial t} + \overrightarrow{\nabla} \cdot [\rho \overrightarrow{V} \otimes \overrightarrow{V} + p \overrightarrow{I} - \overrightarrow{\tau_{t+t}}] = 0 \\
\frac{\partial \rho \widetilde{E}}{\partial t} + \overrightarrow{\nabla} \cdot [\rho \widetilde{E} + p \overrightarrow{V} - \left\{ \gamma \left( \overrightarrow{\tau_k} + \frac{\mu_t}{\alpha_k} \right) \overrightarrow{\nabla} + \overrightarrow{\tau_{t+t}} \overrightarrow{V} + \left( \overrightarrow{\mu} + \frac{\mu_t}{\alpha_k} \right) \overrightarrow{\nabla} \widetilde{k} \right\}] = 0
\end{cases}$$  \hspace{1cm} (A.19)

where $\overrightarrow{\tau_{t+t}}$ is the sum of the viscous strain tensor $\overrightarrow{\tau_l}$ and the Reynolds stress tensor $\overrightarrow{\tau_t}$.

A.2.3.3 Turbulent variables transport equations

In order to complete the closure of the differential partial equations system, the kinetic turbulent energy $k$, and the turbulent viscosity $\mu_t$ must be determined by solving their respective transport equation.

Transport equation on $k$

The transport equation for the turbulent kinetic energy $k$ is obtained by multiplying the instantaneous momentum equation A.2 by the velocity fluctuations $\overrightarrow{u_i} \overrightarrow{u_j}$, making a Reynolds average, and taking the trace of the resulting equation.
It comes:

\[
\frac{\partial \rho . k}{\partial t} + \frac{\partial \rho . k \bar{V}_i}{\partial x_i} = -\frac{\partial \bar{p}.u_i u_j}{\partial x_j} \tag{A.20}
\]

**Production term**

\[
- \frac{\partial \bar{p}.u_i}{\partial x_j} \quad \text{Turbulent diffusion terms}
\]

\[
+ \frac{\partial \tau_{ij} u_i}{\partial x_j} \quad \text{Viscous diffusion term}
\]

\[
- \frac{\partial \bar{u}_i}{\partial x_j} \quad \text{Dissipation term}
\]

\[
- u_i \frac{\partial \bar{p}}{\partial x_j} + p' \frac{\partial u_i}{\partial x_j} \quad \text{Compressibility terms}
\]

Equation A.20 is exact and contains higher correlation terms which are needed to be modelled:

- The production term III can be modelled using Boussinesq’s approximation:

\[
III = \tau_{ij} \frac{\partial \bar{U}_j}{\partial x_j} = \mu \left( S_{ij} - \frac{2}{3} \frac{\partial \bar{V}_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \frac{\rho k \delta_{ij}}{\bar{V}_j} \tag{A.21}
\]

- The terms IV and V can be interpreted as diffusion by the fluctuating movement of the fluid. Both are modelled together using Boussinesq’s approximation:

\[
IV + V = \frac{\partial}{\partial x_j} \left( \frac{\mu \partial k}{\sigma_k \partial x_j} \right) \tag{A.22}
\]

- The viscous diffusion term is modelled as follow:

\[
VI = \tau_{ij} u_i = \mu \frac{\partial^2 k}{\partial x_j^2} \tag{A.23}
\]

- The term VII, also noted \( \bar{\rho} \varepsilon \), is the dissipation rate of the turbulent kinetic energy \( k \) and is modelled using a dimensional analysis:

\[
VII = \tau_{ij} \frac{\partial u_i}{\partial x_j} = \bar{\rho} \varepsilon = C_k \omega k \tag{A.24}
\]
The terms VIII and IX originate compressibility effects. They represent the pressure work and pressure dilatation due to density fluctuations, and thus vanish for incompressible flow. In compressible flow however, DNS research (Zeman, 1991) has shown that their influence is very small unless the flow is supersonic or hypersonic, and both terms are therefore neglected in Wilcox’s $k - \omega$ model.

Finally, the modelisation of the Reynolds averaged transport equation for the turbulent kinetic energy $k$ is:

$$
\frac{\partial \rho k}{\partial t} + \nabla \cdot \left[ \rho k \vec{V} - \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] = \frac{\tau_t}{\tau_t} \nabla \vec{V} - C_k \rho \omega k \quad (A.25)
$$

with: $\sigma_k = 2.0$

$C_k = 0.09$

The notation $\div$ is used to specify the mixte product between two tensors of order $m$ and $n$ ($m \geq n$). The result of this product is a tensor of order $(m - n)$. Thus, the mixte product between two second order tensors is a scalar: $\vec{a} \div \vec{b} = a_{ij} b_{ij}$

**Transport equation on $\omega$**

Proceeding the same way as for equation A.20, the Reynolds averaged transport equation for the specific dissipation $\omega$ can be written:

$$
\frac{\partial \rho \omega}{\partial t} + \nabla \cdot \left[ \rho \omega \vec{V} - \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \nabla \omega \right] = C_{\omega_1} \frac{\omega}{k} \left( \frac{\tau_t}{\tau_t} \nabla \vec{V} \right) - C_{\omega_2} \rho \omega^2 \quad (A.26)
$$

with: $\sigma_\omega = 2.0$

$C_{\omega_1} = \left( C_{\omega_2} - \frac{K^2 \sqrt{C_k}}{\sigma_\omega} \right) \simeq \frac{5}{5}$

$C_{\omega_2} = \frac{1}{30}$

**A.2.4 Non dimensional formulation**

With the aim to bring all variables to an order of magnitude close to unity, the RANS equations have to be made dimensionless. For this purpose, six distinct reference scales have been defined as follow:

- One geometrical scale:
  $L_0 \left[ m \right]$, the reference length scale

- Two kinematic scales:
  $\rho_0 \left[ \frac{kg}{m^3} \right]$ and $\vec{V}_0 \left[ \frac{m}{s} \right]$, the reference density and velocity
APPENDIX A. NUMERICAL MODEL

- Two thermodynamic scales:
  \[ r_0 \left( \frac{J}{kg.K} \right) \text{ and } \mu_0 \left( \frac{kg}{m.s} \right) \text{, the reference constant of a thermally perfect gas and the reference dynamic viscosity} \]

- One thermal scale:
  \[ \kappa_0 \text{, the reference thermal conductibility} \]

Some dimensionless numbers can then be defined from those reference scales:

\[ R_e = \frac{\rho_0 L_0 V_0}{\mu_0} \quad P_r = \gamma - 1 \frac{\mu_0}{\kappa_0} \]

Let’s define the new dimensionless variables:

\[
\begin{cases} 
  \tau^* = \frac{t}{(L_0 V_0)^2} & \mu^* = \frac{\mu}{\mu_0} \quad T^* = \frac{\tilde{T}}{T_0} \\
  \rho^* = \frac{\rho}{\rho_0} \quad x_t^* = \frac{\tilde{x}}{L_0} \quad \tilde{\tau} = \frac{\rho_0 V_0}{\mu_0} \quad \tilde{\varepsilon} = \frac{\tilde{\varepsilon}}{\tilde{\eta}^*} \\
  p^* = \frac{p}{\rho_0 V_0^2} \quad u_t^* = \frac{\tilde{u}}{V_0} \quad E^* = \frac{\tilde{E}}{\tilde{\eta}^*} \quad \omega^* = \frac{\tilde{\omega}}{\tilde{\eta}^*} 
\end{cases}
\]

By injecting the dimensionless variables into the RANS equations, we obtain the dimensionless RANS equations that will be used to simulate the motion of a compressible turbulent flow:

**Mass conservation:**
\[
\frac{\partial \rho}{\partial t} + \nabla \cdot \left[ \rho \tilde{V} \right] = 0 \tag{A.28}
\]

**Momentum conservation:**
\[
\frac{\partial \rho \tilde{V}}{\partial t} + \nabla \cdot \left[ \rho \tilde{V} \otimes \tilde{V} + p \tilde{I} - \frac{1}{R_e} \tilde{\tau}_{t+t} \right] = 0 \tag{A.29}
\]

**Energy conservation:**
\[
\frac{\partial p E}{\partial t} + \nabla \cdot \left[ (\rho E + p) \tilde{V} - \frac{1}{R_e} \left\{ \gamma \left( \frac{\mu}{P_r} + \frac{\mu_t}{P_t} \right) \tilde{e} + \tilde{\tau}_{t+t} \cdot \tilde{V} + \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla \tilde{k} \right\} \right] = 0 \tag{A.30}
\]

with:
\[ p = \rho(\gamma - 1) \left( E - \frac{1}{2} \tilde{V}^2 - k \right) \]
\[ \tilde{\tau}_{t+t} = (\lambda + \lambda_t) \nabla \tilde{V} \cdot \tilde{V} + (\mu + \mu_t)(\nabla \tilde{V} + \nabla \tilde{V}^T) - \frac{2}{3} R_e \rho k \tilde{I} \]

**Transport equation on \( k \):**
\[
\frac{\partial \tilde{p} k}{\partial t} + \nabla \cdot \left[ \rho \nabla \tilde{V} - \left( \tilde{p} + \frac{\mu t}{\sigma_k} \right) \nabla \tilde{k} \right] = \frac{1}{R_e} P_k - C_k \rho \omega k \tag{A.31}
\]

**Transport equation on \( \omega \):**
\[
\frac{\partial \tilde{p} \omega}{\partial t} + \nabla \cdot \left[ \rho \omega \tilde{V} - \left( \tilde{p} + \frac{\mu t}{\sigma_\omega} \right) \nabla \tilde{\omega} \right] = C_{\omega} \omega \frac{1}{k} \frac{1}{R_e} P_k - C_{\omega \omega} \rho \omega^2 \tag{A.32}
\]

with:
\[
\begin{align*}
  P_k &= \frac{\tilde{\omega}}{k} \nabla \tilde{V} \\
  \sigma_k &= 2.0 \quad \sigma_\omega = 2.0 \quad C_k = 0.09 \quad C_{\omega} = \left( C_{\omega} - \frac{K_2 \sigma_\omega}{\sigma_\omega} \right) \approx \frac{5}{9} \\
  C_{\omega \omega} &= 0.01
\end{align*}
\]

256
A.2.5 Conclusion

This section was dedicated to the edification of a closed equations system describing the unsteady motion of a compressible turbulent flow. The fully 3D instantaneous Navier-Stokes equations are averaged to statically account for turbulence effects, and the two-equations model developed by Wilcox is used to close the mathematical system.

As a result, the unsteady motion of a 3D compressible turbulent flow is fully described by a system of seven partial differential equations with seven unknowns described as the following conserving variables: $\rho$, $\rho V_x$, $\rho V_y$, $\rho V_z$, $\rho E$, $\rho k$, and $\rho \omega$. The numerical resolution of such system will be presented in the next section.

A.3 Numerical methods

A.3.1 Introduction

The unsteady motion of a 3D compressible turbulent flow has been modelled in the previous section by a partial differential equations system defined in a bounded continuous domain. In many cases, however, the resolution of such system does not admit any analytical solution and has to be numerically resolved. To do so, both the domain and the mathematical system have to be discretised. That is to say, the partial differential equation system is being replaced by an algebraic equation system for variables calculated only at a finite number of discrete positions in time and space. The discrete solution is thus an approximation of the initial problem, as accurate as the discretization is fine and close to a continuity.

There exists many different techniques to discretize the differential operators. The choice regarding a numerical method mostly depends on the physics of the phenomenon, as well as on the complexity of the geometry, and on the development and exploitation costs. The spatial discretization of the computational model (Aubert, 1993) used within this framework is based on a finite volume formulation with vertex variable storage, developed on a structured mesh. The convective terms are evaluated using an upwind flux vector splitting scheme coupled to a MUSCL approach, whereas the viscous and turbulent terms are computed using a second order finite difference scheme. The time discretization is performed using an explicit five-steps Runge-Kutta time marching algorithm.

The following sections present more in detail the numerical treatment of the different terms of the equation system. All the developments below concern an internal node, assuming the boundaries far enough to complete the discretization. The boundary nodes will be treated in a separated section.

A.3.2 Finite volume formulation

The basic idea of a finite volume formulation is to fragment the domain into elementary contiguous volumes, on which the differential conservative equations are integrated. The
cells have to be small enough so that the flow properties slowly vary within each integrating volumes.

This approach presents several advantages. First of all, it is conservative, which makes the discretization of the differential equations more natural. Moreover, discontinuities like shock waves and contact surfaces are naturally treated as the are weak solutions of the problem. Indeed, although the flow field might be discontinuous, the fluxes through the surfaces of the integrating cells remain continuous. At last, there is no real strain concerning the size and shape of the cells as far as the mesh is fine enough to consider the flow properties constant inside each cell. In practice however, the accuracy of the solution is degraded by grid distortion and an orthogonal or near-orthogonal grid should be used.

Looking at the partial differential equations A.28 to A.32, one can note the following general form:

\[
\frac{\partial A}{\partial t} + \vec{\nabla} \cdot \vec{B} = S
\]  

(A.33)

where: \(A\) and \(S\) are scalar variables (respectively vector)  
\(\vec{B}\) is a vector variable (respectively tensor)

In a finite volume formulation with structured mesh, the above equation is integrated in a curvilinear system, on each single contiguous elementary volume. Using Green’s theorem, it comes:

\[
\frac{d}{dt} \int_{V(t)} A.dV + \oint_{S(t)} \vec{B} \cdot \vec{n}.dS = \int_{V(t)} S.dV
\]  

(A.34)

where: \(S(t)\) is the surface of the integrating cell.  
\(V(t)\) is the volume of the integrating cell.  
\(\vec{n}\) is a unit vector normal to \(S(t)\), pointing outside \(V(t)\)

The evaluation of those integrals is performed by introducing a transformation from physical \((x, y, z, t)\) space to a generic curvilinear coordinate \((\xi^1, \xi^2, \xi^3, t)\) space, which is well adapted for the description of a structured mesh (figure A.1).

![Figure A.1: Correspondence of the physical and generic coordinate domains](image)

The governing equations are then expressed in terms of the generic coordinates as independent variables and the discretization is undertaken in the generic coordinates space.
\[ \int_{V_i} \frac{\partial}{\partial t} \left( \sqrt{g} A \right) \prod_{j=1}^{3} d\xi_j + \sum_{i=1}^{3} \left\{ \int_{S_{i \pm}} \sqrt{g} \vec{B} \cdot \vec{a}^i \prod_{j=1}^{3} d\xi_j \right\} = \int_{V_i} \sqrt{g} S \prod_{j=1}^{3} d\xi_j \] (A.35)

where \( \sqrt{g} = \frac{D(x_1, x_2, x_3)}{D(\xi_1, \xi_2, \xi_3)} \) is the Jacobian of the geometric transformation

\[ \vec{a}^i = \vec{e} \xi^i \] is the unity vector of the counter-varying basis associated to \( \xi^i \)

The integrals of equation A.35 can be evaluated assuming that the flow properties vary slowly inside each integrating cell. The integrand can then be replaced by a mean value over the elementary volume, but this approximation supposes that the cells are fine enough and that their size and shape vary slightly compared to adjacent ones.

It comes the following approximations:

\[ \int_{V_i} \frac{\partial}{\partial t} \left( \sqrt{g} A \right) \prod_{j=1}^{3} d\xi_j \approx \frac{\partial}{\partial t} \left( \sqrt{g} A \right) \cdot \mathcal{V}^i \] (A.36)

\[ \int_{S_i} \sqrt{g} \vec{B} \cdot \vec{a} \prod_{j=1}^{3} d\xi_j \approx \sqrt{g} \vec{B} \cdot \vec{a} \cdot S^i \] (A.37)

\[ \int_{V_i} \sqrt{g} S \prod_{j=1}^{3} d\xi_j \approx \sqrt{g} S \cdot \mathcal{V}^i \] (A.38)

The finite volume formulation chosen in PROUST (Aubert, 1993) assumes that the conservative variables are known at the nodes of the mesh (figure A.2). Indeed, the integrating cells have been chosen so that \( \xi^{i+1} = \xi^i \pm \frac{1}{2} \), \( \mathcal{V}^i = 1 \) and \( S^i = 1 \).

As a result, equation A.35 can then be rewritten as follow:

\[ \left. \frac{\partial}{\partial t} \left( \sqrt{g} A \right) \right|_{\xi^1, \xi^2, \xi^3} + \sum_{i=1}^{3} \left\{ \pm F^i \right\} = \sqrt{g} S \left|_{\xi^1, \xi^2, \xi^3} \right. \] (A.39)
where \( F^i \) is the flux defined by: \( F^i = \sqrt{g} \vec{B} \cdot \vec{a}^i \)

More precisely, the expression of the fluxes can be developed for the conservative equations from A.28 to A.32:

**Mass conservation:**

\[
\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot \left[ \rho \vec{V} \right] = 0 \quad \text{(A.40)}
\]

\[
\text{hence: } A = \rho \\
\vec{B} = \rho \vec{V} \\
S = 0 \\
F^i = \sqrt{g} \rho V^i \quad \text{with } V^i = \xi^i u_1 + \xi^i x_2 u_2 + \xi^i x_3 u_3
\]

**Momentum conservation:**

\[
\frac{\partial \rho \vec{V}}{\partial t} + \vec{\nabla} \cdot \left[ \rho \vec{V} \otimes \vec{V} + p \ I - \frac{1}{Re} \tau_{l+t} \right] = 0 \quad \text{(A.41)}
\]

\[
\text{hence: } \vec{A} = \rho \vec{V} \\
\vec{\tilde{B}} = \rho \vec{V} \otimes \vec{V} + p \ I - \frac{1}{Re} \tau_{l+t} \\
\vec{\tilde{S}} = 0 \\
\vec{\tilde{F}}^i = \sqrt{g} \left\{ \rho \vec{V} V^i + p \vec{a}^i - \frac{1}{Re} \tau_{l+t} \cdot \vec{a} \right\}
\]

**Energy conservation:**

\[
\frac{\partial (\rho E + p)}{\partial t} + \vec{\nabla} \cdot \left[ \left( \rho E + p \right) \vec{V} - \frac{1}{Re} \left\{ \gamma \left( \frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \vec{\nabla} e + \tau_{l+t} \cdot \vec{V} + \left( \mu + \frac{\mu_t}{\sigma_k} \right) \vec{\nabla} k \right\} \right] = 0 \quad \text{(A.42)}
\]
hence: \[ A = \rho E \]
\[ \vec{B} = (\rho E + p) \vec{V} - \frac{1}{Re} \left\{ \gamma \left( \frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \vec{\nabla} c + \vec{\tau}_{\text{t}} \cdot \vec{V} + \left( \mu + \frac{\mu_t}{\sigma_k} \right) \vec{\nabla} k \right\} \]
\[ S = 0 \]
\[ F^i = \sqrt{g} \left\{ (\rho E + p) V^i - p \xi^i \right\} - \sqrt{g} \left\{ \frac{1}{Re} \left[ \gamma \left( \frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \vec{\nabla} c + \vec{\tau}_{\text{t}} \cdot \vec{V} + \left( \mu + \frac{\mu_t}{\sigma_k} \right) \vec{\nabla} k \right] \cdot \vec{a} \right\} \]

Transport equation on \( k \):
\[
\frac{\partial \rho k}{\partial t} + \vec{\nabla} \cdot \left[ \rho k \vec{V} - \frac{1}{Re} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \vec{\nabla} k \right] = \frac{1}{Re} P_k - C_k \rho \omega k \quad (A.43)
\]

hence: \[ A = \rho k \]
\[ \vec{B} = \rho k \vec{V} - \frac{1}{Re} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \vec{\nabla} k \]
\[ S = \sqrt{g} \left( \frac{1}{Re} P_k - C_k \rho \omega k \right) \]
\[ F^i = \sqrt{g} \left\{ \rho k V^i - \frac{1}{Re} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \vec{\nabla} k \cdot \vec{a} \right\} \]

Transport equation on \( \omega \):
\[
\frac{\partial \rho \omega}{\partial t} + \vec{\nabla} \cdot \left[ \rho \omega \vec{V} - \frac{1}{Re} \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \vec{\nabla} \omega \right] = C_\omega \frac{\omega}{k} \frac{1}{Re} P_k - C_{\omega_1} \rho \omega^2 \quad (A.44)
\]

hence: \[ A = \rho \omega \]
\[ \vec{B} = \rho \omega \vec{V} - \frac{1}{Re} \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \vec{\nabla} \omega \]
\[ S = \sqrt{g} \left( C_\omega \frac{\omega}{k} \frac{1}{Re} P_k - C_{\omega_1} \rho \omega^2 \right) \]
\[ F^i = \sqrt{g} \left\{ \rho \omega V^i - \frac{1}{Re} \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \vec{\nabla} \omega \cdot \vec{a} \right\} \]
Finally, the equation system to be solved can be entirely written in a semi-discrete way as follow:

\[
\frac{\partial}{\partial t} \left( \sqrt{g}q \right) \bigg|_{\xi^1, \xi^2, \xi^3} + \sum_{i=1}^{3} \left[ F^i(\xi^i + \frac{1}{2}) - F^i(\xi^i - \frac{1}{2}) \right] = \sqrt{g} S|_{\xi^1, \xi^2, \xi^3} \tag{A.45}
\]

with:

\[
q = \begin{bmatrix}
\rho \\
\rho V^3 \\
\rho E \\
\rho k \\
\rho \omega
\end{bmatrix} \tag{A.46}
\]

\[
F^i = F^i_c - \frac{1}{R_e} F^i_v \tag{A.47}
\]

\[
F^i_c = \sqrt{g} \begin{bmatrix}
\rho V^i \\
\rho V^i V^i + p \bar{a}^i \\
(\rho E + p) \bar{V}^i - p \xi^i \\
\rho k V^i \\
\rho \omega V^i
\end{bmatrix} \tag{A.48}
\]

\[
F^i_v = \sqrt{g} \begin{bmatrix}
0 \\
\vec{\tau}_{t+t} \\
\gamma \left( \frac{\bar{w}^i}{\bar{a}^i} + \frac{\bar{V}^i}{\bar{a}^i} \right) \vec{\nabla} e + \gamma \vec{\tau}_{t+t} \cdot \vec{V} + \left( \mu + \frac{\sigma_k}{\sigma_e} \right) \vec{\nabla} k \\
\left( \mu + \frac{\sigma_k}{\sigma_e} \right) \vec{\nabla} k \\
\left( \mu + \frac{\sigma_\omega}{\sigma_e} \right) \vec{\nabla} \omega
\end{bmatrix} \cdot \bar{a} \tag{A.49}
\]

\[
S = \begin{bmatrix}
0 \\
0 \\
\sqrt{g} \vec{C}_{1+} P_k - C_k \rho \omega k \\
\sqrt{g} \vec{C}_{1+} \bar{C}_{1+} P_k - C_\omega \rho \omega \bar{a}^2
\end{bmatrix} \tag{A.50}
\]

\[
V^i = \xi^i u_1 + \xi^i u_2 + \xi^i u_3 + \xi^i \tag{A.51}
\]

\[
p = \rho (\gamma - 1) \left[ E - \frac{1}{2} \bar{V}^2 - k \right] \tag{A.52}
\]

where \( F^i_c \) (respectively \( F^i_v \)) represents the convective (resp. diffusive) flux in direction \( \xi^i \).
A.3.3 Time discretization

It comes, from the finite volume formulation A.45, that the numerical evolution of the aerodynamic flow field is described by the following semi-discrete formulation:

\[
\frac{\partial}{\partial t} \left( \sqrt{g}q \right)_{\xi^1,\xi^2,\xi^3} = -R|_{\xi^1,\xi^2,\xi^3,t} \tag{A.53}
\]

where \( R \), so called residual, is defined by:

\[
R = \sum_{i=1}^{3} \left[ F^i(\xi^i + \frac{1}{2}) - F^i(\xi^i - \frac{1}{2}) \right] - \sqrt{g}S \tag{A.54}
\]

The time integration of equation A.53 then allows to calculate the solution \( q^{(n+1)} \) at the next time step \( (n+1) \), from the solution \( q \) at one or several previous time steps \( q^{(n)}, q^{(n-1)}, q^{(n-2)}, ... \).

From a numerical point of view, there exists two main types of time discretization scheme:

- **Implicit scheme:**
  The solution on one node at the time step \( (n+1) \) depends on the entire flow field at the same time step, which creates a spatial dependency. Whereas this type of time schemes allows large time step, it also necessitates the resolution of a complex linear system, extremely consuming in CPU and memory resources. Most of the time, matrix conditioning and simplifications are used to simplify the resolution of the system, which introduces numerical jamming during unsteady flow simulations.

- **Explicit scheme:**
  The solution at the next time step \( (n+1) \) is calculated from solutions at previous time steps only. The resolution of such system is simple, straightforward, and does not require important CPU and memory resources. The drawback of the method is, however, that small time step are needed to keep the system numerically stable. This constrain can sometimes lead to unacceptable computing time, especially during unsteady flow simulations. A certain number of numerical methods like residual smoothing or multi-grid techniques are available in order to speed up the convergence of the numerical solution.

Considering the objective of the present work, i.e. the analysis of interactions in unsteady transonic flows, the use of an explicit time marching method was adopted in regards to the numerical perturbations inherently introduced in implicit schemes. One of those, the Runge-Kutta scheme offers the advantage to be a second order precision technique for a simple and straightforward application.
A.3.3.1 Runge-Kutta scheme

The Runge-Kutta scheme technique, based Jameson’s formulation (Jameson, 1993), consists on the evaluation of \( p \) intermediate solutions between the current (\( n \)) and next (\( n+1 \)) time step. Each intermediate time step correction is calculated from the previous one, as follow:

\[
\begin{align*}
q_{0}^{n+1} &= q^n \\
q_{1}^{n+1} &= q^n - \alpha_1 \frac{\Delta t}{\sqrt{g}} R_0 \\
q_{2}^{n+1} &= q^n - \alpha_2 \frac{\Delta t}{\sqrt{g}} R_1 \\
&\quad \vdots \\
q_{k}^{n+1} &= q^n - \alpha_k \frac{\Delta t}{\sqrt{g}} R_{k-1} \\
&\quad \vdots \\
q_{p}^{n+1} &= q^n - \alpha_p \frac{\Delta t}{\sqrt{g}} R_{p-1} \\
q^{n+1} &= q_{p}^{n+1}
\end{align*}
\]  

(A.55)

where \( k \) represents the \( k^{th} \) intermediate time step between (\( n \)) and (\( n+1 \)).

In the present work, a five time steps Runge-Kutta scheme was used with the following coefficients tuned to optimize the convergence of steady state computations:

\[
\begin{align*}
\alpha_1 &= 1/4 \\
\alpha_2 &= 1/6 \\
\alpha_3 &= 3/8 \\
\alpha_4 &= 1/2 \\
\alpha_5 &= 1
\end{align*}
\]  

(A.56)

A.3.3.2 Time step calculation

As any explicit time marching technique, the Runge-Kutta scheme obeys the CFL (Courant-Friedrichs-Levy) condition (Hirsch, 1988a) which basically stipulates that the numerical propagation of information (\( \frac{\Delta t}{\Delta x} \)) cannot be faster than the physical one (\( U + c \)).

\[
CFL = (U + c) \frac{\Delta t}{\Delta x} \leq 1 
\]  

(A.57)

with: \( U \) characteristic velocity scale
\( c \) sound speed
\( \Delta x \) characteristic length scale
The time step $\Delta t$ between each time marching iteration is then calculated based on the CFL value and the local velocity and length scales. In unsteady flow simulation however, the user will prefer to use a global time step, corresponding to the smallest value calculated in the entire domain, in order not to introduce any phase lag in the calculation.

However, the use of a unique characteristic length scale is not suitable to a multidimensional flow calculation. There are indeed regions like boundary layers where diffusion is the preponderant effect. In such regions, the velocity is much smaller in the direction normal to the surface. As a result, the grid density is greater in one specific direction. The time step is therefore calculated in each direction and the smallest value is retained:

$$\Delta t = \min_{i=1,2,3} (\Delta t_i)$$  
with  
$$\Delta t_i = \frac{CFL \parallel \vec{a}_i}{V_i + c}$$  

where: 
- $\Delta t_i$ is the time step associated with the direction $\xi_i$ 
- $\vec{a}_i$ is the covariant basis vector associated to the direction $\xi_i$ 
- $V_i = \vec{V} \cdot \frac{\vec{a}_i}{|\vec{a}_i|}$ is the flow velocity in the direction $\xi_i$.

Besides, as mentioned previously, viscous effects are predominant in certain regions and a time step based on a diffusive scale based on Liamis formulation (Liamis, 1993) was therefore used.

### A.3.3.3 Numerical smoothing

The main disadvantage of explicit schemes is their stability criterion which greatly lower the time step between each numerical iteration. In order to speed up the steady state calculations, two different residual smoothing techniques are available and described further below.

It should however be noted that those smoothing techniques are not suited to unsteady simulations since they either limit the accuracy, scatter the aerodynamic flow properties, or even introduce numerical phase lag which interferes with the unsteady information propagation.

Finally, whereas we speak about residual smoothing, it is actually the correction $\Delta q$ which is smoothed.

#### A.3.3.3.1 Explicit residual smoothing

The direct effect of an "explicit" smoothing is to filter local peaks during the numerical evaluation of the correction between two time steps. At the time step $(n)$, the residuals4 calculated on a specific node are weighted by the residuals of the direct neighboring nodes at first, and then by the second order neighboring nodes. As a result, the numerical information also propagates faster inside the rest of the domain.

---

4 As this smoothing procedure depends on the way the nodes are nested, the same node swiping technique was used in each direction, at each time step.
\[ \Delta q(i, j, k)^* = \left(1 - \frac{\beta_{exp}}{12}\right) \Delta q(i, j, k) + \frac{\beta_{exp}}{6 \times 24} \sum \left[ \begin{array}{c} \Delta q(i \pm 1, j, k) + \Delta q(i \pm 1, j \pm 1, k) + \\ \Delta q(i, j \pm 1, k) + \Delta q(i, j, k \pm 1) \end{array} \right] \]  
(A.59)

\[ \Delta q(i, j, k)^L = \left(1 - \frac{\beta_{exp}}{12}\right) \Delta q(i, j, k)^* + \frac{\beta_{exp}}{6 \times 36} \sum \left[ \begin{array}{c} \Delta q(i \pm 2, j, k) + \Delta q(i \pm 2, j \pm 2, k) + \\ \Delta q(i, j \pm 2, k) + \Delta q(i, j, k \pm 2) \end{array} \right] \]  
(A.60)

where \( \Delta q(i, j, k)^L \) is the smoothed correction
\( \Delta q(i, j, k)^* \) is a temporary estimation of the correction
\( \beta_{exp}(0 \leq \beta_{exp} \leq 1) \) is the weight put on the neighboring residuals.

### A.3.3.3.2 Implicit residual smoothing

The "implicit" smoothing used in PROUST is based on Lerat (Lerat, 1985) and Liamis (Liamis, 1993) formulations. The technique consists of weighting the correction depending on the correction of the surrounding nodes:

\[ \left[ \hat{I} + \beta_{imp}^\star \hat{\sigma}^2 \right] \circ \Delta q_{imp} = \Delta q_{exp} \]  
(A.61)

where \( \hat{I} \) is an operator
\( \Delta q_{exp} = q_0 - q_0^{n-1} \) is the correction before applying the implicit smoothing
\( \Delta q_{imp} = q_i - q_i^{n-1} \) is the implicit correction
\( \beta_{imp}^\star \) is the weight coefficient

In the case of a 3D flow calculation, Lerat’s implicit correction smoothing, coupled to an ADI method can be re-written as follow:

\[ \left[ \Delta q^\star + \beta_{imp}^\star \sigma_i^2 \delta_i \left( (\rho_{spect}')^2 \delta_i (\Delta q^\star) \right) \right] = \Delta q_{exp} \]  
(A.62)

\[ \left[ \Delta q^{**} + \beta_{imp}^\star \sigma_j^2 \delta_j \left( (\rho_{spect}')^2 \delta_j (\Delta q^{**}) \right) \right] = \Delta q^\star \]  
(A.63)

\[ \left[ \Delta q_{imp} + \beta_{imp}^\star \sigma_k^2 \delta_k \left( (\rho_{spect}')^2 \delta_k (\Delta q_{imp}) \right) \right] = \Delta q^{**} \]  
(A.64)

where \( \sigma_i^2 = \frac{\Delta t}{\Delta x^2} \) is the inverse of a velocity propagation of the numerical information
\( \rho'_{spect} = |V'| + a \| \mathbf{a}_i \| \) is the spectral radius of the Jacobian matrix \( \frac{\partial F}{\partial q} \)
\( \delta_i \) is a spatial operator centered on the direction \( i \)
\( \beta_{imp}^\star = \frac{\beta_{imp}}{\beta_{imp}} \) is the implicit weight factor

The resolution of the above system is done by a Thomas algorithm and has a low CPU and memory cost. Besides, the order in which the directions are swiped changes randomly in order not to foster a particular direction. The smoothing correction is also performed at each intermediate time step of the Runge-Kutta scheme.
A.3. NUMERICAL METHODS

Whereas the scheme is stable for $\beta_{\text{imp}} < 1$, it also introduces a second order error, which definitely limits its use in unsteady flow calculations.

A.3.4 Spatial discretization

From the finite volume formulation A.45, we now have to evaluate the fluxes through the boundaries of the integration cell. The equations A.28-A.32 describing the unsteady motion of a compressible turbulent flow, constitute an hybrid partial differential equation system. That is to say, the diffusive and thermal conductivity terms are described by elliptic equations whereas the convective terms are hyperbolic by nature. Those two types of equation systems correspond to different physical phenomena and thus require different numerical discretization. As a result, the flux evaluation will be dissociated into two parts, as suggested in equation (A.47): the convective fluxes (eq. A.48) and the diffusive fluxes (eq. A.49).

The hyperbolic character of a certain equations suggests a wave propagation system, with favored velocity and directions of propagation in the flow, as well as possible discontinuities. It appears that upwind schemes are especially suited to represent such physical phenomena. They are based on the separation of both the upstream and downstream influences of physical perturbations which propagate following the characteristic waves of a Niemann problem. This kind of schemes allows a good capture of the discontinuities and naturally possess an artificial dissipation, which makes them unconditionally stable. They, moreover, nicely deal with undulatory phenomena and thus allow a good representation of unsteady flows.

On the other hand, elliptic equation systems, basically describing diffusive phenomena, does not feature preponderant directions in the flow field, and will be discretised using a centered scheme.

A.3.4.1 Discretization of the convective terms

From the finite volume formulation A.45, the convective terms are represented by equation A.48:

$$
F^i_c = \sqrt{\gamma} \begin{bmatrix}
\rho V^i \\
\rho V^i V^j + p\bar{a}^i \\
(pE + p)V^i - p\xi^i_j \\
\rho kV^i \\
\rho \omega V^i
\end{bmatrix}
$$

(A.65)

The above equation has to be evaluated at the interface between two finite volume cells. The last two terms, however, correspond to the turbulence closure problem and will be evaluated later on.
A.3.4.1.1 Upwind Roe scheme

As mentioned previously, upwind schemes rely on basic wave propagation concepts and allow a fairly good description of unsteady transonic flows.

Different schemes have been developed in PROUST in order to offer alternatives to solve complex flow configurations. In the present framework, however, only one scheme which was thought to give the best results was used and will thus be described in the paragraph. The reader is kindly advised to refer to other references (Aubert, 1993; Smati, 1997; Oliveira, 1999) to complete his knowledge on the different options offered by the CFD tool PROUST.

In order to solve an undulatory problem, Godunov (Godunov et al., 1979) proposed to solve a Riemann problem at the interface between two cells of the discretised domain. However, considering the high resolution costs of such problem, Roe (Roe, 1981) proposed to an alternative resolution by approaching the solution only.

Roe scheme is usually associated to a Flux Difference splitting method (figure A.3) in which the fluxes are separated into an upstream and downstream propagating parts (in opposition to a Variable Difference Splitting). The convective flux at the interface $\xi_{i-\frac{1}{2}}$ can then be written:

$$F^i_c(Q^L_{i-\frac{1}{2}}) = \frac{1}{2} \left[ F^i_c(Q^L_{i-\frac{1}{2}}) + F^i_c(Q^R_{i-\frac{1}{2}}) \right] - \sqrt{g} \left| K^i \right| \frac{Q^L_{i-\frac{1}{2}} - Q^R_{i-\frac{1}{2}}}{2}$$

(A.66)

with:

$$|K^i| = R^i(Q_{Roe}) |\Lambda^i| L^i(Q_{Roe})$$

The fluxes $F^i_c(Q^{L,R}_{i-\frac{1}{2}})$ are evaluated from the complete convective flux expression (equation A.65), while $K^i$ is the Jacobian matrix of the convective fluxes based on variables pondered by the density, and
A.3. NUMERICAL METHODS

\[ Q_{\text{roe}} = \begin{bmatrix} \sqrt{\rho_L \rho_R} \\ \sqrt{\rho_L^2 \rho^2 + \rho^2} \\ \sqrt{\rho_L^2 + \rho^2} \\ \sqrt{\rho_L^2 \rho_h + \rho^2 \rho_h} \\ \sqrt{\rho^2 + \sqrt{\rho^2}} \end{bmatrix} \]  \hspace{1cm} (A.67)

The matrix \( K^i \) is evaluated from the eigenvectors matrices \( R^i \) and \( L^i \) composed by the eigenvectors right or left written either in lines or in columns. The diagonal matrix \( \Lambda^i \) is composed by the eigenvalues of the endomorphism \( \left( \frac{\partial F^i}{\partial \sqrt{gq}} \right) \) which are \( \lambda_i = V^i \pm c ||\vec{a}^i||. \)

A.3.4.1.2 Higher order scheme

Whereas for a first order scheme, the convective fluxes are expressed using the direct surrounding nodes, high order schemes need an interpolation using further nodes. However, a classical linear interpolation usually ends up with an oscillation of the solution (Hirsch, 1988b) since high order schemes are no longer monotone.

In order to prevent those oscillations to occur, a Monotone Upstream-centered Scheme for Conservation Laws (MUSCL) technique (Van-Leer, 1979) was used together with a Total Variation Diminishing (TVD) scheme. In this approach, the conservative variables are interpolated on the cells interfaces (Figure A.4), and the fluxes are thereafter evaluated from those values. In opposition with a classical method in which the fluxes are fist evaluated at the nodes, and then interpolated at the cell’s interface.

\[ Q_{i+1/2} \]

Figure A.4: High order interpolation, MUSCL approach

In order to avoid any oscillation or overshoot nearby a discontinuity of the solution (such as a shock wave) and ensure a monotone behavior of the scheme, the amplitude of local gradients is limited using a non-linear function called limiter. Moreover, the metrics \( ||\vec{a}^i|| \) and the Jacobian \( \sqrt{g} \) of the physical transform are then evaluated in an accurate way directly on the interfaces and not interpolated from the center of the cells.

Finally, the expression of the convective fluxes is written using two contributions (Left and Right), which are interpolations of the respective upstream and downstream influences.
at the interface $\xi^{i-\frac{1}{2}}$.

\[ F^c_0(\xi^{i-\frac{1}{2}}) = F^{i+}(Q^c_{\xi^{i-\frac{1}{2}}}) + F^{i-}(Q^a_{\xi^{i-\frac{1}{2}}}) \]

(A.68)

where

\[ Q^c_{\xi^{i-\frac{1}{2}}} = Q^{c/1}(q_{\xi^{i-1}}, q_{\xi^{i-2}}) \]
\[ Q^a_{\xi^{i-\frac{1}{2}}} = Q^{a/1}(q_{\xi^{i-1}}, q_{\xi^{i+1}}) \]

with

\[ Q^{\xi/1}(q_0, q_1, q_2) = q_1 + \frac{1}{4} \Phi [(1 + \kappa \Phi) (q_0 - q_1) + (1 - \kappa \Phi) (q_1 - q_2)] = \frac{1}{4} \Phi [1 + \kappa \Phi] q_0 + \left[1 - \frac{1}{2} \kappa \Phi^2\right] q_1 - \frac{1}{4} \Phi [1 - \kappa \Phi] q_2 \]

(A.69)

and

\[ \Phi = \Phi(q_0, q_1, q_2) \]

(A.70)

The spatial accuracy is thus set up by the function $\Phi$, the limiter, which value is bounded between 0 and 1, respectively for a first and second order scheme. As a result, the parameter $\kappa$ is such that, for $\Phi = 1$, we obtain:

- $\kappa = -1$: Second order totally uncentered scheme
- $\kappa = 0$: Fromm’s scheme
- $\kappa = \frac{1}{3}$: Third order partially uncentered scheme
- $\kappa = 1$: Second order centered scheme

The evaluation of the spatial accuracy assumes a mono-dimensional and uniform node distribution, and does not account for any other assumptions. Thus a non uniform grid distribution may decrease the accuracy. This is in contradiction with the finite volume formulation in which absolutely no regularity of the mesh is needed. Practically, a cell size ratio of 1.3 between neighboring cells has to be respected in order to keep a second order accuracy.

### A.3.4.1.3 Spatial accuracy limiters

In order to prevent any oscillation of the solution, a different value of the limiter $\Phi$ is evaluated, at each interface, as a function of the local gradients of the aerodynamic flow field. The function $\Phi$ then varies continuously between 0 and 1. Moreover, a different limiter is used for each variable in order to limit the accuracy only when necessary.

Basically, the limiter is a function of the local gradients, calculated on each side of the interface:

\[ \Phi(q_0, q_1, q_2) = \Phi(q_0 - q_1, q_1 - q_2) \]

(A.72)

The important criteria when selecting a limiter are its continuity and derivability in order to ensure a monotone change of the spatial and temporal order of the scheme. The
A.3. NUMERICAL METHODS

The limiter used here is based on Van Albada’s formulation, which was modified by Mulder (Mulder and Vann-Leer, 1983).

\[ \Phi(q_0, q_1, q_2) = \frac{2 |(q_0 - q_1)(q_1 - q_2)| + \epsilon}{(q_0 - q_1)^2 + (q_1 - q_2)^2 + \epsilon} \]  \hspace{1cm} (A.73)

where \( \epsilon \) is a very small constant (\( \epsilon \approx 10^{-6} \)) introduced to avoid a division by zero when the local flow gradient are zero.

A.3.4.2 Discretization of the diffusive terms

The diffusive flux \( F_v^i \) (eq. A.47) through the interface of an cell in the direction \( \xi^i \) is defined by:

\[ F_v^i = \sqrt{g} \vec{F}_v \cdot \vec{a}^i \]  \hspace{1cm} (A.74)

where:

\[ \vec{F}_v = \left. \begin{array}{c} 0 \\ \bar{\tau}_{t+t} \end{array} \right| \gamma \left( \frac{\mu}{\nu} + \frac{\mu_t}{\nu_t} \right) \vec{e} + \tau_{t+t} \vec{V} + \left( \mu + \frac{\mu_t}{\sigma} \right) \vec{k} \]  \hspace{1cm} (A.75)

with:

\[ \bar{\tau}_{t+t} = \left( \lambda + \lambda_k \right) \vec{\nabla} \cdot \vec{V} \vec{I} + \left( \mu + \mu_t \right) \left( \vec{\nabla} \vec{V} + \vec{\nabla} \vec{V}^t \right) - \frac{2}{3} R e \rho k \vec{I} \]  \hspace{1cm} (A.76)

\[ \vec{\nabla} u_k = \sum_{j=1}^{3} \vec{a}^j \frac{\partial u_k}{\partial \xi^j} \]  \hspace{1cm} (A.77)

\[ \vec{\nabla} \cdot \vec{V} = \sum_{j=1}^{3} \vec{a}^j \frac{\partial V}{\partial \xi^j} \]  \hspace{1cm} (A.78)

\( \vec{\nabla} \vec{V} \) (resp. \( \vec{\nabla} \vec{V}^t \)) is composed by the vectors \( \left( \vec{\nabla} u_k \right)_{k=1,2,3} \) written in line (resp. in column).

The aerodynamical variables are calculated using a centered scheme at the interface:

\[ Q_{\xi^i-\frac{1}{2}} = \frac{1}{2} \left[ q(\xi^i - 1) + q(\xi^i) \right] \]  \hspace{1cm} (A.79)

The spatial derivative of a scalar \( A \) (\( u_k, \rho \) or \( k \)) is discretised as follow:

\[ \vec{\nabla} A |_{\xi^i-\frac{1}{2}} = \vec{a}^i |_{\xi^i-\frac{1}{2}} \left[ A(\xi^i) - A(\xi^i-1) \right] + \sum_{j=1}^{3} \vec{a}^j |_{\xi^i-\frac{1}{2}} \frac{A(\xi^j+1) - A(\xi^j-1)}{2} \]  \hspace{1cm} (A.80)

The above expression then allows to calculate all spatial derivative (\( \vec{\nabla} k, \vec{\nabla} u_k, \vec{\nabla} e \) et \( \vec{\nabla} \vec{V} \)).
APPENDIX A. NUMERICAL MODEL

A.3.4.3 Discretization of the turbulent equations

The seven equations A.28-A.32 constitute a coupled mathematical system. This coupling occurs through the pressure, which contains the turbulent kinetic energy, and the Reynolds tensor \( \tau \) in the conservative equations. Besides, the turbulent kinetic energy gradient \( \nabla k \) is present in the conservative equation of energy.

From a numerical point of view, the coupling between the different equations does not exist any more with an explicit time resolution. The solution at the time \( n + 1 \) only depends on the solution at \( n \). However, the evaluation of the convective fluxes relies on physical criteria which implicitly accounts for the coupling. Thereby, the resolution of turbulent variables transport equations theoretically need a modification of the convective fluxes formulation. As a result, the resolution of the turbulent equations will be decoupled from the rest of the system.

A.3.4.3.1 Turbulent convective fluxes

The equations A.31 and A.32 are basically conservative transport equations for the turbulent quantities \( k \) and \( \omega \), generically noted \( Y \). By the convective fluxes discretization, those variables will be seen as passive scalars, that is to say they will be passively convected with the flow. This also implies that the resolution of the turbulent transport equations has no influence on the rest of the system, which is in coherent with the decoupling assumption mentioned previously.

As the variable \( Y \) is passively convected by the flow, the determination of the mass flux allows the calculation of the convective flux of the quantity \( Y \) at the interface between two cells:

\[
F^i_c(\xi^{i-\frac{1}{2}}) = F^i(\xi^{i-\frac{1}{2}}) = (\rho Y)^{\xi^{i-\frac{1}{2}}} \frac{F^i_{\xi}}{\rho} \tag{A.81}
\]

where \( F^i_{\xi}(\xi^{i-\frac{1}{2}}) = \sqrt{g} \rho V i^{i-\frac{1}{2}} \) is the total convective flux associated with the continuity equation.

By interpolation at the interface \( \xi^{i-\frac{1}{2}} \), we obtain:

\[
F^i_c(\xi^{i-\frac{1}{2}}) \simeq \rho Y_{\xi^{i-\frac{1}{2}}} \frac{F^i_{\xi}(\xi^{i-\frac{1}{2}})}{\rho_{\xi^{i-\frac{1}{2}}}} \tag{A.82}
\]

Within the frame of a MUSCL formulation and a Flux Difference Splitting approach as described in the previous sections, one can either use a centered or uncentered formulation (upstream or downstream depending on the mass flux):
A.3. NUMERICAL METHODS

\[ F_i^p(\xi^i - \frac{1}{2}) = \begin{cases} 
\rho Y_{\mid L}^i F_i^{p, L}(\xi^i - \frac{1}{2}) & \text{if } F_i^{p, L}(\xi^i - \frac{1}{2}) \geq 0 \\
\rho Y_{\mid R}^i F_i^{p, R}(\xi^i - \frac{1}{2}) & \text{if } F_i^{p, R}(\xi^i - \frac{1}{2}) < 0 
\end{cases} \] (A.83)

A.3.4.3.2 Turbulent diffusive fluxes

The diffusive flux \( F_i^v \) (equation A.49) through a cell boundary in the direction \( \xi^i \) is only function of turbulent variables gradients \( \vec{\nabla} k \) and \( \vec{\nabla} \omega \).

\[ F_{i \mid k, \omega}^v = \sqrt{g} \left[ \left( \frac{\mu + \frac{\mu_t}{\alpha}}{\mu + \frac{\mu_t}{\alpha}} \right) \vec{\nabla} k \right] \cdot \vec{a} \] (A.84)

This flux is discretised using a centered formulation. Thus, the gradient of a scalar quantity can be written following a finite difference formulation in the curvilinear space:

\[ \vec{\nabla} Y_{\mid \xi^i - \frac{1}{2}} = \vec{a} \mid \xi^i - \frac{1}{2} \left[ Y_{\xi^i} - Y_{\xi^i - \frac{1}{2}} \right] + \sum_{j \neq i} \vec{a} \mid \xi^i - \frac{1}{2} \frac{Y_{\xi^j + \frac{1}{2}} - Y_{\xi^j - \frac{1}{2}}}{2} \] (A.85)

The source term \( S \) (equation A.50) is a volume term calculated using a centered formulation:

\[ S_{\xi} = \sqrt{g} \left[ \frac{S_k}{S_\omega} \right] \] (A.86)

The expressions of \( S_k \) and \( S_\omega \) show conservative variables and a turbulent kinetic energy production term \( P_k \). In order to evaluate \( S \), the gradient and divergence of the velocity vector have to be calculated:

\[ \vec{\nabla} u_{\mid \xi^1, \xi^2, \xi^3} = \vec{a} \mid \xi^1, \xi^2, \xi^3 \sum_{m=1}^{3} \frac{u_{\xi^m}^{m+1} - u_{\xi^m}^{m-1}}{2} \] (A.87)

The velocity gradient is composed by the vectors \( \vec{\nabla} u_{\mid i = 1, 2, 3} \) written in lines, and its divergence equals to the trace of the velocity gradient tensor.

A.3.4.3.3 Numerical filters

The stability of the turbulent equations is very to be sensitive, especially during an explicit time resolution. Indeed, the sources are dominant in those equations, which make them very sensitive to a small perturbation. Moreover, their modelling is criticizable and can lead to non physical solutions during the convergence. This tendency becomes more pronounced with severe numerical conditions such as higher CFL, local time stepping,
Therefore, when the turbulent variables reach negative values, many authors (Vasiliopoulos, 1996; Tsanga, 1997) use limiters and numerical filters. However, the formulation presented here is slightly different and based on the fact that any change of the solution creates a perturbation in the shape of a numerical wave, which has to be evacuated. Thus, a minimum of numerical correction is applied to the turbulent variables:

- First of all, no threshold is used for the turbulent variables, thus $k$ and $\omega$ can be zero.

- Then, a filter of the negative values below a certain threshold (negative) is used. Two different thresholds, $\iota_k$ et $\iota_\omega$, are used for $k$ and $\omega$. The filter is a simple interpolation between the neighboring nodes. When filtering the dissipative variable $\omega$, the level of the turbulent viscosity $\mu_t$ is checked and bounded to $\alpha_l \mu_l$, with $\alpha_l = 1000$. If a filtered node belongs to a boundary, the imposed value is then the one fixed by its boundary condition.

### A.3.5 Boundary discretization

The description of the numerical method has, until now, not accounted for the limitation of the computational domain. The set of equations A.28-A.32 has been discretised for any node inside the computational domain. In order to account for external influences such as guiding adiabatic walls, steady subsonic inflow or unsteady outflow properties, a special treatment has to be realized on the nodes contiguous to a boundary. The method used in PROUST (Aubert, 1993) is based on the creation of external supplementary nodes, and the expression of compatibility relations. The nodes located on the boundaries are then treated exactly like the nodes inside the computational domain. This treatment ensures the coherence between the spatial-temporal schemes used both inside and at the boundaries of the computational domain. The scheme accuracy order is then preserved, avoiding at the same time any eventual parasite time phase lags.

Five different types of boundary conditions will be considered in the present work: "free" boundaries like inlet/outlet boundaries, adherent and sliding solid walls, symmetry planes, and inter-domain boundaries. In the general case, boundary conditions can be separated into two kinds: geometrical and physical conditions. In the first category, a geometrical relation is established between the boundary nodes and the internal nodes, whereas in the second category, a physical condition is introduced in the system as a new equation to resolve.

Those two types of conditions are applied in an unsteady local mono-dimensional way at the boundaries. Therefore, in the case of an edge or a corner, there is an excess of boundary conditions compared to the set of equations and mixing those conditions sometimes ends up in numerical interferences. Whereas geometrical conditions can easily be combined, physical conditions are more delicate to handle because of the physical balance that has to be respected. Which relation should be preponderant in case, for instance, of a corner between an outlet boundary, an adiabatic adherent wall, and a symmetrical...
plane boundary? The choice concerning the boundary condition to apply thus depends on the physics one wants to introduce into the computation.

The present section intends to describe the treatment of the boundary conditions used during the different computations performed in this work. First, the general method based on the creation on supplementary planes and compatibility relations will be presented. Then, the different geometrical and physical boundary conditions and their discretization will be described.

A.3.5.1 Extension of the computational domain

The spatio-temporal discretization scheme inside the computational domain uses, in order to determine the solution at any given node, several surrounding nodes as described in the previous sections of this Appendix. Depending on if the calculation is inviscid or viscous, and if the spatial scheme is a first or second order scheme, the calculation molecule will be composed of seven to twenty five nodes as illustrating in Figure A.5.

In order to treat the boundary nodes as internal nodes, that is to say apply the same scheme on the nodes located at the boundaries, the computational domain is extended to two supplementary planes in each direction outside the physical domain. As the "viscous" computing molecules also include the nodes in the cross directions, a supplementary edge has to be created at each intersections between the planes. As a result, the entire computational domain (Figure A.6) is presented with the same spatio-temporal scheme, which greatly simplify the accounting of the boundary conditions. All necessary information, geometrical and aerodynamical, will be calculated on the supplementary nodes from both the internal nodes and the boundary conditions.

Practically, the extend of the computational domain simplifies the management of the boundary conditions but it increases the size of the mesh and thus the required memory resources.
A.3.5.2 Geometrical boundary conditions

The imposition of the geometrical boundary conditions is directly obtained from the external supplementary nodes. Indeed, their geometrical position $\vec{r} = f(x, y, z)$ and the aerodynamical field at their location are determined by geometrical properties obtained from each different boundary condition.

The different geometrical boundary conditions used in this framework have been summarized below:

**Plane symmetry:**
With this condition, the boundary is seen as a symmetry plane so that the external supplementary nodes simply mirror the internal nodes. The symmetry plane is determined by a normal vector $\vec{n}$ and a node at the boundary. The following relations can therefore be written:

Geometrical position: $\vec{r}^{ext} = \vec{r}^{int} - 2(\vec{r}^{int}.\vec{n})\vec{n}$
Scalar variable: $A^{ext} = A^{int}$

**Inter domain connection:**
The connection between two adjacent domains is obtained by a collocative overlapping of the boundary nodes. All necessary information is thereafter copied from one domain to the external supplementary nodes of the other domain. The exchange of information is therefore performed without any interpolation and loss of accuracy. Between the two collocative and adjacent domains I and II, the boundary condition can be written:

Geometrical position: $\begin{cases} \vec{r}^{ext}_I = \vec{r}^{int}_I \\ \vec{r}^{ext}_{II} = \vec{r}^{int}_{II} \end{cases}$
Scalar variable: $\begin{cases} A^{ext}_I = A^{int}_I \\ A^{ext}_{II} = A^{int}_{II} \end{cases}$
A.3.5.3 Physical boundary conditions

For each boundary associated with the aerodynamical conditions, an external constrain has to be imposed while resolving the equation system. This can be done using the compatibility relation method (Chakravarthy, 1983), which is based on the hyperbolic property of the convective part of the equations. The undulatory characteristics of the equations is used to introduce physics in the numerical treatment of the boundary conditions.

A.3.5.3.1 Mono-directional formulation

The hyperbolic part of the equations is decomposed into derivatives which are either secant or tangent to the boundary, while the other terms are grouped in a source term $S$. Whereas the tangential derivative are treated like internal nodes, the others take part in the boundary condition treatment. As a result, the equation A.39 on page 259 can be rewritten as:

$$
\frac{\partial}{\partial t} \left( \sqrt{g}q \right) + \sum_{j=1}^{3} \frac{\partial F_j}{\partial \xi_j} + S = 0 \quad (A.88)
$$

The projection of the conservative equations on the characteristic surface is performed by multiplying the above equation by the left eigenvector matrix of the Jacobian $K^i$ associated to the convective flux $F^i_c$. The new system corresponds to the compatibility equations:

$$
L^i \frac{\partial}{\partial t} \left( \sqrt{g}q \right) + L^i \left( \frac{\partial F_c^i}{\partial \xi} + \sum_{j=1}^{3} \frac{\partial F_j}{\partial \xi_j} + S \right) = 0 \quad (A.89)
$$

with:

$$
K^i = \left[ K_a^i \right]_{a \in [1;5]} \quad K_a^i = \frac{\partial F^i_c}{\partial \left( \sqrt{g}q \right)} \quad (A.90)
$$

By introducing the Jacobian matrix, it comes:

$$
L^i \frac{\partial}{\partial t} \left( \sqrt{g}w^i \right) + L^i K^i \frac{\partial}{\partial \xi} \left( \sqrt{g}w^i \right) + L^i \left( \sum_{j=1}^{3} \frac{\partial F_j}{\partial \xi_j} + S \right) = 0 \quad (A.91)
$$

Finally, by change of variable is performed, and equation A.91 can be rewritten:

$$
\frac{\partial}{\partial t} \left( \sqrt{g}w^i \right) + \Lambda^i \frac{\partial}{\partial \xi} \left( \sqrt{g}w^i \right) + L^i \left( \sum_{j=1}^{3} \frac{\partial F_j}{\partial \xi_j} + S \right) = 0 \quad (A.92)
$$

where: $\Lambda^i$ is the diagonal matrix containing the eigenvalues $\lambda_i$ of $K^i$ ($\Lambda^i L^i = L^i K^i$)
APPENDIX A. NUMERICAL MODEL

The above system basically describes the transport of characteristics variables \( w^i \) in the direction \( \xi^i \), at the speed \( \lambda_i \). Thus, the eigenvalues of the Jacobian matrix \( K^i \) can be interpreted as speed of propagating waves perpendicular to the characteristic surface. The sign of the eigenvalues thus tells in which direction the information is propagating through the boundary. If \( \lambda_i \) is positive, the associated wave is exiting the computational domain, and its describing equation is kept the same. In case of a negative value of \( \lambda_i \), however, the associated equation is replaced by an equation which accounts for the fact that the associated wave is entering the domain and includes external information.

In order to keep the same temporal scheme everywhere, and easily integrate the compatibility equations, the external information is expressed as a partial derivative equation. Basically, the temporal variation \( B^*(t) \) of a physical variable \( B(q) \) has to be imposed at the boundary at any time.

\[
B(q) = B^*(t) \tag{A.93}
\]

It comes:

\[
B^{(n+1)} - B^*^{(n+1)} = \left\{ B^{(n)} + \left[ \frac{\partial B}{\partial (\sqrt{g}q)} \right]^{(n)} \left( \frac{\partial (\sqrt{g}q)}{\partial t} \right)^{(n)} \Delta t + O(\Delta t) \right\} - B^*^{(n+1)} \tag{A.94}
\]

and then:

\[
\left[ \frac{\partial B}{\partial (\sqrt{g}q)} \right] \frac{\partial}{\partial t} (\sqrt{g}q) = -\frac{1}{\Delta t} \{ B(q^n) - B^*(t^{n+1}) \} \tag{A.95}
\]

This equation is similar to equation A.89. By merging them together, it comes:

\[
\frac{\partial}{\partial t} (\sqrt{g}q) + \left[ P^{1i} \right]^{-1} \left[ P^{2i} \right] \left\{ \sum_{j=1}^{3} \left[ F^i_j (\xi^j + \frac{1}{2}) - F^i_j (\xi^j - \frac{1}{2}) \right] + S \right\} - \left[ P^{3i} \right] = 0 \tag{A.96}
\]

where:

\[
\left[ P^{1i} \right] = \left[ \frac{L^i_{\lambda_i>0}}{\sqrt{|g|}} \right], \quad \left[ P^{2i} \right] = \left[ L^i_{\lambda_i>0} \right], \quad \text{and} \quad \left[ P^{3i} \right] = \left[ \begin{array}{c} 0 \\ -\frac{B(q)-B^*(t)}{\Delta t} \end{array} \right]
\]

It should be noted that the terms in the direction \( \xi^i \) are discretised in an uncentered way towards the inside of the computational domain since associated with an outgoing information. The diffusive and turbulent fluxes, as well as the tangential terms are treated exactly like inside the domain.

A.3.5.3.2 Different types of physical boundary conditions

In this paragraph have been summarized the different types of physical boundary conditions used in the CFD tool PROUST (Smati, 1997).

**Free boundary:**
A free boundary is such that the fluid can either enter or exit the computational domain, depending on its speed and direction. Four different configurations can therefore be encountered:
A.3. NUMERICAL METHODS

① Supersonic outlet:
The information cannot propagate upstream. The flow is entirely described by the
inlet conditions. No condition are therefore imposed at the outlet, and all equations
are preserved.

② Subsonic outlet:
The system contains only one negative eigenvalue. The corresponding equation
is then exchanged by the linearized expression of $p$ in order to impose the static
pressure at the outlet:

$$B(q) = P_t = \rho(\gamma - 1) \left[ E - \frac{1}{2} \vec{V}^2 \right] \quad \text{with} \quad \frac{\partial B}{\partial (\sqrt{gq})} = \frac{\gamma - 1}{\sqrt{g}} \begin{vmatrix} \frac{1}{2} \vec{V}^2 \\ -u_1 \\ -u_2 \\ -u_3 \\ 1 \end{vmatrix}$$

(A.97)

③ Subsonic inlet:
This time, four physical boundary conditions have to be imposed. Those are the
total pressure, total temperature, and two information on the velocity vector:

- $B(q) = P_t = \rho(\gamma - 1) \left[ E - \frac{1}{2} \vec{V}^2 \right] \quad \text{with} \quad \frac{\partial B}{\partial (\sqrt{gq})} = \frac{\gamma - 1}{\sqrt{g}} \begin{vmatrix} \frac{1}{2} \vec{V}^2 \\ -u_1 \\ -u_2 \\ -u_3 \\ 1 \end{vmatrix}$

(A.98)

- $B(q) = T_t = 1 \gamma C_v \rho \left[ p + \rho E \right]$ \quad \text{with} \quad \frac{\partial B}{\partial (\sqrt{gq})} = \frac{1}{\gamma C_v \rho} \left[ \frac{\partial p}{\partial (\sqrt{gq})} + \frac{1}{\sqrt{g}} \left( \frac{1}{\rho} \left[ p + \rho E \right] \delta_k + \delta_k \right) \right]$ \quad \text{(A.100)}

$B(q) = \tan (\alpha_{ij}) = \frac{u_j}{u_i}$ \quad \text{with} \quad \frac{\partial B}{\partial (\sqrt{gq})} = \frac{1}{\sqrt{g} \rho u_i^2} \left[ u_i \delta_k(i+1) - u_j \delta_k(i+1) \right]$ \quad \text{(A.102)}

(A.101)

(A.103)

④ Supersonic inlet:
All eigenvalues are negative, which means that all information are entering the do-
mains. Thus five physical boundary conditions have to be imposed. Those are the
total pressure, total temperature, and three velocity components.
APPENDIX A. NUMERICAL MODEL

Adiabatic sliding boundary:
With this condition, the flow is tangent to the boundary and the heat flux is zero perpendicularly to the surface. Three eigenvalues are zero and one is strictly negative. Practically however, only one condition can be imposed. Therefore, only the tangential velocity component is imposed. The adiabatic condition is indirectly respected by the fact that there is no heat exchange in perfect fluid flow.

\[
B(q) = V^i \quad (A.104)
\]

\[
B^*(t) = 0 \quad \text{with} \quad \frac{\partial B}{\partial(\sqrt{g}q)} = \frac{1}{\sqrt{g}} \begin{vmatrix}
-(V^i - \xi^i_1) \\
\xi^i_2 \\
\xi^i_3 \\
0
\end{vmatrix} \quad (A.105)
\]

Adiabatic adherent boundary:
This time, all velocity components are zero on the boundary, as well as the heat flux.

Non reflective boundary:
Whenever the physical conditions are not well known at a boundary, a non reflecting condition avoids unknown incoming waves to enter the computational domain. Besides, the condition also evacuates numerical waves generated inside the domain. To do so, any information travelling with a negative speed is transformed in a stationary wave by cancelling its speed. This type of condition differs from the free boundary condition in which an incoming wave is partly reflected back inside the domain.

The method used is based on Giles (Giles, 1988) and consists of cancelling all negative eigenvalues, and implicitly transform the incoming waves into stationary waves. The matrices \([P^1]\) and \([P^2]\) are then strictly made of left eigenvectors, whereas the \([P^3]\) matrix is entirely zero.

This condition is essential during unsteady simulation in order to evacuate all instabilities generated by the flow field. Its major disadvantage is that no other physical boundary condition can be imposed at the same time. It is although possible to partly mix this condition with another one, as presented below.

Permeability boundary:
In order to avoid reflections of incoming travelling waves back into the computational domain but still impose a physical quantity, a permeability condition has been developed as a linear combination between the non reflecting condition and a free boundary condition. As a result, the matrices \([P^1]\), \([P^2]\), and \([P^3]\), of equation A.96 can be rewritten using
A.4. CONCLUSION

χ as a permeability coefficient:

\[
\begin{align*}
[P^1_i] &= (1 - \chi) \left[ \begin{array}{c} L_{i}^{(\lambda_i>0)} \frac{\partial B}{\partial q} \\ 0 \end{array} \right] + \chi (L^i) \\
[P^2_i] &= (1 - \chi) \left[ \begin{array}{c} L_{i}^{(\lambda_i>0)} \\ 0 \end{array} \right] + \chi (L^i) \\
[P^3_i] &= (1 - \chi) \left[ \begin{array}{c} 0 \\ \frac{B(q) - B(t)}{\Delta t} \end{array} \right]
\end{align*}
\]

The value of the permeability coefficient has to be chosen as a compromise between the imposition of physical condition and the non reflecting condition. Practically a value below 50% is sufficient to considerably reduce the wave reflections.

A.3.6 Parallel computation

Many reasons make a parallel computation really attractive. First of all, a parallel code leads to a consequent decrease in the memory load per processor, and most important, to acceptable computing times. This is indeed a requirement considering the huge need of an unsteady turbulent 3D Navier-Stokes simulation on a fine mesh. Moreover, the use of a structured mesh makes the parallel architecture of the code optimal. The mesh is then divided into several sub-domains which are individually mastered by a single process on its respective processor. On the highest hierarchy level of a parallel computation stands a Consul process which reads the initial data and spawns the different Master processes. Each process thereafter manages its respective sub-domain and communicate with its surrounding neighbors to exchange information on their collocative boundaries.

A.4 Conclusion

This Appendix was dedicated to the detailed description of the numerical methods used to resolve the set of equations modelling an unsteady compressible turbulent flow.

First of all, the instantaneous Navier-Stokes equations were written in a conservative form, statistically averaged to account for turbulence effects, and finally made dimensionless to facilitate the numerical treatment.

Secondly, the equations were discretised using a finite volume approach in curvilinear structured mesh. Their time integration was realized using a five steps Runge-Kutta explicit scheme. Different acceleration techniques, based on correction smoothing, were described. The discretization of the convective fluxes was thereafter presented using centered or upwind uncentered schemes based on a MUSCL approach. The turbulent equations were discretised following the same method. Finally, the numerical treatment of different geometrical and physical boundary conditions, based on compatibility relations and the extension of the computational domain, was exposed.
Figure A.7: parallel computation diagram
Appendix B

Redesigned test section

One of the first requirements to carry on the present study of shock boundary layer interaction in unsteady transonic flows was to build a facility that would, first, allow such an investigation, and secondly, still be adaptable for future experimental research on the subject. As a result, an already existing facility was taken apart and a brand new test section was built as modular as possible to suit the present investigation, and allow future developments.

This subsection of the experimental model description intends to describe the redesign objectives and modular features, now available in the new test section. A detailed sketch of the new test section is presented in Figure B.3.

B.1 Design specifications

At the early status of the project, the Chair of Heat and Power Technology was only equipped with a supersonic wind tunnel test section fitting into the test section location. The rather small settling chamber of 250mm x 250mm is equipped with a flow straightener and turbulence grids. A first contraction in the horizontal plane guides the air flow from a 250mm high and 100mm wide inlet into a flexible de Laval nozzle on the upper and lower wall (Figure B.1). The height of the throat can be adjusted using a piston to accelerate the flow from low subsonic up to $M_{inlet} = 2.75$ in the 100mm x 106mm test object section. The flat side walls are equipped with 388x163mm optical glass windows at the test object section. They provide excellent conditions for optical measurement techniques as Schlieren technique, Particle Image Velocimetry (PIV), Laser-Two-Focus Anemometry (L2F) and Laser-Doppler-Anemometry (LDA). Figure B.1 shows the old test section with the demounted front side wall. The flexible metal sheets on the upper and lower walls extend far into the test object section. They are not parallel and open in a sharp step to the channel exit height of 120mm.

Since the disturbing end of these sheets could not be changed, a completely new inlet contraction and test object section was designed to replace the flexible de Laval nozzle. However, the settling chamber, the first horizontal contraction, and the diffusor were kept in the design of the new test facility since they were decoupled from the flexible de Laval
APPENDIX B. REDESIGNED TEST SECTION

Figure B.1: Already existing supersonic wind tunnel

nozzle. Finally, in order to fulfil the requirements of the test objects and the geometric boundary conditions the following design specifications were formulated:

① Up- and downstream fitting dimensions to inlet and outlet of the existing de Laval test section.

② Upper and lower wall curvatures are identical.

③ Existing side walls with the windows can be used, i.e. test object location remains where it is.

④ The boundary layer can grow over a distance of about 250mm between maximum contraction and test object.

⑤ Homogeneous, parallel and steady flow into the test object section for $P_{in} = 170\text{kPa}$ and $M_{inlet} = 0.6 - 0.8$, no boundary layer separation between the throat and the test section.

⑥ Optional boundary layer cut off on the lower wall in front of the test objects and controlled blow out to the atmosphere.

⑦ Possible insert of different test objects on both upper and lower walls.

⑧ Easy change of configuration through the side windows.

⑨ Easy and large external access to the test object for pressure tubes and wires.

⑩ Good optical accesses through upper test section wall for optical measurements or flow visualizations.

The design specifications ① to ⑤ deal with the design of a new inlet contraction. Specifications ① to ③ were translated into: the original channel width (100mm), inlet height (250mm) and outlet height (120mm) are kept. The new channel height over the entire empty test object section was increased to 120mm. Specifications ④ and partly ⑤:
B.1. DESIGN SPECIFICATIONS

slightly diffusing upper and lower walls between maximum contraction and test object compensated the growth of the boundary layer displacement thickness. The resulting opening angle has to be connected smoothly to the contraction curvature. The boundary layer displacement thickness was calculated with a boundary layer code (Klingmann, 1997) to be about 1.8mm, the boundary layer thickness about 14.5mm at the inlet to the test object section.

To obtain the contraction curvature, fulfilling specification 5, two different approaches were compared:

• The first one was extremely simple: For the contraction curvature a fourth-order polynomial function was assumed. This required six geometric boundary conditions, which were:
  
  – axial position and channel height at inlet (1st) and at maximum contraction (2nd),
  – no discontinuity in the first and second derivative at the inlet (3rd + 4th),
  – while the first and second derivative at the maximum contraction were given by the opening angle of the diffusing walls (5th + 6th).

After solving the set of equations the criterion of no separation in the contraction and diffusing section was checked with the boundary layer code.

Figure B.2: Overall drawing of the newly designed VM100 facility

• The second approach was more "classical": Morel (1977) described a guideline for the design of two-dimensional wind tunnel contractions. Using inviscid flow analysis design charts were developed for a one-parameter family of wall shapes, based on two cubic arcs. The parameters were the maximum wall pressure coefficients at the inlet (as an indicator of the danger of separation at the inlet) and at the exit (related to the exit velocity non-uniformity and separation). Simple working forms of the separation criteria by Stratford (1959) were used.
Both approaches resulted in very similar contraction curvatures and the geometry of the second approach was finally chosen.

The specifications 6 to 9 required a highly flexible design for the section where the test objects and the optional boundary layer cut off are located. In order to insert different test objects through the side windows on both upper and lower walls, and also have an easy access for instrumentation, the solution of mounting geometrically minimized test objects on the upper and lower frames was chosen. Any test object with a rectangular base would therefore fit in a support part included in the frames as a whole. The dimensions were chosen to be 290mm long, 100 wide and 25mm thick to be able to exchange the test objects easily through the side windows. Large opening through the model support parts were created to easily access and instrument the test objects from the outside. Besides, an excellent optical access from above can be obtained, for PIV or oil surface visualizations for instance, by inserting a small Plexiglas window into the upper support part. It should also be noted that the model support parts, placed on both upper and lower walls, were aligned to mirror each others and make the channel symmetrical. Finally, pressure tapings and probe inserts were located upstream and downstream the test section in order to measure the exact flow conditions at the inlet and outlet of the test objects. Of course, all connections between surfaces were carefully sealed with O’rings to prevent any leakage flow to the atmosphere. Moreover, all wind tunnel surfaces and connections between the different parts were manufactured with a very smooth finish to avoid BL tripping.
B.2 Boundary layer cut off device

In order to cut the boundary layer off to the atmosphere (requirement 6), the lower frame of the wind tunnel was designed with an opening just upstream of the test section location (Figure B.3). The design was made in such way that the model support part on the lower wall was isolated from the other parts so that it could easily be lifted up together with the test object. A cutting part, so called “nose” could thereafter be inserted upstream of the support part, cut and redirect a part of the mass flow out to the atmosphere. Two different noses are available at the moment: a sharp one and a round one as illustrated in Figure B.4.

Two different situations can thereafter be encountered: either the pressure in the channel is above, or under atmospheric conditions. In the first case, the part of the incoming flow that is cut off by the nose will naturally go out to the atmosphere, whereas a pump is needed in the second case. Either case, the delicate task is to control the outgoing mass flow accurately enough to properly cut off the incoming streamlines. Indeed, only streamlines located below the nose shall be cut off, whereas the others, above the nose, shall remain parallel. A too high or too low mass flow would result in bend streamlines in the free stream over the nose part, that is to say a pressure gradient in the free stream as shown in Figure B.5.

The control of the outgoing mass flow is therefore ensured by an accurate throttle valve manually adjusted. In order to monitor whether the incoming streamlines are aligned with the nose, pressure taps were located on the leading edge of the “noses” the way as for aerodynamic probes (see Figure B.6). The setup procedure thereafter simply consists of manually adjusting the position of the throttle valve until the pressure values on both sides of the nose are equal to each others.

Assuming that the edge of the nose is located exactly at the height of the boundary layer, the latter will be entirely blown out to the atmosphere. A new boundary layer will
therefore develop itself from the leading edge of the nose, providing interesting test cases for transition or Shock Boundary Layer Interaction studies.
Appendix C

Settling chamber instrumentation

Stagnation conditions, i.e. pressure and temperature, should be measured in the settling chamber in which the flow velocity should ideally be zero. However, it appeared that the static pressure measured in the settling chamber with a simple pressure tapping did not exactly match the total pressure value, revealing the existence of a dynamic pressure component. As a result, it was decided to use a total pressure probe and a total temperature probe to measure the stagnation conditions.

C.1 Total pressure probe

Probe design

The total pressure probe is a standard L-probe designed with an opening angle of $30^\circ$ in order to increase the sensitivity angle. A drawing of the total pressure probe is presented in Figure C.1.

![Figure C.1: Total pressure probe located in settling chamber](image)

The probe was inserted 30mm deep into the settling chamber to locate the probe’s head in the free stream without disturbing too much the flow field. Besides, the influence of the probe’s wake was checked out by partially and fully inserting the Pitot probe in the settling chamber and measuring the total pressure further downstream in the test section with another total pressure probe. The results showed no difference on the Pitot probe reading whether the first probe was partially or fully inserted in the settling chamber.
APPENDIX C. SETTLING CHAMBER INSTRUMENTATION

This constatation can easily be explained by the fact that first the velocity component within the settling chamber is rather small which obviously limits the velocity deficit in the wake, and secondly the presence of two convergents, as they accelerated the flow; also lower the magnitude of any flow disturbances.

Probe calibration

The total pressure probe was calibrated towards its sensitivity angle to check the influence of a possible flow angle within the settling chamber. The probe was therefore placed in the free stream of the test section and rotated from $-45^\circ$ to $+45^\circ$ towards the main flow direction. The results, presented in Figure C.2, showed that the total pressure probe has a sensitivity angle of approximately $\pm 25^\circ$.

![Sensitivity angle of the total pressure probe](image)

Figure C.2: Total pressure probe sensitivity towards incoming flow angle

Data acquisition

Total and static pressure measurements in the settling chamber were performed using the PSI 8400 system described in the measuring technique section. The probe and pressure tapping were simply connected to the scanners through a 2m long plastic pipe.

Total pressure measurement accuracy

The uncertainty of total and static pressure measurements in the settling chamber is exactly the same as for other pressure measurements and is described in details in the measuring technique section. It should, besides, be noted that the 2m long plastic pipes have a damping effect on the pressure fluctuations i.e the standard deviation value, and theoretically introduce a time lag in the pressure readings. Whereas this "side effects" do not disturb steady state measurements, one should be aware of their presence and
possible effects on the measurements.

C.2 Total temperature probe

Steady state total temperature measurements were performed in the settling chamber (Figure 5.11) using a special high accuracy thermocouple. However, as a non negligible velocity component was found in the settling chamber (see Appendix on "Air flow quality"), the thermocouple was inserted in a Pitot probe in order to "re-create" stagnation conditions around the "hot" junction.

Thermocouple-Pitot probe design

The Pitot probe simply consists of a metallic pipe with inner and outer diameters respectively equal to 2 and 4mm, which was bent to face the incoming flow. An opening angle was machined on the leading edge in order to increase the sensitivity angle of the probe. A special high accuracy exposed miniature T-type thermocouple was previously inserted into the pipe and glued with respect not to touch the pipe as illustrated in Figure C.3.

![Figure C.3: Total temperature probe](image)

Temperature measurement

A thermocouple in its simplest form is a pair of dissimilar material, Copper and Constantan (a Copper-Nickel alloy) for a T-type thermocouple, that are soldered together at one end. The operation principle is the so called "Seebeck" effect by which a voltage is generated when the junction of two dissimilar metals are heated to different temperatures as illustrated in Figure C.4. Knowing the temperature of the "cold junction", also called the reference temperature, the type of metal used and the voltage generated between the two junctions, one can calculate the "hot junction" temperature. However, in the present experiments, the reference temperature was not the usual ice melting temperature, but the room temperature. Besides thermocouples are highly non-linear. Both the compensation and linearization were undertaken by the acquisition module presented below.
APPENDIX C. SETTLING CHAMBER INSTRUMENTATION

Data acquisition system

The data acquisition system used for temperature measurements was an IMP (Isolated Measurement Pod) 35951C "PC-logger" connected to a 35954B IBM-PC interface, from Solartron Inc.¹. This AAC (Analog to ASCII Converter) features signal conditioning, a 16 bit ADC, communication to the host PC, and a built-in sensor energization, all housed in an environmental protected case and built according to ISO 9001 standards. A microprocessor located within the IMP responds to commands received from the host and controls the measurement setup, as well as the data acquisition and communication to the host. Measurements are stored within the IMP until requested by the host. The input ranges were set, adjusted and calibrated at factory for each type of thermocouple. The preset ranges are then automatically set when selecting the type of thermocouple within the software.

The thermoelectric output voltage is measured from the thermocouple and then adjusted to account for the cold junction temperature according to the type specified. A solid state sensor is located within the IMP and automatically samples the ambient temperature every time a measurement is taken. The adjusted voltage is thereafter used as input variable into a linearizing table, which was constructed (one for each thermocouple type) so that its contribution to the measurement error is negligible.

Data acquisition method

The data acquisition was performed using a serial communication program developed for the purpose of these measurements. Temperatures were sampled at 1Hz and directly saved to disk during any other measurements in order to monitor any temperature drift or fluctuation of the cooler efficiency. It was for instance noticed that, depending on the mass flow and the atmospheric temperature, a minimum of 30-40 minutes was necessary to reach quasi-steady flow conditions due to the iterating cooler adjustments (see section on "Warm up time and Transient regime").

Total temperature measurement accuracy

¹The Roxboro Group plc, Byron House, Cambridge Business Park, Cambridge, CB4 4WZ, UK
C.2. TOTAL TEMPERATURE PROBE

Following the measurement chain, the following components will introduce their relative source of error, which will be accounted with the respective accuracy of each individual component as listed below:

- Accuracy special T-type thermocouple (given by manufacturer): ±0.5°C
- Thermocouple cable related error: neglected
- Accuracy A/D conversion: ±1bit = 0.08°C
- Thermocouple temperature range: −100°C to 200°C
- Resolution AD card: 16bit
- "Cold junction" compensation (given by manufacturer): ±0.5°C
- T/C linearisation (given by manufacturer): 0.1°C

As a result the overall temperature measurement accuracy can be calculated as follow:

\[ U = \pm \sqrt{0.5^2 + 0.08^2 + 0.5^2 + 0.1^2} = \pm 0.71°C \] (C.1)
Appendix D

Air flow quality investigation

Once the new test facility had been designed and manufactured, an experimental investigation of the air flow quality was performed with the aim to characterize the flow field in the test section. This investigation mainly addresses the following issues. First, the acoustic signature of the wind tunnel had to be characterized in order to validate the use of this facility for SBLI studies. With the aim at establishing the possible sources of noise among the different components of the wind tunnel, both the amplitude level and frequency content of the acoustic noise, inherent to the facility, had to be determined. Secondly, a comparison between the imposed back pressure perturbations and the noise level had to be performed in order to determine a possible interaction while "artificially" perturbing the shock. Finally, the repeatability and reproducibility of the measurements have to be either confirmed in order to proceed with further measurements.

D.1 Flow quality in settling chamber

As previously mentioned, the difference between the static and total pressure measured with a Pitot probe in the settling chamber was found non negligible. The velocity component corresponding to the dynamic pressure was therefore estimated using equation D.1 (assuming the flow incompressible in the settling chamber).

\[ P_t = P_s + \frac{1}{2} \rho U^2 \quad \Rightarrow \quad U = \left( \frac{P_{\text{probe}} - P_s}{P_s} \right)^{\frac{1}{2}} \]  

(D.1)

Both static and total pressures in the settling chamber were measured over the full range of the foreseen operating conditions, that is to say for the inlet total pressure value around 160kPa and the outlet static pressure value varying, from 130kPa down to 96kPa for the 3D nozzle experiments, and from 118kPa down to 98kPa for the 2D nozzle. The velocity components within the settling chamber was thereafter calculated for all operating points and are presented as function of the outlet static pressure in Figure D.1.

As a result, the flow velocity component in the settling chamber was found to increase as the outlet static pressure decreases until the nozzle is choked (approximately at \( P_{out} \approx 108kPa \)). For lower outlet static pressure values, the density increases but the
velocity component within the settling chamber remains constant as the flow if choked in the test section.

D.2 Flow quality in test section

Introduction

During the preliminary investigations of the new VM100 wind tunnel, high amplitude pressure fluctuations were observed in the test section. The origin and influence that those fluctuations might have on the shock location and imposed motion had thus to be investigated. Consequently, a measuring campaign including both steady state and high frequency pressure measurements over a wide range of operating conditions was conducted to enhance the knowledge about the air flow quality. A full description of the measurement procedure, data acquisition and data reduction can be found in the corresponding report (Bron, 2001).

Investigation procedure

With the aim to characterize the pressure fluctuations observed in the wind tunnel, high frequency and low speed pressure measurements were performed, both in the settling chamber and in the test section. The sampling rates were respectively set to 100kHz and 10Hz. The sampling time was approximately 53 seconds and a 50kHz low pass filter was used to avoid aliasing during post treatment.

In order to determine the origin, and possibly the source of emission of the pressure fluctuations, the measurements were performed at many different operating points by slowly varying one of the control valves (inlet, by-pass, and outlet valve). Besides, in order to avoid any interaction of the pressure perturbations with a shock wave, this investigation was performed without any test object inserted in the test section.
Data reduction

The amplitude level and frequency composition of the pressure fluctuations were determined at two locations in the wind tunnel facility. Peaks appearing in the frequency domain for both of the recorded signals were then isolated by filtering the signals, and a cross correlation algorithm was finally used to determine the direction of propagation of certain pressure waves, at different operating points.

A relative comparison between imposed perturbations (using the perturbation generator) and the observed pressure fluctuations was performed in order to quantify the possible influence of the "acoustic" perturbations on the shock motion.

Achieved results

For the operating points where all measurement are foreseen ($M_{\text{inlet}} > 0.6$), the following observations could be drawn:

- Time-dependent pressure signal always shows large amplitude fluctuations, such as seen in Figure D.2, when the cam is fixed either in vertical or horizontal position.

![Time dependent pressure signal in test section](Figure D.2: Time dependent pressure signal in wind tunnel test section with non rotating cam)

- However, the computed FFT could show that the previously observed fluctuations are mainly composed by low frequencies (under 200Hz), and even very low frequencies (between 0.1 and 2 Hz) as illustrated in Figure D.3. Besides, the amplitude\(^1\) of such perturbations is limited to 450Pa, which represents less than 0.4% of the static pressure level.

- The origin of such low frequency perturbations seems to related to other components of the wind tunnel facility, like for instance the fan whose passing blade frequency seems to correspond with the peak around 25Hz seen in Figure D.3.

\(^1\)In FFT algorithm, amplitude of FFT depends on number of points used to make FFT. The more points taken to compute the FFT, the more the energy content is spread over the frequency domain, the lower is the resulting amplitudes of FFT. Here, amplitude of FFT was normalized (integrated over 1Hz, which gives high amplitudes)
Figure D.3: FFT of fluctuating pressure in wind tunnel test section with non rotating cam (normalized)

• A high frequency component (around 25,000Hz) was found to be related to acoustic emissions due to the presence of the cam, either in its vertical or horizontal position. Indeed, the observed peak disappear as soon as the cam is removed, whereas the low frequency components remains still.

• A cross correlation analysis illustrated in Figure D.4 revealed that some of those pressure fluctuations seem to have their direction of propagation correlated to the position of the control valves. As a result, some of the observed pressure travelling waves are believed to originate unsteady flows induced by the control valves, and especially by the bypass valve.

Figure D.4: Illustration of a cross correlation on two filtered pressure signals from the settling chamber and the test section

• Furthermore, when the rod is rotating, the time-dependent pressure signal shows a rather undisturbed periodic signal at twice the rotating frequency of the cam, as seen in Figure D.5.
D.2. FLOW QUALITY IN TEST SECTION

Figure D.5: Time dependent pressure signal in wind tunnel test section with rotating cam at 37Hz (2220rpm)

- The computed FFT of the forced excited pressure signal (Figure D.6) clearly shows harmonics of the rotating cam frequency. Besides, the random pressure fluctuations in the VM100 wind tunnel appear to be of a much lower order of magnitude than the imposed perturbations.

Figure D.6: FFT of fluctuating pressure in wind tunnel test section with rotating cam at 37Hz (2220rpm)

- The harmonic composition of the pressure signal while perturbing the back pressure seems to be related to the aerodynamical force acting on the rotating cam, that is to say, on the flow velocity at the outlet of the test section.

Achieved conclusions

- The observed high amplitude time dependent pressure signal seems to correspond to a sum of many travelling waves, mostly not in phase. Indeed, although each frequency contribution might not be important, the resulting time dependent signal can have a higher amplitude if the perturbations have a random phase.
• In all configurations, the pressure fluctuations are mainly composed by low frequencies (under 200Hz), or even very low frequencies (between 0.1 and 2 Hz). Their amplitude is limited to 450 Pa and they are believed to originate unsteady flows induced by the different components of the wind tunnel facility. Therefore, those pressure fluctuations can be defined as the acoustic signature of the wind tunnel.

• The frequency composition of the inherent pressure perturbations seems rather to depend on the positions of the different valves than on the inlet Mach number or Reynolds number.

• The difference between the imposed perturbations and the inherent pressure fluctuations is always noticeable in a frequency analysis.

• The amplitude of the imposed perturbations become noticeable as soon as the inlet Mach number is approximately above 0.5. The imposed perturbations indeed propagate upstream with the relative velocity $|c - U|$, the difference between the sound velocity and the flow velocity. As a result, when the Mach number is low, the high relative velocity of the perturbations combined with the damping effect through the boundary layer seems to affect the pressure fluctuation directly on the wall.

### D.3 Measurements repeatability and reproducibility

Introduction and definitions

As part of the air flow quality, it is important to evaluate to which extent the overall experimental apparatus is able to reproduce the same results for the same operating conditions.

According to the international vocabulary of the basic and general terms in metrology defined by the International Organization for Standardization (VIM, 1993), the Measurements repeatability and reproducibility correspond to the exact following definitions:

• **Repeatability:** Closeness of the agreement between the results of successive measurements of the same measurand carried out under the same conditions of measurements (VIM, 1993).

• **Reproducibility:** Closeness of the agreement between the results of successive measurements of the same measurand carried out under different conditions of measurements (VIM, 1993).

It is important to understand the meaning and limit of each term since it is rare, in experiments, to be able to exactly reproduce the very same operating conditions. In our case, for instance, an operating point is determined by setting up the inlet stagnation pressure and temperature, as well as the outlet static pressure. As described in the "Experimental apparatus" section, those measurements are themselves function of the different valves which basically control the pressure level and mass flow within the test section. Since the adjustment of such valves is rather coarse, it is really difficult, time consuming and almost impossible to reach the desired values. Moreover, each of the above
measurements addresses its own accuracy and it is thus impossible to know with exactitude the true value. As a result, it is virtually impossible to pretend reaching the very same operating point. Therefore, what we will call here the measurement repeatability will also include a part of reproducibility.

Investigation procedure and data reduction

The measurand chosen to determine the repeatability and reproducibility was directly the shock position in the channel obtained by Schlieren visualization over the 2D nozzle. The mean shock position seems indeed highly sensitive and dependent on the operating conditions, especially the outlet static pressure.

The investigation procedure therefore consists of reaching a steady flow regime (see next paragraph on "Start up and transient regime"), setting up the same operating point \((Pt \sim 160kPa, \, Tt \sim 303K, \, Ps_{\text{out}} \sim 110kPa)\), and recording the Schlieren pictures, as described in the "Schlieren visualization" section, over a certain time to capture any low frequency "steady state" fluctuation. The "repeatability" measurements have been performed for a week, at different times of the day, using conventional Schlieren and the steady state pressure measurement system to determine the exact operating conditions. Table D.1 presents the flow properties for each measurement (see respective sections on measuring techniques for further details).

<table>
<thead>
<tr>
<th>Name</th>
<th>(Pt_{\text{inlet}}[kPa])</th>
<th>(Tt_{\text{inlet}}[K])</th>
<th>(M_{\text{inlet}}[-])</th>
<th>(Ps_{\text{outlet}}[kPa])</th>
<th>(Q[kg/s])</th>
</tr>
</thead>
<tbody>
<tr>
<td>Repeat 01</td>
<td>160.185</td>
<td>304.6</td>
<td>0.682</td>
<td>110.260</td>
<td>3.695</td>
</tr>
<tr>
<td>Repeat 02</td>
<td>159.873</td>
<td>305.4</td>
<td>0.682</td>
<td>110.055</td>
<td>3.701</td>
</tr>
<tr>
<td>Repeat 03</td>
<td>159.910</td>
<td>304.7</td>
<td>0.682</td>
<td>109.959</td>
<td>3.705</td>
</tr>
<tr>
<td>Repeat 04</td>
<td>159.859</td>
<td>304.9</td>
<td>0.682</td>
<td>110.110</td>
<td>3.693</td>
</tr>
<tr>
<td>Repeat 05</td>
<td>160.260</td>
<td>304.6</td>
<td>0.683</td>
<td>110.086</td>
<td>3.709</td>
</tr>
<tr>
<td>Repeat 06</td>
<td>160.017</td>
<td>303.7</td>
<td>0.683</td>
<td>109.932</td>
<td>3.712</td>
</tr>
</tbody>
</table>

Table D.1: Operating conditions during repeatability measurements

Practically, the recording frequency of the high speed CCD video camera was set to its lowest value, 50Hz, while the acquisition card’s memory set the maximum number of recordable pictures (exactly 2048 at this frame rate) and thus the total recording time was about 40 seconds. The pictures were thereafter post-treated as described in the "Schlieren visualization" section and a mean shock position was determined through the height of the channel, for each measurements serie. The results were thereafter plotted on the same Schlieren picture to give a good overview of the repeatability.

Achieved results

The results are presented in Figure D.7 with their respective operating conditions. The exact value of the mean shock position at the height \(y=30mm\) have been reported, for each measurement serie, in Table D.2 to obtain a precise comparison and analysis.
It is noteworthy that the mean shock positions are not exactly at the same location over the bump. For instance, at y=30mm in the channel (see arrow in Figure D.7) the most upstream and downstream mean shock positions are respectively 62.23mm and 65.53mm, thus 3.3mm distant from each others.

Now, before we can draw any conclusion, it is important to keep in mind that the Schlieren system accuracy is at least \( \pm 0.3 \text{mm} \) (see experimental techniques for further details) due to the CCD camera resolution and without considering the additional error when determining the scale factor and reference positions during the post treatment process.

Coming back to the above observation, the slight difference in the mean shock position can easily be explained by the differences in the operating conditions obtained during each separate measurement. As mentioned previously, it is rather difficult (if not impossible)
to reach the exact same operating conditions due to a coarse adjustment of the control valves. We have, here, a perfect example of the shock sensitivity to the determinant parameters, which are the stagnation pressure and temperature and most important, the outlet static pressure. The most downstream mean shock position ("Repeat 06") corresponds, for instance, to the lowest outlet static pressure value. On the other hand, the highest outlet static pressure value, which is reached in "Repeat 01", does not correspond to the most upstream mean shock position ("Repeat 02"). The two other parameters also have an important influence. Thus, comparatively, the stagnation pressure and temperature are respectively lower and higher in "Repeat 02" than in "Repeat 01", which contributes in setting the mean shock position slightly upstream. It is noteworthy that the relation between the operating conditions and the mean shock position is not equally dependent on the three independent control variables: $P_{int}$, $T_{int}$ and $P_{out}$.

It is also noteworthy that the standard deviation calculated from the time serie of Schlieren visualization is about ±1mm for all experiments, which means that the shock can be found, with a 95% probability, within ±2mm with respect to the mean shock position for any of the performed experiments. This result basically indicates a good trend towards experimental repeatability since it shows a similitude in how the shock responds to inherent unsteadiness in the wind tunnel.

Achieved conclusions

- Considering both the experimental technique’s accuracy and the slight differences between each operating conditions reached during the experiments, one can say that, globally, the measurements have a fairly good repeatability.

- On the other hand, the reproducibility has not been found so good, mainly due to the high sensitivity of the shock towards the operating parameters, namely the stagnation pressure, stagnation temperature, and outlet static pressure. This result, if not really surprising, informs us on how sensitive the shock is towards small variations of the operating conditions.

- Finally, the sensitivity of the shock position towards the operating conditions is confirmed not equally distributed between each separate operating parameters. The outlet static pressure, for instance, has a much greater influence on the shock position than the stagnation pressure or temperature.

D.4 Start up and transient regime of the compressor

Introduction

The determination of the start up time and transient regime addresses the estimation of the period of time after which the main air flow properties are considered stable enough to assume the flow as "steady state" or quasi-steady. Indeed, the different components
of the overall facility require some time to reach stable working conditions or to adapt to a change in the flow conditions (mass flow, pressure level or temperature change). For instance, the screw compressor needs a certain time to warm up and reach its steady state operating conditions. Similarly, the cooling system adjusts the quantity of water necessary to cool down the air mass flow in the test section. Coupled to a N-type thermocouple, the cooling system samples the temperature of the air flow every 5 minutes and then adjusts the valve controlling the cooling water mass flow in order to reach the desired temperature.

Investigation procedure and data reduction

Basically, the investigation procedure and data reduction both follow the steady state pressure measurements described in the “measuring techniques” section. The inlet stagnation pressure and temperature as well as the outlet static pressure were sampled during a certain time in order to determine the period of time after which the operating conditions can be considered as stable.

Achieved results

The inlet total pressure, inlet total temperature and outlet static pressure have been plotted in Figure D.8 versus time. The origin of the abscissa corresponds to the last valve adjustment.

![Transient regime after compressor start-up](image)

Figure D.8: Transient regime of the

Clearly, the pressure stabilizes within 10 to 15 minutes whereas the temperature requires approximately 30 to 40 minutes to reach a stable level. Some fluctuations can be observed on the temperature curve. They correspond to the water mass flow regulation for the cooling system. Indeed, air temperature measurements are continuously updated every 5 minutes and the cooling water mass flow is automatically adjusted to reach the selected air temperature.
Appendix E

Principle of mass flow measurements

E.1 Working principle

The principle of the mass flow measurement method is based on the installation of a primary device into the pipeline in which the flow is running full. This primary device is, in our case, an orifice plate (Figure E.1), that is to say a simple metal plate in which a circular aperture has been machined. The presence of such device causes a static pressure difference between its upstream and downstream sides. The rate of flow can then be determined from the measured value of this pressure difference and from the knowledge of the characteristics of the flowing fluid as well as the circumstances under which the device is being used. It is assumed that the device is geometrically similar to the one on which the calibration has been carried out and that the conditions of use are the same, i.e. that it is in accordance with the international standard (ISO-5167-1, 1998). It was especially checked that the general requirements for measurements as well as all installation requirements were properly fulfilled.

![Figure E.1: Drawing of the orifice plate device](image)

The size of the aperture hole and the pipe diameter equal respectively $d=148.9\text{mm}$ and $D=301.7\text{mm}$. The differential pressure measurement between the upstream and downstream side of the device is performed using a circular slot to which two standard pressure
E.2 Determination of mass flow

The mass flow rate can be determined, since it is related to a differential pressure within the uncertainty limits stated by ISO 5167, by the following formulae:

\[ q_m = C \epsilon_1 \frac{\pi}{4} d^2 \sqrt{\frac{2\Delta P}{\rho_1}} \sqrt{1 - \beta^4} \]  
(E.1)

where:

- \( C \) is the discharge coefficient, determined by the Reader-Harris/Gallagher equation:

\[
C = 0.5961 + 0.0261\beta^2 - 0.216\beta^8 \\
+ 0.000521 \left( \frac{10^6 \beta}{Re_D} \right)^{0.7} + (0.0188 + 0.0063A)\beta^{3.5} \left( \frac{10^6}{Re_D} \right)^{0.3} \\
+ (0.043 + 0.08e^{-10L} - 0.123e^{-7L}) (1 - 0.11A) \beta^4 \\
- 0.031(M - 0.8M^{1.1})\beta^{1.3} \]  
(E.2)

- \( \epsilon_1 \) is the expansibility factor:

\[
\epsilon_1 = 1 - (0.41 + 0.35\beta^4) \frac{\Delta P}{\kappa P_1} 
\]  
(E.3)

- \( \beta \) is the diameter ratio: \( \frac{d}{D} \)
- \( Re_D \) is Reynolds number based on D: \( Re_D = \frac{U_1 D}{\mu_1} = \frac{q_m}{\pi \mu_1 D} \)
- \( M = \frac{2L}{1 - \beta} \)
- \( A = \left( \frac{19000}{Re_D} \right)^{0.8} \)
- \( L = \frac{0.0254}{D} \)

As seen in equation E.2, the discharge coefficient \( C \) is dependent on \( Re_D \) which itself is dependent on \( q_m \). In such case, the final value of \( C \), hence of \( q_m \), has to be obtained by iteration. From an initial guess, the formulae gives an estimation of the flow rate which can be input back as a new initial estimation, etc. The procedure has to be repeated until the difference between the initial and calculated values is below a certain precision, typically, the uncertainty of the mass flow measurement.
E.3 Uncertainty of flow rate measurement

The uncertainty on the flow rate measurement is defined here as a range of the values within which the true value of the measurement is estimated to lie at the 95% probability level. It is, in practice, equivalent to twice of the standard deviation used in statistical terminology and it is obtained by combining the partial uncertainties on the individual quantities which are used in the calculation of the flow-rate, assuming them to be small, numerous and independent from each other.

In fact, the various quantities which appear on the right-hand side of the mass rate flow formulae (equation E.1) are not independent, so that it is not correct to compute the uncertainty of \( q_m \) directly from the uncertainties of these quantities. However, it is sufficient, for most practical purposes (see ISO 5167), to assume that the uncertainties of \( C, \epsilon_1, d, \Delta p, \text{and} \rho_1 \) are independent of each other.

A practical working formulae for \( \delta q_m \) may then be derived from equation E.1 which accounts for the interdependence of \( C \) and \( \epsilon_1 \) on \( d, D, \text{or} \beta \). It shall be noted that \( C \) may also be dependent on \( Re_D \), however the deviations of \( C \) due to these influences are of a second order and are included in the uncertainty on \( C \).

From the derivation of equation E.1, it comes:

\[
\frac{\delta q_m}{q_m} = \left[ \left( \frac{\delta C}{C} \right)^2 + \left( \frac{\delta \epsilon_1}{\epsilon_1} \right)^2 + \left( \frac{2 \beta^4}{1 - \beta^2} \right)^2 \left( \frac{\delta D}{D} \right)^2 + \left( \frac{2}{1 - \beta^2} \right)^2 \left( \frac{\delta d}{d} \right)^2 + \frac{1}{4} \left( \frac{\delta \Delta p}{\Delta p} \right)^2 + \frac{1}{4} \left( \frac{\delta \rho_1}{\rho_1} \right)^2 \right]^{\frac{1}{2}}
\]

where:

\[ \delta C = \begin{cases} 
0.5\% & \text{if } \beta \leq 0.6 \\
(1.667\beta - 0.5)\% & \text{if } 0.6 < \beta \leq 0.75 
\end{cases} \]

\[ \delta \epsilon_1 = 4 \frac{\Delta p}{P_i} \]

\[ \delta D = \delta d = 0.1 \text{mm} \]

\[ \delta \rho = \rho_1 \left( \frac{\delta P}{P_i} - \frac{\delta T}{T_i} \right) \]

A distinction should be made between the uncertainty linked to the measurements made by the user and those linked to quantities specified by ISO-5167-1 (1998). The latter uncertainties are on the discharge coefficient \( C \) and the expansibility factor \( \epsilon_1 \). They give the minimum uncertainty with which the measurement is unavoidably tainted, since the user has no control over these values. They occur because of small variations in the geometry of the device are allowed, and because the investigations on which the values have been based could not be made under "ideal" conditions, nor without some uncertainty. Practically, the uncertainty on \( C \) and \( \epsilon_1 \) are given by the ISO5167 whereas the others have to be determined by the user. The details on the uncertainties used in the formula for \( \delta P, \delta T, \delta \rho \) can therefore be found in the respective sections corresponding to the different measuring techniques.
APPENDIX E. PRINCIPLE OF MASS FLOW MEASUREMENTS

The uncertainty on mass flow measurement can finally be estimated using the following constants:

\[
\begin{align*}
D &= 0.3017 \text{ m} & \text{Pipe diameter} \\
d &= 0.14887 \text{ m} & \text{Orifice plate diameter} \\
N_{u1} &= 18.71 \times 10^{-6} & \text{Dynamic viscosity on the upstream face of the plate} \\
\kappa &= 1.4 & \text{Heat constant ratio} \\
\epsilon_p &= 10^{-6} \text{ kg/s} & \text{Desired precision for the mass flow calculation} \\
rgas &= 287 \text{ J/kg/K} & \text{Perfect gas constant} \\
\end{align*}
\]

and uncertainties:

\[
\begin{align*}
\delta P_1 &= 42 \text{ Pa} & \text{Accuracy on pressure measurement} \\
\delta \Delta P &= 42 \text{ Pa} & \text{Accuracy on pressure loss measurement} \\
\delta D &= 10^{-4} \text{ m} & \text{Accuracy on pipe diameter measurement} \\
\delta d &= 10^{-4} \text{ m} & \text{Accuracy on orifice diameter measurement} \\
\delta T_s &= 0.7 \text{ C} & \text{Accuracy on temperature measurement} \\
\end{align*}
\]

Finally, the uncertainty on mass flow measurements was calculated to be:

\[
\delta Q_m = \pm 0.032 \text{ kg/s}.
\]
Appendix F

Static calibration of fast response transducers

The static calibration of the fast response transducers addresses several issues. First of all, one might want to check the linearity of the transducer, that is to say if the bridge output voltage is a linear function of the pressure applied onto the membrane. Secondly, the two coefficients which determine the transfer function between the transducer’s voltage and the pressure, have to be evaluated over the full pressure range. Finally, the last but not least of the check-up would be to determine whether or not there is a time drift in the transducer’s transfer function. This might not have a huge importance for a qualitative study, but the exact coefficients have to be established for a quantitative analysis of the pressure amplification and comparison with other similar transducers.

The experimental procedure of the static calibration simply consisted of measuring the output voltage of each transducers for different pressure values, over the full pressure range of the transducers. Practically, each transducer was inserted into a small chamber in which the pressure could be set by a manual pump and directly monitored using a reference transducer with a 0.025% full scale accuracy, located in a portable pressure calibrator. Consequently, the output voltage of each transducer was recorded using the unsteady pressure measurement system, for different pressure levels form -80kPa up to 100kPa relative to the atmosphere. The pressure was simultaneously controlled using the ±25Pa accuracy transducer and the ±11.5Pa accuracy digital atmospheric barometer, which together provided a ±27.5Pa accuracy pressure reading. On the other hand, the accuracy of the fast response transducers also introduced an error in the output measurement voltage, which was estimated to ±0.1mV based on the accuracy of the unsteady pressure measurement system.

The data reduction thereafter consists of establishing the agreement between the measured data and the linear model assumed as a transfer function for the transducers. This is achieved by determining the best-fit parameters yielding a minimum in the merit function which is basically the difference between the model and the set of measured data. However, an important issue that goes beyond the mere finding of best-fit parameters lies in the fact that data are not exact but subject to measurement errors. We therefore need to assess the goodness-of-fit against some statistical standards, or, in other words, the likely errors of the best-fit parameters since obviously the measurement errors in the
data must introduce some uncertainty in the determination of the sensitivity and offset coefficients. The data reduction method is basically a linear regression based on the minimum assessment of the Chi-square merit function (Press et al., 1992).

Concretely, let us consider the problem of fitting a set of N data points \((P_i, V_i)\) to a straight line model:

\[
P(V) = P(V; a, b) = a + bV
\]

(F.1)

If experimental data are subject to measurement error both on \(P\) and \(V\), as it is in our case (see “Accuracy estimation on measuring techniques”), the Chi-square merit function will be written:

\[
\chi^2(a, b) = \sum_{i=1}^{N} \frac{(P_i - a - bV_i)^2}{\sigma_{P_i}^2 + b^2\sigma_{V_i}^2}
\]

(F.2)

where \(\sigma_{P_i}\) and \(\sigma_{V_i}\) are respectively the \(P_i\) and \(V_i\) standard deviations on the \(i\)th point.

The weighted sum of variances in the denominator of equation F.2 can be understood both as the variance in the direction of the smallest \(\chi^2\) between each data point and the line with slope \(b\), and also as the variance of the linear combination \(P_i - a - bV_i\) of two random variables \(V_i\) and \(P_i\). Therefore,

\[
Var(P_i - a - bV_i) = Var(P_i) + b^2Var(V_i) = \sigma_{P_i}^2 + b^2\sigma_{V_i}^2 \equiv \frac{1}{\omega_i}
\]

(F.3)

We want to minimize equation F.2 with respect to \(a\) and \(b\). Unfortunately, the occurrence of \(b\) in the denominator makes the resulting equation for the slope \(\frac{\partial\chi^2}{\partial b} = 0\) non-linear. However, the corresponding condition for the intercept, \(\frac{\partial\chi^2}{\partial a} = 0\), is still linear and yields:

\[
a = \frac{\sum_i \omega_i (P_i - bV_i)}{\sum_i \omega_i}
\]

(F.4)

The resolution strategy therefore consists of minimizing a general one-dimensional function with respect to \(b\), while using equation F.2 at each stage to ensure that the minimum with respect to \(b\) is also minimized with respect to \(a\). No further details will be developed here, but can be found in Press et al. (1992).

The algorithm used to solve the above non-linear system not only provides the sensitivity and offset coefficients, but also the likely errors of the best-fit parameters, in other words, the standard deviation of both coefficients. As an illustration, Figure F.1 presents the linear curve that best-fitted the set of experimental data, providing, in this specific example, a sensitivity of 49.11mV/bar and an offset of 2.75mV with the respective error of \(\pm 0.010\%\) and \(\pm 0.24\%\).

In parallel of this static calibration performed for each transducer, a repeatability check was conducted on one specific transducer by reproducing the very same measurement procedure every other day during a few weeks. The results showed a difference which, at its maximum, was about 0.06mV/bar for the sensitivity and 0.54mV for the offset, which represents an error of approximately \(\pm 0.051\%\) in the sensitivity determination.
As a result, a time drift in the transfer function of the fast response transducers does exist, but it should, however, be noted that the effect on the sensitivity lies within the combined value evaluated by the manufacturer and representing the non-linearity, hysteresis, and repeatability behavior of each transducer.

Consequently, although the effect of the time drift always lies within the accuracy of the transducers, in order to lower the systematic error contribution, a three point calibration should be performed on each transducer prior and after each unsteady measurement.

Finally, the main conclusions that can be drawn out from the static calibration procedure have been summarized below:

- The linearity of the fast response transducers was measured within the specified (or even better) value given by the manufacturer.
- The sensitivity and offset coefficients of the linear transfer function was calculated for each transducer and stored in a database for easy access and future post treatment.
- An estimation of the time drift was evaluated for one transducer and it was found within the repeatability specification. A six point calibration was however suggested prior and after each unsteady measurements in order to limit the systematic error contribution.
Appendix G

Sound propagation in closed pipes

G.1 The organ pipe model
In this example, the pressure tap and capillarity tube are assimilated to a simple closed end pipe of length \( L \), exposed to a sound wave with frequency \( f \), as sketched in Figure G.1. For a given frequency, the wavelength of the travelling wave depends on the speed of sound in air \( \lambda = \frac{c}{f} \). A few conditions are then required in order to obtain standing waves in the closed pipe: basically, the closed and open ends of the pipe should correspond respectively to a node and a maximum. Consequently, only certain combinations of wavelength and length of the pipe will result in a standing wave or resonance. The conditions are given by:

\[
\lambda_n = \frac{4L}{n} \quad \text{for } n=1, 3, 5, 7...\text{etc} \quad (G.1)
\]

\[
f_n = \frac{c}{\lambda_n} = \frac{nc}{4L} \quad \text{for } n=1, 3, 5, 7...\text{etc} \quad (G.2)
\]

\[
f_n = nf_0 \quad \text{with} \quad f_0 = \frac{c}{4L} \quad (G.3)
\]

The frequencies \( f_n \) are called the resonance frequencies of the pipe and should be avoided as sampling frequency.

According to the instrumentation of the test objects, the longest capillarity tubes on the 3D and 2D bumps are respectively equal to 24.4mm and 11.5mm, which have a resonance frequency of 3.8kHz and 7.4kHz respectively. Although those values seem to be low, they actually suit the objectives of the present work since our main interest concerns the few first harmonics while the fundamental corresponds to the imposed perturbation frequency. For instance, a sampling frequency of 4kHz with a low pass filter at 2kHz would still allow the calculation of the first twenty harmonics if the imposed perturbation frequency is 100Hz.

G.2 The Helmotz resonator model
In this second example, the effect of having a volume of gas inside the capillarity tubes is accounted by making an analogy with the Helmoltz oscillator, which basically consists
APPENDIX G. SOUND PROPAGATION IN CLOSED PIPES

Figure G.1: Working principle of an organ pipe resonator

of a container of gas (usually air) with an open hole (or neck) as sketched in Figure G.2. Considering a lump of air at the neck of the container (shaded in the diagram), the pressure fluctuations above it force this lump of air a little way down the neck, thereby compressing the air inside the container. Subsequently, the pressure drives the lump of air back to its original position, but its momentum pushes it outside the neck over a small distance, which now rarefies the air inside the container and results in sucking back in the lump of air, and so on. The lump of air can thus vibrate adlibidum just like a mass on a spring (see Figure G.2).

Figure G.2: Working principle of an Helmoltz resonator

In analogy with the mass-spring oscillating system, the eigen frequency of the Helmoltz resonator is given by:

\[ \omega = c \sqrt{\frac{S}{VL}} = 2\pi f \]  \hspace{1cm} \text{(G.4)}

where \( S \) is the sectional area of the neck, \( V \) the volume of the container, and \( L \) the length of the neck.
Appendix H

Elements of statistical analysis

Statistical analysis remains the best way to quantify and characterize the behavior of scattered experimental data. Assuming any steady state pressure measurement as an ergodic process, that is to say a random process in which the statistical properties from a single time-serie will approach definite limits independent of the particular series as the length of the series increases, we can evaluated the first central moments without calculating the respective distribution probability\(^1\).

Central moments

First of all, the \(n\)\(^{th}\) central moment of an univariate probability function \(P(x)\) is defined, according to Papoulis (1984), as:

\[
\mu_n = \langle (x - \langle x \rangle)^n \rangle = \int_{-\infty}^{\infty} (x - \mu_1)^n P(x) dx
\]  \(\text{(H.1)}\)

Arithmetic mean

There are several statistical quantities called means, but the quantity referred to as "the" mean is the arithmetic mean, also called the average of a population and denoted \(\mu_1\), \(\bar{x}\), \(<x>\), or \(A(x)\), which is given by:

\[
\mu = \int_{-\infty}^{\infty} x P(x) dx \quad \overset{\text{ergodic process}}{\iff} \quad \bar{x} = \frac{1}{N} \sum_{i=1}^{N} x_i
\]  \(\text{(H.2)}\)

Standard deviation

The standard deviation \(\sigma\), also called the root-mean-square, is defined as the square root of the variance \(\sigma^2\), which is the second central moment \(\mu_2\):

\[\sigma = \sqrt{\mu_2 - \mu_1^2}\]

---

\(^1\) This is actually not true for every series produced by an ergodic process, but the set for which it is false has probability zero. Thus, for stationary ergodic processes it is possible to estimate the process statistics (mean, variance, autocorrelation function...etc) from the observed values of a single time serie
APPENDIX H. ELEMENTS OF STATISTICAL ANALYSIS

\[ \sigma^2 = \mu_2 = \int_{-\infty}^{\infty} x^2 P(x) dx \quad \text{ergodic process} \quad \sigma_x^2 = \frac{1}{N} \sum_{i=1}^{N} (x_i - \bar{x})^2 \] (H.3)

Confidence interval

The probability that a measurement will fall within a given closed interval \([a, b]\). For a continuous distribution,

\[ CI(a, b) = \int_{a}^{b} P(x) dx \] (H.4)

where \(P(x)\) is the probability distribution function. Usually, the confidence interval of interest is symmetrically placed around the mean, so

\[ CI(x) \equiv C(\mu - x, \mu + x) = \int_{\mu-x}^{\mu+x} P(x) dx \] (H.5)

where \(\mu\) is the mean. For a normal distribution, the probability that a measurement falls within \(n\sigma\) of the mean \(\mu\) is

\[ CI(n\sigma) \equiv \frac{1}{\sigma \sqrt{2\pi}} \int_{\mu-n\sigma}^{\mu+n\sigma} \exp \left( \frac{-(x - \mu)^2}{2\sigma^2} \right) dx \] (H.6)

\[ = \frac{2}{\sigma \sqrt{2\pi}} \int_{\mu}^{\mu+n\sigma} \exp \left( \frac{-(x - \mu)^2}{2\sigma^2} \right) dx = erf \left( \frac{n}{\sqrt{2}} \right) \] (H.7)

<table>
<thead>
<tr>
<th>range</th>
<th>CI</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\sigma)</td>
<td>0.6826895</td>
</tr>
<tr>
<td>(2\sigma)</td>
<td>0.9544997</td>
</tr>
<tr>
<td>(3\sigma)</td>
<td>0.9973002</td>
</tr>
<tr>
<td>(4\sigma)</td>
<td>0.9999366</td>
</tr>
<tr>
<td>(5\sigma)</td>
<td>0.9999994</td>
</tr>
</tbody>
</table>

Table H.1: Confidence interval for a normal distribution

As a result, and providing that the results are statistically independent, i.e. normally distributed, one can say that 95% of the data will fall into the interval \([-2\sigma, +2\sigma]\).

Skewness

The skewness, as defined by Kenney and Keeping (1962), is a normalized form of the 3\(^{rd}\) central moment and characterize the degree of asymmetry of a distribution:

\[ \gamma_1 = \frac{\mu_3}{\mu_2^{3/2}} = \int_{-\infty}^{\infty} x^3 P(x) dx \quad \text{ergodic process} \quad S = \frac{1}{N\sigma^3} \sum_{i=1}^{N} (x_i - \bar{x})^3 \] (H.8)
Flatness

The flatness, also called "excess" or kurtosis, as defined by Kenney and Keeping (1962), is a normalized form of the 4th central moment and characterizes the degree of peakedness of a distribution:

\[
\gamma_2 = \frac{\mu_4}{\mu_2^2} - 3 = \int_{-\infty}^{\infty} x^4 P(x) dx - 3 
\]

\[
F = \frac{1}{N \sigma^4} \sum_{i=1}^{N} (x_i - \bar{x})^4 - 3 \quad (H.9)
\]
Appendix I

Steady state pressure measurement system accuracy analysis

In order to comprehend the uncertainty analysis of the PSI system, it is important to understand the design of the electronic pressure scanning system under discussion. A full detailed description can be found in the "8400 User's Manual" (PSI, 1993).

Electronic pressure scanning system description

The electronic pressure scanning system itself typically consists of three components: The main SP already presented above, the PCUs also mentioned earlier, and the electronic pressure scanners (ESP), which basically correspond to the pressure scanners inside of a single PSU.

More precisely, the ESP scanners consist of an array of silicon pressure transducers (one per measurement port), internal multiplexers and a single instrumentation amplifier. Through electronic addressing, the transducer analog outputs are passed through the internal amplifier to provide a high level low impedance output signal to the Analog to Digital (A/D) converter in the SP. In addition, the scanner incorporates an integral calibration valve through which a series of calibration pressures can be applied to all transducers simultaneously. This feature in tandem with the PCU allows the scanners to be frequently recalibrated against a secondary transfer standard thereby substantially (if not entirely) eliminating environmental effects and long term instability of the analog circuitry.

The microprocessor-based PCU consists of pneumatic solenoids to activate the scanner calibration valve, a servo-valve to regulate recalibration pressures and a secondary pressure transfer standard to accurately measure these calibration pressures. The PCU generates five discrete pressures based on preprogrammed information ranging from negative full scale to positive full scale of the scanner range under calibration. At each pressure, the voltage response of each scanner transducer and the frequency output of the resonant quartz transfer standard are recorded. At the completion of the calibration process, a linear regression is conducted to generate a fourth-order polynomial equation for each transducer of the form:
APPENDIX I. STEADY STATE PRESSURE MEASUREMENT SYSTEM
ACCURACY ANALYSIS

\[ P_x = C_0 + C_1 V_x + C_2 (V_x)^2 + C_3 (V_x)^3 + C_4 (V_x)^4 \]  
(I.1)

Where:  
\( P_x \) = unknown pressure  
\( C_2 \) = first linearity  
\( V_x \) = measured voltage  
\( C_3 \) = second linearity  
\( C_0 \) = offset  
\( C_4 \) = third linearity  
\( C_1 \) = sensitivity

These equations are used by the SP to calculate engineering units during subsequent data acquisition prior to transfer to the host computer. The SP houses a microprocessor-based unit, the SDU, designed to electrically interface the ESP scanners to the SP. The latter incorporates a 16 bit A/D converter as well as the circuitry necessary to address and provide power to the scanners. The SDU transfers raw data to the SP where a 32 bit microprocessor converts the data into IEEE floating point engineering units using the equations generated from the previous calibration.

Effects of on-line calibration

The uncertainty analysis of this system is unique in that the on-line calibration function eliminates many of the classical errors associated with pressure measurement. In addition to compensating for transducer characteristics, this system calibration compensates for errors in the signal conditioning and excitation circuitry, the analog cable and the A/D converter. Environmental effects on both the transducer and the cables as well as any long term drift of the A/D and transducer excitation can be characterized and negated by performing periodic on-line calibrations against an accurate transfer standard. Of primary importance in achieving specified system accuracy, is the accuracy of the transfer standard and the short term stability of the analog circuitry including the A/D converter. For the purpose of this analysis, it is assumed that the secondary transfer standard has been recently calibrated against a primary transfer standard of sufficient accuracy (Marks, 2002). In addition, the analog circuitry, including transducer excitation and the A/D converter, of the 8400 system has been proven to have excellent short term stability. These factors limit uncertainty to precision errors of repeatability and resolution and bias errors associated with the transfer standard and curve fit. The following uncertainty analysis of the system, based on Juanarena and Ridenhour (1987), is presented with this in mind as well as the assumption that an online calibration operation has just been completed.

Uncertainty analysis

According to ISO (1993), the expression used to calculate measurement uncertainty is:

\[ U = \pm (B_t + T_{95}S_t) \]  
(I.2)

Where:  
\( B_t \) is the Root-Sum-Square (R.S.S.) of bias errors  
\( S_t \) is the R.S.S. of precision errors  
\( T_{95} \) is assumed to be 2.

The precision and bias errors are derived from two sources; the transfer standard measurement and the transducer measurements (including signal conditioning). The resultant
The root-sum-square equations for $S_t$ and $B_t$ are:

\[
S_t = S_s^2 + S_v^2 \quad \text{and} \quad B_t = B_t^2 + B_v^2 \quad (I.3)
\]

Where:
- $S_s$ is the precision error, transfer standard
- $B_s$ is the bias error, transfer standard
- $S_v$ is the precision error, transducer
- $B_v$ is the bias error, transducer

The following is a breakdown of the precision and bias errors for these two sources.

### A. Transfer Standard Measurement

#### I. Transfer Standard & Signal Processing Errors

<table>
<thead>
<tr>
<th>AMOUNT</th>
<th>TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S_1$</td>
<td>$\pm 0.005% F.S.$</td>
</tr>
<tr>
<td>$S_2$</td>
<td>$\pm 0.005% F.S.$</td>
</tr>
<tr>
<td>$S_3$</td>
<td>$\pm 0.0005% F.S.$</td>
</tr>
<tr>
<td>$B_1$</td>
<td>$\pm 0.01% F.S.$</td>
</tr>
<tr>
<td>$B_2$</td>
<td>$\pm 0.005% F.S.$</td>
</tr>
<tr>
<td>$B_3$</td>
<td>$\pm 0.005% F.S.$</td>
</tr>
</tbody>
</table>

#### II. Data Reduction Errors

- $S_4 = 2^{-64} = 0.0$ (Floating Point Resolution)
- $B_4 = \pm 0.005\% F.S.$ (Curve Fit Error)

### B. Transducer Measurement

#### I. Transducer & Signal Processing Errors

<table>
<thead>
<tr>
<th>AMOUNT</th>
<th>TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S_5$</td>
<td>$\pm 0.01% F.S.$</td>
</tr>
<tr>
<td>$S_6$</td>
<td>$\pm 0.005% F.S.$</td>
</tr>
<tr>
<td>$S_7$</td>
<td>$\pm 0.003% F.S.$</td>
</tr>
</tbody>
</table>

#### II. Data Reduction Errors

- $S_8 = 2^{-64} = 0.0$ (Floating Point Resolution)
- $B_5 = \pm 0.005\% F.S.$ (Curve Fit Error)

Table I.1: Breakdown of the bias and precision errors of the PSI 8400 system

The root-sum-square equations are then used to calculate the precision and bias errors to be used in the uncertainty formula as follows:

\[
S_t = \pm \sqrt{S_1^2 + S_2^2 + S_3^2 + S_4^2 + S_5^2 + S_6^2 + S_7^2} = 0.014\% \quad (I.4)
\]

\[
B_t = \pm \sqrt{B_1^2 + B_2^2 + B_3^2 + B_4^2 + B_5^2} = 0.014\% \quad (I.5)
\]

The uncertainty can now be calculated as:
APPENDIX I. STEADY STATE PRESSURE MEASUREMENT SYSTEM
ACCURACY ANALYSIS

\[ U = \pm (B_t + 2S_t) \]
\[ = \pm (0.014 + 0.028) \]
\[ = \pm 0.042\%F.S. \] (I.7)
Appendix J

Dynamic calibration of capillarity tubes

J.1 Introduction and objectives

Basically, a calibration is the establishment of a known relation or transfer function (TF) between the input or driving function and the output or response function. For a simple static calibration (as the one addressed in the "unsteady flow instrumentation" section), the TF is the ratio of output to input, that is to say the relationship (usually linear) between the transducer's output signal and the measurand. For a dynamic calibration however, the TF is a complex function of frequency in which are included certain time constants. This function may be found in two ways: if the system can be described by a characteristic differential equation, the TF may be obtained by analytical solution of the equation. But if the characteristic differential equation is not known or impracticable to describe, the TF must be obtained by a knowledge of a pair of associated input and output functions which must be measured experimentally.

In case of an experimental calibration method, the imposed input signal may either be a non periodic or a periodic function. The non periodic function may be an impact of a short duration which is quickly released, like in step function generators, quick opening devices, explosive devices, or shock tubes. In alternative, an ideal periodic function generator should produce a pure sinusoidal wave with controlled frequency and amplitude like acoustical shock generator, rotating valves, sirens, piston in cylinder and mechanical oscillators.

In the present case, due to the space constrains and practical reasons, the instrumentation holes on both 2D and 3D bumps were foreseen such that any Kulite transducers could be mounted at any location underneath the test objects as seen in Figure 5.14. Each transducer was thus located at a different distance from the surface depending on the local thickness of the test object. As a result, the pressure waves travelling within the channel and propagating in those capillarity tubes are reaching the transducers with different time delays and amplitude attenuations, depending on the measurement locations.

This subsection is concerned with the estimation of the transfer function of the pressure fluctuations through each capillarity tube. First, a quick overview of the different
APPENDIX J. DYNAMIC CALIBRATION OF CAPILLARITY TUBES

calibration devices and methodologies used nowadays are presented, followed by a
description of the device and analysis methodology chosen for this task. Finally, the data
acquisition procedure and data reduction is presented, as well as the method used to ac-
count for the damping and phase-lag of the capillarity tubes during the unsteady pressure
measurements.

J.2 A non-exhaustive overview on dynamic calibration procedure

Theoretical modelisation of capillarity tubes

The response of pressure measuring systems to a sinusoidal input has been considered
by several authors. In the early days, Taback (1949) presented a methodology based on
the analogy between the propagation of sinusoidal disturbances in electrical transmission
lines and pressure measuring systems. To predict the response of such system, he used a
propagation velocity calculated from the Rayleigh formula combined with measured values
of the attenuation. The fact that those damping constants had to be determined experi-
mentally was, at that time, the main disadvantage of his method. Then, Iberall (1950)
presented a derived formulae from the fundamental flow equations (continuity, momentum
and energy) but no comparison with experimental values was performed. Later on, Davis
(1958) presented asymptotic forms of the dynamic response formulae that could be used
as basic guide lines for the selection of pressure measuring systems. Bergh (1964) then
started developing a technique to better understand the pressure propagation through
thin circular tubes with connected volumes. The very next year, proceeding along the
same lines as Iberall, Bergh and Tijdeman (1965) derived a recursion formula for pressure
disturbances propagating through a serie of N pipes and N volumes as illustrated in Figure
J.1(a). Theoretical results for single pressure measurement system (N=1) were presented
that demonstrate the influence of different parameters such as pipe length and diameter,
chamber’s volume... Moreover, calculations on a double pressure measurement system
(N=2) were presented as an illustrative example of a system with a discontinuity in pipe
diameter (Figure J.1(b)). Comparison with experimental results showed that the response
characteristics of a pressure measurement system could be theoretically predicted with
a high degree of accuracy. Considering the yet current relevance of this work whenever
a comparison with experimental data is needed, the main points of the discussion over
theoretical results are presented below:

For single measurement systems (N=1) (illustrated in Figure J.1(a)):

- In most cases, resonance peaks do occur due to standing waves at the system’s
  resonance frequency.
- For a given length, a wider tube produces higher resonance peaks at higher resonance
  frequency.
- For a given diameter, a longer tube produces smaller resonance peaks at lower
  resonance frequency.
J.2. A NON-EXHAUSTIVE OVERVIEW ON DYNAMIC CALIBRATION PROCEDURE

(a) Basic layout of N measuring systems connection serie

(b) Particular cases of pressure measuring systems

Figure J.1: Illustration of theoretical model for dynamic system characteristic determination (Bergh and Tijdeman, 1965)

- Furthermore, certain combinations of the ratio Length/Diameter show no resonance peak at all.

- Surprisingly, a larger volume chamber produces lower resonance peaks, but lowers the resonance frequency.

- In all cases, the transducer's diaphragm deformation has little influence.

- Similarly, the constant for the pressure expansion in the transducer volume (range from polytropic to isentropic transformation) has little influence.

- The mean pressure value has a strong influence on the dynamic response of the system (higher peaks at higher resonance frequency as the pressure increases).

- A contrary, mean temperature changes (within a certain limit) have a negligible influence on the dynamic characteristics of the system.

For double measuring pressure system (pipe diameter discontinuity illustrated in Figure J.1(b)):

- A wider second tube does not always have an unfavourable effect on the dynamic characteristics. On the contrary, a smaller second tube does not have a favourable effect either.

- Basically, for a system consisting a smaller tube followed by a wider one, the latter tube is more easy to overcome for a pressure disturbance due to relative smaller effect of wall friction. On the other hand, the wider tube acts like a kind of additional volume, thus reducing the output of the first tube. This two opposite effects are responsible for the final behaviour of the total system. The same effects but working in the opposite sense are responsible for the system of a wider tube followed by a smaller one.

- Changing the length ratio between the two tubes will also change the ratio of the mentioned effects and thus give other dynamic characteristics.
• Globally however, a smaller tube at the entrance usually results in lower amplitude ratio than in the case of a large tube at the entrance.

• By a proper selection of tube lengths and diameters, a response characteristic can be obtained which is most suitable for a given purpose. In practice however, the discontinuities in the tube offer the possibility of non-linearity.

A simpler formulation, derived from the single pressure measurement system described above, was introduced by Tijdeman (1975) to describe the propagation of small amplitude pressure fluctuations into a cylindrical pipe. Tijdeman introduced four dimensionless parameters: the shear wave number, reduced frequency, Prandtl number, and the specific heat ratio. The formulation was undertaken by Dibelius and Minten (1983) who explicitly expressed the damping and phase coefficients for low frequency perturbations and took into consideration the volume between the tube and the transducer. A parametric study yielded the same conclusive points as above. A good comparison between experimental and theoretical results were achieved in Bohn and Schnittfeld (1992) and Boerrigter and Charbonnier (1997). Even more recently, Boer (1988) considered boundary layer entrainment effects in the investigation of the characteristics of the transfer function over pneumatic lines.

Dynamic calibration devices

Theoretically, system identification methods require tests that excite all modes of the system. Since the 60’s the transfer functions of pressure transducers have been determined using various experimental pressure generators: periodic and non-periodic pressure functions. Schweppe et al. (1963) presented an exhaustive survey of dynamic calibration methods, describing not only the devices but also their respective methodologies. Analytical and experimental methods for both linear and non-linear transducers are presented in this work.

Basically, dynamic calibration can be divided into two classes, which differ in the obtained signals and therefore in the analysis performed. Both have advantages and inconveniences and usually concern different type of applications with different range of amplitude and frequency. The choice between one or the other will really depend on the requirements and constrains of the experiments.

On one hand, periodic function generators include various type of devices, from electrical and mechanical oscillators to rotating valves, sirens, or moving piston in cylinder. In their work, Bohn and Schnittfeld (1992) used an electric sine wave generators coupled to a driver to transform the signals into mechanical energy and amplified it using an exponential horn. The entire device was placed in an isolated chamber so that they could study the influence of mean pressure and temperature on the propagation of pressure waves into a cylindrical tube. Boerrigter and Charbonnier (1997) used an impinging air jet in a cavity connected to a cylindrical tube and used the sustained acoustic waves as driving function for dynamic calibrations. This device was however not practical since the frequency of the sustained wave was a function of the dimensions of the pipe. Most
contributions featuring periodic input functions actually use rotating devices. Körbächer and Bolcs (1996), for instance, used a rotating valves device similar to the one used by Weyer (1973) in his thesis work. A rotating disk was alternatively connecting high and low pressure chambers to an outgoing pipe. The fluctuating signal was thereafter directly applied on the pressure taps using a small hand held device. Closely, Boerrigter and Charbonnier (1997) used an impacting jet directly on a rotating wheel with holes to create a fluctuating pressure signal. Another illustration is the siren developed by Dibelius and Minten (1983) in which the periodic break of the air flow through a rotor wheel induced pressure fluctuations in a pipe.

On the other hand, non-periodic pressure functions like pressure steps are in practice advantageous compared to periodic functions because only one test of short duration is sufficient to cover the entire frequency domain of interest. Typical aperiodic function generators include white noise generators, dropping ball, quick opening and explosive devices. Quick acting valves rapidly appeared to be well suited to generate pressure steps for dynamic calibrations. Their simple working principle is based on the opening of a passage between two chambers, initially at two different pressures, creating a sudden pressure rise in one of the chamber. Pallant (1966) presented a note on low frequency calibrators using fast opening valves as well as a comparison with shock tube and static calibration methods. Already mentioned, Scheppe et al. (1963) also presented a review on aperiodic function generators. Among others, Pennelegion et al. (1966) successfully used fast opening valves to calibrate pressure transducers in short duration wind tunnel facilities but were limited to frequencies up to a few kHz. During a pneumatic probe compensation procedure, Paniagua and Denos (2000) used an exploding balloon as calibration device. The backward step function generated was sufficient to calibrate a five hole probe up to 700Hz. Another original method was used by Ciocan et al. (1998), who tested the response of an unsteady five-sensor probe in a water tank, where a shock wave was generated by the implosion of a vapor bubble produced with an electrical discharge in water.

During the last two decades however, the need for higher frequency measurements required the use of steeper step function generators. Shock tubes were then considered as calibration devices. In such apparatus, a high-pressure chamber is pressurized until a diaphragm bursts, a shock is then generated and propagates at the speed of sound along the low-pressure chamber. At the end wall the pressure rise is almost a perfect stepwise pressure rise, allowing dynamic calibrations easily up to 500kHz. Ainsworth and Allen (1990); Ainsworth et al. (2000) used a shock tube to determine the dynamic characteristics of surface mounted sensors up to 4MHz. Popp (1999) also used a shock tube to characterize the frequency response of screened and unscreened Kulite transducers and determine the influence of small cavities screened sensors. Other authors (Gossweiler, 1993; Paniagua and Denos, 2000) simply used shock tube to calibrate short capillarity tube and flush mounted fast response transducers. Most often, the experiments are compared with the analytical formulation from Tijdeman. The main drawback however in shock tube experiments is the practicability to establish the dynamic characteristics of capillarity tubes directly on the test objects, as well as the respective influence of mean pressure and temperature on the dynamic response. Indeed, Gossweiler (1993) presented a discussion on the physical properties and error sources of fast response sensors and mention the distinct influences of
APPENDIX J. DYNAMIC CALIBRATION OF CAPILLARITY TUBES

pressure and temperature transient response. Additionally, Boerrigter and Charbonnier (1997) assessed the respective parameters influence in the analytical formulation from Tijdeman (1975) and basically retrieved the conclusions achieved by Bergh and Tijdeman (1965). Furthermore, Bohn and Schnittfeld (1992) experimentally showed the distinct dynamic characteristics of pneumatic lines for two different pressure and temperature values.

Transfer Function identification methodologies

Nowadays, most corrections on pressure measuring systems are performed by assuming that the transfer function of both the capillarity tubes and the transducer are described by a second order differential equation system. A analytical formulation of the TF is then established based on experimental results. Ainsworth and Allen (1990) showed a good agreement between theory and measurements for flush mounted sensors in a shock tube. Furthermore, Rottmeier and Mullhaupt (2002) recently developed an unsteady pressure signal processing method using a transfer function modelling technique based on Laplace harmonic analysis. It is noteworthy that experiments are still necessary to confirm the analytical formulation of the dynamic system’s transfer function, including pneumatic lines or not. Most of the time, harmonic analysis such as Fourier and Laplace transforms (extensively described in Press et al. (1992) and Kamen and Heck (2000)) is used to process the signal and work in the frequency domain. It should however be noted that there actually exist alternative methods, so called parametric methods, to directly correct the signal in the time domain.

With conventional harmonic techniques, the transfer function is determined in the frequency domain as the ratio between the Fast Fourier Transform (FFT) of the output and the FFT of the input at each frequency. If the signals are sampled at $N = 2^n$ instants, the transfer function is a set of $\frac{N}{2}$ complex numbers. In contrast, a parametric method is performed in the time domain and only requires few parameters to be stored, but needs more computational resources as shown in Paniagua and Denos (2000). Besides, FFT algorithms usually require periodic excitations and the continuity of both the signal and its first derivatives, which is an important constraint in its use but can easily be processed by signal analysis. In the case of a rising step test for instance, the step signal can be converted into a periodic function by imposing the signal to return to its initial level before the test, e.g. through an inverted step or a sinusoidal decrease (Popp, 1999). Gossweiler (1993) solved the problem by performing derivatives of the signals before computing the FFT. Another concern is the Gibbs phenomenon, discussed extensively by Canuto et al. (1991), which is a characteristic oscillatory behaviour in the neighborhood of the step with an overshoot and alternating local minima and maxima. The overshoot tends towards the point of discontinuity as the number of retained frequencies is increased. Due to this unavoidable problem, the FFT ratio between the perfect step (input data) and the measurement chain output is not correct, which introduces "ghost" frequencies. Thus, the reconstructed signal can never be the "true" pressure. Parametric methods in the time domain avoid the previous two problems by their nature.
J.3 Dynamic calibration methodology

According to the above literature survey, the perfect input signal to the calibration fast response transducers seems to be a sharp step function, usually obtained in shock tubes, as it contains a large number of frequencies and thus provides a (more) complete frequency response. However, in the present case, a shock tube calibration seems completely unpracticable since the dynamic calibration of pneumatic lines is of interest and has thus to be performed directly on the test objects. Besides, the dynamic calibration above a few kilo Hertz does not present, here, a major interest as the capillarity tubes probably feature a low resonance frequency beyond which the calibration is unnecessary. Furthermore, the amplitude of perturbation for a given frequency is impossible to adjust, a priori, in a shock tube experiment since only the initial and final state of the gas can be changed. Concerning the post processing, dynamic calibrations using aperiodic input functions also present a few drawbacks as overshoots often appear and require special signal treatment. In case of Fourier or Laplace decomposition, the input step function also requires some preprocessing to satisfy basic assumptions in harmonic analysis. Finally, from a practical point of view, the present dynamic calibration method and apparatus should be applicable to other facilities, as part of a newly developed measuring technique at the Chair.

As a result, it was decided to apply a periodic fluctuating pressure on each pressure tap, directly on the surface of the instrumented test object and measure both the input and response signals as sketched in Figure J.2.

![Figure J.2: Overview of the dynamic calibration principle](image)

The dynamic calibration apparatus basically consists of a small device, so called calibration head, which contains a small chamber opened at one end and connected to a Kulite transducer on the other end, as shown in Figure J.3. Two holes were drilled on the side of the calibration head in order to introduce and evacuate the pressure fluctuations. One of them is connected to an external source of pressure fluctuation, whereas the second one
is simply connected to a long pipe which damps the fluctuations and avoids any reflected pressure waves back in the small chamber. The interface between the tap surface and the chamber’s opening is sealed with an O’ring to prevent any leakage. Besides, the design was made symmetrical so that the pressure fluctuations on the surface and recorded by the transducer could be assumed the same, which was thereafter experimentally confirmed. As a result, the fluctuating pressure within the chamber is measured simultaneously by the reference transducers in the calibration head, and a “test” transducer at the other end of the capillarity tube.

Figure J.3: Dynamic calibration unit - Calibration head

The fluctuating pressure generator was designed by Vogt (2001) and is similar to the device used by Boerrigter and Charbonnier (1997) and consists of a nozzle air jet impacting on a rotating wheel with holes as seen in Figure J.4. Different shapes of holes on the wheel are available and provide a slightly different pressure signal. The control of the air jet pressure and the rotating speed of the motor allows a fine adjustment over the amplitude and frequency of the pressure fluctuations. Besides, the casing can be connected either to a pressure reduction valve or to a pump in order to adjust as well the mean pressure value. However, this feature was not functional at the time of the dynamic calibration and the influence of the mean pressure value on the damping and time-lags should therefore be checked during later experiments. Finally, a TTL pulse signal is output from the pressure fluctuation generator in order to set a common reference for the pressure signals recorded from the reference and test transducers.

J.4 Practical considerations

This paragraph is dedicated to the introduction of a few practical considerations regarding the dynamic calibration. Although they are presented prior to the measurement methodology description, it should be noted that they were precisely obtained using the data acquisition and data reduction procedures introduced further down. Measurements were performed, dependencies were observed, and measuring procedures and methodologies were adapted to account for those influences.
J.4. PRACTICAL CONSIDERATIONS

Static calibration drift of Kulite transducers

As already noticed by Cherett and Bell (Cherrett, 1990; Bell, 1999) and presently, in the "Unsteady flow instrumentation" subsection, a drift over time of the offset and sensitivity values of the fast response transducers was observed during various unsteady pressure measurements as illustrated in Figure J.6(a). A static calibration was therefore performed during each unsteady measurements using a calibration unit, the DPI603 portable calibrator, which allows to setup and measure a certain pressure within a small cavity with an excellent accuracy ($\pm 0.025\% FS$). A small device was designed to directly connect all Kulite transducers to the calibration unit as shown in Figure J.5. The portable calibration unit was also connected to a $\pm 1kPa$ range external steady state pressure transducer in order to increase the accuracy to $\pm 25Pa$.

A six points static calibration was performed before and after each measurement in order to calculate the range of pressure variation and obtain a realistic estimation of the error relative to the drift of the fast response transducers. The acquisition procedure and data reduction method are similar to the one described in the instrumentation section. The results are plotted in Figure J.6(a) and illustrate the time drift of one Kulite transducer over a three days unsteady measurement campaign on the 2D sliding bump. The time elapsed between the prior and post-measurement static calibrations is approximately of eight hours. An averaged value for the offset and sensitivity was then used to calculate the corresponding pressure from the measured voltage signals.

As shown in Figure J.6(b), the pressure variation range resulting from the time-drift is function of the pressure level. At 100kPa for instance, the measurement error relative to such unstable behaviour of the fast response transducers would be approximately 390Pa. As previously observed in the unsteady instrumentation section, the above value, although quite high, falls within the error range evaluated by the manufacturer and representing a combined estimation for the non-linearity, hysteresis, and repeatability behaviour of the Kulite transducer.
APPENDIX J. DYNAMIC CALIBRATION OF CAPILLARITY TUBES

(a) Small chamber device used for simultaneous static calibration of all Kulite transducers

(b) Static calibration chamber connected to the portable DPI pressure calibrator

Figure J.5: Overall static calibration apparatus

(a) Sensitivity and offset drift illustration over a three days period

(b) Pressure variation range relative to a typical time-drift

Figure J.6: Static calibration drift of Kulite transducers

The present paragraph illustrates perfectly the benefit of a static calibration on the limitation of the systematic error related to unsteady pressure measurements. Although it is difficult to state in which proportion the error has been reduced, it is logical to assume that the unsteady pressure measurement accuracy is lower than the 0.141%FS (±240Pa for a 1.7bar Full Scale Kulite transducer) evaluated by the manufacturer.
J.4. PRACTICAL CONSIDERATIONS

Signal quality from the fluctuating pressure generator

As mentioned earlier, the pressure fluctuations are generated using a rotating wheel with regularly spaced holes, on which an air jet impacts. Although one could assume that the fluctuating pressure is close to a rectangular wave, the pressure signal measured by the reference transducer placed in the calibration head, on the bump, is actually mainly composed by the fundamental. A Fourier decomposition revealed the presence, in an extremely low proportion, of a few higher harmonics as seen in Figure J.7(b). Other frequencies, non harmonic of the perturbation frequency are totally non existent. Anyway, whatever the spectral composition of the signal is, all frequency components could be used to establish the transfer function of the respective capillarity tube. In the present case however, only the fundamental was used for the dynamic calibration as it contains most of the energy of the fluctuating signal.

![Figure J.7: Signal quality characteristics of the fluctuating pressure source used during dynamic calibration](image)

(a) Reference signal at 50Hz  (b) Reference signal at 500Hz

Parametric influence on dynamic calibration

A small parametric study was conducted prior to the overall dynamic calibration over the 2D and 3D test objects in order to determine the characteristics and limitations of such calibration. No need, for instance, to establish a transfer function of the capillarity tubes up to 10kHz if the resonance frequency occurs at 4kHz. Consequently, a dynamic calibration was performed over the shortest and longest capillarity tubes on the
APPENDIX J. DYNAMIC CALIBRATION OF CAPILLARITY TUBES

2D and 3D bumps to get a realistic idea on damping and time lag behaviour, determine the boundaries within which the dynamic responses of all other pneumatic lines can be expected, establish physical criteria in order to evaluate the resonance frequency or determine the frequency beyond which the damping becomes too high. During the procedure, several amplitudes and frequencies of pressure perturbation were tested. Furthermore, as the pressure fluctuation generator unit is limited to a maximum frequency of 4kHz, in the present preliminary investigation, the five first harmonics were used to estimate the damping and time lag at higher frequencies, up to 20kHz.

Practically, the acquisition procedure follows the methodology introduced in the previous paragraph and the data reduction method is similar to the one presented in the next paragraph. Briefly, a fluctuating pressure is applied on the surface of the test object and simultaneously recorded by a reference transducer placed on the pressure tap and a test transducer placed underneath the bump, on the other side of the pneumatic line to be calibrated. The damping and phase-lag are thereafter calculated and plotted as a function of the perturbation frequency.

The results presented in Figure J.8 illustrate the damping and phase-lag as functions of the perturbation frequency for the shortest and longest pressure taps on the 2D and 3D test objects. A similar behaviour to the one described by Ainsworth and Allen (1990) can be observed. Globally, the damping and phase-lag increase with the perturbation frequency depending on the length of the pneumatic line being calibrated. The longer the capillarity tube, the more important the damping and phase-lag as illustrated, for instance, in Figure J.8(a). However, close to the natural resonance frequency of the capillarity tube, an amplification of the propagating pressure within the tube and a 180° phase angle change can be observed. Typically, the resonance frequency occurs at 7kHz and 3.5kHz respectively for the shortest (2mm long) and longest (11.5mm long) capillarity tubes on the 2D bump, and around 3kHz and 2kHz respectively for the shortest (11mm long) and longest (24.5mm long) capillarity tubes on the 3D bump. It should be noted that results beyond the maximum frequency of the pressure fluctuation generator, which is 4kHz, were calculated using higher harmonics and thus have a much lower accuracy. The measurement were repeated a couple of times over a few days to ensure the repeatability of the results.

According to the above observations, it seems possible to model the damping and phase-lag behaviour as deterministic functions of the frequency. A set of criteria was defined to limit the modelling process to experimental data lying underneath the resonance frequency value. Those criteria were based on damping limitation (set to 75% of damping), damping behaviour (sudden amplification), and phase-lag shift. Finally, the damping and phase-lag were modelled respectively with a second order polynomial and an exponential functions of the frequency. The modelling process was performed using a robust least square fitting algorithm (see Numerical Recipes (Press et al., 1992)). Experimental values and fit model functions are shown in Figure J.9 for different jet pressures, i.e. different amplitudes of perturbation. In order to estimate the goodness of the fit, the residuals are plotted also in Figure J.9. As observed, the difference between the experimental values and the fitted model functions is always lower than 0.4% and 0.75° for the damping and phase-lag respectively. Furthermore, according to the results, the damping
J.4. PRACTICAL CONSIDERATIONS

(a) Extreme damping values on 2D bump

(b) Extreme damping values on 3D bump

(c) Extreme phase-lag values on 2D bump

(d) Extreme phase-lag values on 3D bump

Figure J.8: Influence of perturbation frequency on Damping and Phase-lag through shortest and longest capillarity tubes on 2D and 3D test objects and phase-lag values increase with both the frequency and the amplitude of perturbation. A difference of 10% and 10° is noted at 1600Hz respectively for the damping and phase-lag between the lowest and highest amplitude of perturbation. The influence of the amplitude of perturbation should therefore be accounted for in the correction for the dynamic behaviour of propagating waves in capillarity tubes.

Phase lag due to measurement equipment

Any experimental equipment, while acquiring, amplifying or filtering an analog signal, affects in some way the processed signal. A complete analysis of the electronic is usually required to estimate in which proportion the signal is distorted or delayed. A Transfer Function is then established as a function of the frequency by measuring the response of the electronic to dirac or a chirp function. In the present case, however, this procedure is unnecessary since all signals, including the TTL reference signal, are sampled at the same time by the same electronic, and should therefore have the same TF. Subsequently, the time delay due to the electronic is also similar for all channel and do not need to be evaluated since only the relative phase lags are of interest in the present evaluation.
APPENDIX J. DYNAMIC CALIBRATION OF CAPILLARITY TUBES

(a) Experimental damping values and respective fit model functions for different amplitude of perturbation

(b) Experimental phase-lag values and respective fit model functions for different amplitude of perturbation

Figure J.9: Influence of perturbation amplitude on Damping and Phase-lag through a 20mm long capillarity tubes on 3D bump

Although the respective time delay due to the electronic response can be assumed identical for all channels and therefore neglected, another time delay specific, this time, to the internal acquisition procedure should be accounted for. Indeed, the unsteady measurement system does not sample all channels in parallel but in a sequential way. The sampling period, also called frame, is divided into as many intervals as selected channels to acquire (plus two others used for synchronization purposes). The channels are individually and continuously sampled, one after the other, until the internal buffer is totally filled up. A second buffer then takes over the acquisition process while the first one transfers the data to the disk, and so on... As a result, no channel is actually sampled exactly at the same instant as any other and a slightly delay is introduced, depending on the sampling frequency and the number of channels. This time delay was directly accounted for in the post processing of the signal.

J.5 Dynamic calibration acquisition procedure

As mentioned previously, a six points static calibration was performed on all transducers before and after the unsteady measurements in order to check the influence of time shift on sensitivity and offset, and characterize the systematic error. Results are presented in the above paragraph on practical considerations, as well as in the "Unsteady flow instrumentation" subsection.

Using the methodology and instrumentation introduced in the paragraph on "chosen methodology" presented earlier, a fluctuating pressure is applied on the surface of the test object and simultaneously recorded by a reference transducer placed on the pressure tap and a test transducer placed underneath the bump, on the other side of the pneumatic line to be calibrated. This procedure was repeated for each pressure tap on both the 2D and 3D test objects, for different frequencies of perturbations (from 50Hz to 4kHz), and different jet pressures (10, 20, and 30kPa), i.e different amplitude of perturbations. In
J.6 Dynamic calibration data reduction

The dynamic calibration data reduction consists of two parts. The first task is to evaluate the damping and phase-lag for each capillarity tube as discrete functions of the perturbation frequency and amplitude. The main purpose of this part is to create a calibration database, which will be used to estimate the transfer function of any pressure tap, for any frequency and any amplitude of pressure perturbation.

Practically, output voltage signals from the reference and test transducers are read and converted into pressure signals using the static calibration coefficients obtained prior to the measurements. The TTL signal from the calibration unit recorded together with the transducer signals is then used as a reference to make an ensemble average (EA) of the data. The ensemble average procedure basically consists of collapsing together all unsteady cycles and make a statistical averaging based on the number of repetitions instead of the usual time-averaging. The obtained single unsteady cycle then represents an average of all unsteady cycles. All mathematical formulations are further described in the "unsteady measurements data reduction" subsection presented below. A Discrete Fourier Series Decomposition (DFSD) is thereafter performed on the ensemble averaged signal evaluated previously and the amplitude and phase angle of the fundamental are obtained. The damping and phase-lag are then calculated by computing respectively the pressure ratio and phase difference between the reference and test transducers. The time delay due to the sequential measurements of the KT8000 system is also accounted here. Finally, the data is saved in ASCII files as function of the perturbation frequency for later post processing, one file for each capillarity tube and each amplitude of perturbation.

The second part of the dynamic calibration data reduction is to fit a model function to the dynamic calibration experimental data and thereafter evaluate the exact damping and phase-lag values corresponding to measured amplitude of perturbation.

As described in the paragraph on "Practical considerations", damping and phase-lag values are modelled respectively with a second order polynomial and exponential functions of the frequency. Typical model function are illustrated for a 20mm long pressure tapping on the 3D bump in Figure J.9. The modelling process was performed using a robust least square fitting algorithm as described in Numerical Recipes (Press et al., 1992). The final phase of the data correction is performed directly while analyzing the unsteady pressure measurements and consists of an interpolation of the damping and phase-lag values depending on the measured amplitude of perturbation. As a result, the exact transfer function is calculated depending on the location of the pressure tap, the perturbation frequency, and the amplitude of the pressure fluctuations. The pressure signals are thereafter corrected and processed as described in the following section.
APPENDIX J. DYNAMIC CALIBRATION OF CAPILLARITY TUBES

J.7 Error and accuracy of dynamic calibration

The accuracy estimation of the dynamic calibration is not a simple task as many different measurements and data reduction procedures are involved. A description of the measurement chain as well as a breakdown of precision errors is presented in the "error and accuracy estimation" of the unsteady pressure measurements. However, a few points concerning the dynamic calibration and more especially the data reduction processes could not be quantitatively expressed and are presented below as further source of uncertainty:

- Evolution of the transducers accuracy when transforming a time signal into the frequency domain.
- The influence of mean pressure and temperature on the transfer function could not be checked at the present time, but should definitely investigated in the future work.
Appendix K

Conventional Schlieren flow visualization

In general three different methods of flow field visualization are available. Namely, the shadowgraph, interferometric and Schlieren methods. All of them are based on the fact that the index of refraction varies with the density of the flow field. Nevertheless, the information obtained by these methods are different.

The simplest to set up is the shadowgraph method, which detects the second derivative of the density field. Therefore, it is mainly applied when fields of very rapid change in density are investigated. The interferometric method is capable of measuring the density distribution of a flow field in a direct and quantitative manner. In a more complex setup, light is split into two beams. One passes the test section and is phase shifted whereas the other one simply bypasses the inhomogeneous density region. The two beams are thereafter recombined revealing a stripe pattern which contains qualitative information about the density distribution. The Schlieren method however is sensitive to the first derivative of the density gradient (Liepmann and Roshko, 1957) of the flow field. They appear in a variety of different configurations like color systems, focus or conventional Schlieren systems, and offer different cutoff mechanisms possibilities.

This section intends to describe the conventional Schlieren system used in the present work as well as the optical setup and the image post processing methods used both in steady state and unsteady visualizations. A qualitative description of the accuracy and errors related to this flow visualization method is also given in order to instruct the reader about the inaccuracies and limit of the method.

K.1 Principle of shock visualization method

The conventional Schlieren method is based, as introduced earlier, on the fact that the speed of light, and consequently the index of refraction, varies with the density of the medium it is passing through. From the basic laws of optics, a changing index of refraction has two effects on a light ray. First, it leads to a rotation of the wave fronts. Secondly, it introduces a phase shift between the different rays. The Schlieren method
APPENDIX K. CONVENTIONAL SCHLIEREN FLOW VISUALIZATION

takes advantage of the first property to visualize regions within the flow field with non uniform density distribution.

Basic details about Schlieren methods can be found in Meyer-Arhendt (1992) but a short description is given below. A schematic description of the system is shown in Figure K.1. On the left hand side of the test section is sketched the light source section. It consists of a continuous white light lamp, a set of lenses and an aperture hole. The main requirement of a Schlieren system is to provide parallel light beams through the test section. This is achieved by focusing the white light from a standard lamp onto an aperture hole using one or several converging lenses. In order to ensure that a parallel light beam passes through the test section, the aperture hole has simply to be placed at the focal plane of lens L2a.

![Figure K.1: Basic layout of a conventional Schlieren system](image)

On the right hand side of the test section is now described the image section, which consists of a converging lens L2b, similar to L2a, a knife edge, another converging lens and a screen or a CCD camera. The parallel light passing through the test section is focussed by the lens L2b onto a point located on the focal plane of L2b. Two images are then created. The first one is the image of the white light source and is located at the focal plane of lens L2b, whereas the second one is the image of the test section. Depending on the distance between the test section and lens L2b, the second image is virtual and located far upstream of lens L2b. Now, whereas both the deflected and non-deflected lights focus on the same focal plane, the deflected light actually focusses onto a point adjacent to the focal point of lens L2b. Therefore, by putting the knife edge at the focal plane, the deflected light can be intercepted, which results in having a dark region in the image of the test section. It should also be noted that the rays turn into the direction of increasing density. As a result, cutting the deflected light vertically or horizontally will better reveal vertical or horizontal gradients. The last converging lens L3 is then used to get a real image of the test section either on a screen or into the sensor area of a CCD camera.

K.2 Optical set-up

The Schlieren system used in the present work mainly consists of two equivalent optical appliances, each of them mounted on an adjustable three feet base. An appliance is basically made of a long empty cylinder which contains a large converging lens (with a
K.2. OPTICAL SET-UP

diameter of 185mm and a focal length of 1310mm) and a set of mirrors that turn the light into a section located at the top of the appliance (see Figure K.2). Along this section is mounted a metal girder on which are fixed other optical instruments like the lenses, the aperture hole, or the knife edge.

Figure K.2: Optical set-up of a conventional Schlieren system

Practically, a 150W Mercury arc lamp was used as the continuous white light source. Two converging lenses, both with a focal length of 150mm, were adjusted to focus as much light as possible onto the small aperture hole located at the exact focal plane of the first Schlieren lens L2. Whereas different interception methods are available, a simple razor blade in an oblique position was found sufficient to cut the deflected light and reveal both the shock wave and the thickening boundary layer.

Whereas the location of the first appliance does not matter, the position of the second one sets both the type and size of the image of the test section. In order to minimize the size of the final image, which will have to match the size of the CCD camera sensors, the second appliance was placed approximatively at 330mm from the center of the test section. The size and location of the virtual image created can then be calculated as follow:

\[
\frac{1}{d_{\text{img}}^2} + \frac{1}{d_{\text{obj}}^2} = \frac{1}{f_2} \Rightarrow \begin{cases} 
   d_{\text{img}}^2 &= f_2 d_{\text{obj}}^2 \\
   D_{\text{img}}^2 &= f_2 \left( D_{\text{obj}}^2 - f_2 \right) \approx -441.1 \text{mm} \\
   d_{\text{obj}}^2 &= (d_{\text{img}}^2 + f_2 + f_3) \\
   D_{\text{obj}}^2 &= D_{\text{img}}^2 \approx -160.4 \text{mm}
\end{cases}
\]  

where \( f_2 \) is the focal distance of lens L2
\( d_{\text{obj}}^2 \) (\( d_{\text{img}}^2 \)) is the distance between the lens L2 and the object (image) plane
\( D_{\text{obj}}^2 \) (\( D_{\text{img}}^2 \)) is the size of the object (image) picture

The last converging lens, with focal length \( f_3 = 30 \text{mm} \), basically provides a real image of the test section and decreases once again its size. As a result, the size and location of the real image of the test section were:

\[
\frac{1}{d_{\text{img}}^3} + \frac{1}{d_{\text{obj}}^3} = \frac{1}{f_3} \quad \text{with} \quad \begin{cases} 
   d_{\text{obj}}^3 &= (d_{\text{img}}^3 + f_2 + f_3) \\
   D_{\text{obj}}^3 &= D_{\text{img}}^3
\end{cases}
\]
\[ d_{img}^3 = f_3. d_{obj}^3 - f_3 = f_3 \left( \frac{d_{img}^2 + f_2 + f_3}{d_{img}^2 + f_2 + f_3} \right) \simeq 30.5 \text{mm} \]
\[ D_{img}^3 = f_3. D_{obj}^3 - f_3 = f_3 \left( \frac{D_{img}^2}{d_{img}^2 + f_2 + f_3} \right) \simeq 2.7 \text{mm} \]

where \( f_3 \) is the focal distance of lens L3
\( d_{obj}^3 \) (\( d_{img}^3 \)) is the distance between the lens L3 and the object (image) plane
\( D_{obj}^3 \) (\( D_{img}^3 \)) is the size of the object (image) picture

Whereas a simple screen was used to set up the system, a high speed CCD camera from Readlake\(^1\) was thereafter placed at the screen plane in order to digitally record pictures and store them on a computer for further post processing.

It should also be noted that an accurate adjustment of all different components in the optical set-up is crucial for the experimental outcome. These adjustments can somewhat be time consuming, but it is worthwhile to spend some extra hours on that issue. An excellent description of such optical set-up can be found in Johansson (2000).

### K.3 Image acquisition system

Both the steady and unsteady Schlieren visualizations were recorded using the MotionScope PCI 8000S CCD camera. This high speed digital imaging system can record a sequence of digital black and white images of an event at a speed of 60 to 8000 frames per second. The system stores these images in the memory of a controller unit, which basically consists of a PCI card installed on a standard host computer. These pictures can thereafter be viewed forward or reverse at selected frame rate, frame by frame, or freeze frame, to analyze motion and time during the event. The sequence can also be saved onto the disk for further image post processing. As the data transfer rate has an upper limit, the image resolution decreases as the the sampling rate increases. However, since the amount of memory in the controller unit remains the same, more pictures can be recorded at higher frame rate. Table K.1 gives the resolution, frame storage and record times for all available frame rates.

The system also provides an electronic shutter control of image exposure that allows the reduction of each frame exposure’s time in order to eliminate image blurring due to motion. Naturally, a shorter time exposure or a higher sampling frame rate both have to be compensated with a higher illumination. The available shutter speeds are 1x, 2x, 3x, 4x, 5x, 10x, 15x, and 20x, where x is the record record rate that divides the exposure time. For example, a 4X shutter speed at the rate of 500 frames/sec provides a exposure time of 1/2000 seconds.

The MotionScope PCI 8000S imaging system also features a few remarkable functions through external BNC cables directly connected to the control PCI card, among which:

- Either a TTL pulse signal or a manual press-button can be used to trigger the start or the stop of the recording.

---

\(^1\)Readlake MASD Inc., 11633 Sorrento Valley road, San Diego, California, USA
K.4. ERROR AND ACCURACY

Table K.1: Performance parameters for the MotionScope 8000S PCI imaging system

<table>
<thead>
<tr>
<th>Frame Rates [frames/sec]</th>
<th>Resolution [pixels]</th>
<th># of frames [-]</th>
<th>Record Time [sec]</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>480x420</td>
<td>2,048</td>
<td>41.0</td>
</tr>
<tr>
<td>50E</td>
<td>240x210</td>
<td>8,192</td>
<td>163.8</td>
</tr>
<tr>
<td>60</td>
<td>480x420</td>
<td>2,048</td>
<td>34.1</td>
</tr>
<tr>
<td>60E</td>
<td>240x210</td>
<td>8,192</td>
<td>136.5</td>
</tr>
<tr>
<td>125</td>
<td>480x420</td>
<td>2,048</td>
<td>16.4</td>
</tr>
<tr>
<td>125E</td>
<td>240x210</td>
<td>8,192</td>
<td>65.6</td>
</tr>
<tr>
<td>250</td>
<td>480x420</td>
<td>2,048</td>
<td>8.2</td>
</tr>
<tr>
<td>250E</td>
<td>240x210</td>
<td>8,192</td>
<td>32.8</td>
</tr>
<tr>
<td>500</td>
<td>320x280</td>
<td>4,096</td>
<td>8.2</td>
</tr>
<tr>
<td>500E</td>
<td>240x210</td>
<td>8,192</td>
<td>16.4</td>
</tr>
<tr>
<td>1000</td>
<td>240x210</td>
<td>8,192</td>
<td>8.2</td>
</tr>
<tr>
<td>2000</td>
<td>160x140</td>
<td>16,384</td>
<td>8.2</td>
</tr>
<tr>
<td>4000S</td>
<td>160x68</td>
<td>32,768</td>
<td>8.2</td>
</tr>
<tr>
<td>4000</td>
<td>100x98</td>
<td>32,768</td>
<td>8.2</td>
</tr>
<tr>
<td>8000S</td>
<td>160x30</td>
<td>65,536</td>
<td>8.2</td>
</tr>
<tr>
<td>8000</td>
<td>60x68</td>
<td>65,536</td>
<td>8.2</td>
</tr>
</tbody>
</table>

- Up to two TTL pulse signals can be displayed as black or white markers directly on the recorded pictures, allowing a correlation with an external even during later post processing of the images.

Further details about the PCI 800S CCD camera can be found in the "Instructions for Operation" manual (PSI, 1999).

K.4 Error and accuracy

To determine the level of uncertainty of a visualization technique is not as straightforward as for a pressure measurement system where a quantitative value can be established to characterize the uncertainty of the measurements. In this subsection, the author has tried to present and discuss the different sources of error and the parameters that are relevant to the accuracy of a Schlieren technique visualization.

K.4.0.1 Schlieren method accuracy

In this paragraph are presented a few parameters that characterize the quality and accuracy of a visualization Schlieren.

Displacement of deflected light in the knife edge plane

In order to achieve a good darkening or brightening of the image in the screen plane,
it is important how light is displaced in the knife edge plane. The displacement \( \Delta h \) of the deflected light over the knife edge is calculated for the standard setup shown in Figure K.3. It is assumed for this calculation that the rays before the test section are parallel, and that an ideal point light source is used.

\[
\epsilon = \frac{L}{C_{GD}} \left( \frac{\partial \rho}{\partial y} \right) \tag{K.3}
\]

with \( L \), the test section width
\( C_{GD} = \frac{\beta}{\rho_s} \), the Gladstone-Glad constant
\( \beta \), a constant (= 0.000292 for Air)
\( \rho_s \), the reference density taken at standard conditions(\( 0^\circ C \) and 100kPa)

Thus, the displacement of a pencil of light at the knife edge plane, assuming that the angular deflection \( \epsilon \) at the lens L2 is small:

\[
\Delta h = f_2 \epsilon \tag{K.4}
\]

The above equation basically implies that the displacement of the deflected light does not depend on the position of the test section, and that the distance from the optical axis has no effect either. Using a lens with larger focal length would cause more deflection and a stronger Schlieren effect.

It also should be noted that a pencil of light which is deflected arrives at the same point on the viewing screen as the undeflected rays, and thus the image on the viewing screen remains sharp. Only the illumination is affected.

**Contrast**

Consequently, the viewing screen (or camera) has a general illumination \( E \) which is proportional to \( h \). A point on the screen that is illuminated by a deflected pencil has an additional illumination \( \Delta E \) which is proportional to \( \Delta h \). The contrast is then defined
K.4. ERROR AND ACCURACY

by:

\[ c = \frac{\Delta E}{E} = \frac{\Delta h}{h} = \frac{f_2 \xi}{h} = \frac{f_3 L}{h C_G D} \left( \frac{\partial \rho}{\partial y} \right) \]  

(K.5)

This equation summarizes the effects that determine the contrast on the viewing screen: the focal length of \( L_2 \), the width of the test section, the refractivity of the fluid, the density gradients, and the uncovered width of the basic image. Thus, for plane flow, the increase or decrease of illumination at the screen is directly proportional to the density gradients.

Sensitivity

Another parameter to evaluate the performances of the system is the sensitivity, defined as the fractional deflection obtained at the knife edge for unit angular deflection of the ray in the test section. Basically, the minimum angle of deflection that can be detected:

\[ s = \frac{c}{\epsilon} = \frac{f_2}{h} \]  

(K.6)

Optical resolution

The resolution of an optical system is defined as its ability to reproduce separate images of objects placed very close to each others (Jenkins and White, 1950). The main limitations for the resolution of flow field features are diffraction effects. In the setup shown in Figure K.1, lens 3 is a critical component. The diffraction at the perimeter limits the quality of the image. Rayleigh determined that two diffraction patterns can be separated until one principal maximum falls over the first minimum of the second pattern.

Noting that lens 3 is positioned close to the knife edge, and referring to its circular aperture with diameter \( D_3 \), Weinstein (1991) stated that the limiting resolution is given by:

\[ d = 1.22 \left( \frac{D_3}{\lambda} + f_3 \right) \lambda \simeq 2.5 \lambda \quad \text{with} \quad \begin{cases} \lambda \in [380; 780] \text{ nm} \\ D_3 = 30 \text{ mm} \end{cases} \]  

(K.7)

Hence, in our case, the limitation of the Schlieren system due to optical aberrations remains non perceivable by the human eye.

K.4.0.2 Some practical considerations

In addition to these basic performance parameters, some other practical matters must be considered and balanced against each others. For instance, equation K.6 indicates that maximum sensitivity is obtained with minimum \( h \), that is, by leaving very little of the source image uncovered. On the other hand, \( h \) must be large enough to furnish sufficient illumination at the screen. Thus its minimum value is limited by the brightness of the source. There is also an upper limit on \( h \) (or the size of the source), determined by the maximum illumination desired (in order not to damage the CCD camera for instance).
In general, high sensitivity is desirable, but even here there is an upper limit, determined by the fact that spurious density gradients are encountered by the light. The system should be not so sensitive as to make these visible. Some such "noise" is always present, in the form of density fluctuations in the surrounding of the test section, as well as in turbulent boundary layer fluctuations on the test-section sidewalls.

If the method is to be used for quantitative measurements, there is another limitation on sensitivity. This is related to the fact $\Delta h$ must not be so large that the secondary image is deflected completely off (or completely onto). The knife edge, for then any additional deflection would not produce corresponding changes in the illumination at the screen. For qualitative work however, this nonlinearity is not so objectionable and may even be desirable.

K.4.0.3 Geometrical and aerodynamical resolution

Aerodynamical resolution

Rather than a quantitative resolution, the aerodynamical resolution is more a physical consideration to be aware of while post processing and analyzing the pictures. Indeed, the bright and dark areas revealed by Schlieren visualizations actually correspond to density gradients integrated through a certain width, including boundary layers on the side walls and possible three-dimensional shock configuration. In order to overcome this problem, the focusing Schlieren method should be recommended as future work. Although it requires a much more sensitive set-up than the conventional Schlieren method, it presents several advantages among which the possibility to focus over a region with a rather small value of depth of focus, thus allowing three-dimensional visualizations. Further details about this method can be found in Weinstein (1991).

CCD camera Resolution

The CCD camera resolution directly contribute to the geometrical resolution and can be quantitatively established. Basically, a Schlieren window with 150mm width and 130mm height was resolved with a 480x420 pixels image giving 3.2 pixels per mm (a geometrical resolution of 0.31mm per pixel). Any other frame rate thereafter uses the same geometrical resolution of 0.31mm per pixel but a different frame resolution like for instance in unsteady visualizations where 8000 frames per second only permit Schlieren window of 160x30 pixels.

Just as Schlieren technique integrates density gradients in space, through the channel width, it also integrates in time the unsteady behavior of the density field. However, a fairly high value of the electronic shutter speed was used both in steady state and unsteady visualizations and provided a exposure frequency 20 times higher than the sampling frequency as mentioned in the acquisition method section.
Appendix L

Laser-Two-Focus working principle

The measuring principle for two-dimensional Laser-Two-Focus (2D L2F) measurements is presented in figure L.1. It shows the measurement volume, consisting of two focused laser beams being generated by the L2F device and working as a light gates. Small oil droplets are injected into the flow at an upstream position and scatters light when passing through each foci. The reflected light impulses are received by photo detectors and the time between them is measured. The particle velocity can thereafter be calculated since the distance between both foci is known from the manufacturer. The second focus point is rotated around the first one to detect the main flow orientation in the plane being normal to the optical axis. To ensure that the same particle travels through both foci a high number of those time-of-flight events must be measured. The mean velocity, flow angle and turbulence intensities parallel and vertical to the main flow direction are thereafter calculated by statistical analysis.

Figure L.1: Working principle of a 2D Laser-Two-Focus system

The 3D-L2F consists of double two-dimensional systems observing the same measurement volume from different angles of incidence. To achieve a high accuracy of the third component along the optical axis, a large angle between both 2D system is required. On the other hand a good optical access into narrow flow channels requires a small angle. For the present system a compromise was found to incline the axes of the two-dimensional system to the central axis to ±7.5°. Figure L.1 displays the principle of operation with
the two single systems, called channel 1 and channel 2. In the actual design, the start beam for channel 1 has a blue wave length (488nm) and the stop beam has a green wave length (514.5nm), while channel 2 has the green wave length for the start beam and the blue wave length for the stop beam.

![Figure L.2: Working principle of a 3D Laser-Two -Focus system](image)

Figure L.2: Working principle of a 3D Laser-Two -Focus system

An Ar-ion laser with a total output of 8 Watt is used as the light source. The emitted laser beams travel from the collimator lens on the left side to the probe volume on the right side. Scattered light from the particles travels the same way back, is collected by the collimator lens and transferred to the photo detectors. The two pairs of beams rotate around the optical axis. The angle between the main flow direction and the reference axis is the finally obtained flow angle. In case of velocity vectors perpendicular to the optical axis (2D flow) the two channels behave as standard L2F systems. Particles traveling along the x-axis are detected in both channels. However, if the velocity has a radial component, the flow information obtained by both channel in two different planes can be correlated to calculate the third flow velocity component. Results are instantaneously treated by the L2F software and saved directly to disk as ASCII data files. Further details and information can be found in Polytec (1997) and Freudenreich (2001).
Appendix M

Steady State results

M.1 Introduction

The present appendix gathers all illustrations of the experimental and numerical steady state results on both the 2D and 3D bump test objects.

The 2D RANS steady state simulations are introduced first. Results from various experiments and 3D RANS calculations are thereafter presented for the 2D and 3D bumps respectively.

From a global perspective, results have been sorted by their operating conditions, from weak to strong shock configurations. Depending on the availability of numerical calculation for a given OP, results are presented in such way that a direct numerical-experimental comparison or an experimental self-comparison is possible.
APPENDIX M. STEADY STATE RESULTS

M.2 Two dimensional RANS computations on 2D nozzle

(a) 2D N-S Mesh No 1

(b) Mach number contours - OP: Psout=120kPa

c) Steady state pressure on 2D bump - OP: Psout=120kPa

(d) Mach number contours - OP: Psout=118kPa

e) Steady state pressure on 2D bump - OP: Psout=118kPa

(f) Mach number contours - OP: Psout=116kPa

g) Steady state pressure on 2D bump - OP: Psout=116kPa
M.2. TWO DIMENSIONAL RANS COMPUTATIONS ON 2D NOZZLE

Figure M.1: Two-dimensional steady state Navier-Stokes simulation of 2D Nozzle
Operating conditions: Pt=160kPa Tt=303K Psout=120-112kPa

(a) 2D N-S Mesh No 2

(b) Mach number contours - OP: Psout=110kPa
(c) Steady state pressure on 2D bump - OP: Psout=110kPa

(j) Mach number contours - OP: Psout=122kPa
(k) Steady state pressure on 2D bump - OP: Psout=112kPa

(h) Mach number contours - OP: Psout=114kPa
(i) Steady state pressure on 2D bump - OP: Psout=114kPa
APPENDIX M. STEADY STATE RESULTS

(d) Mach number contours - OP: $P_{\text{out}}=108\text{kPa}$

(e) Steady state pressure on 2D bump - OP: $P_{\text{out}}=108\text{kPa}$

(f) Mach number contours - OP: $P_{\text{out}}=106\text{kPa}$

(g) Steady state pressure on 2D bump - OP: $P_{\text{out}}=106\text{kPa}$

Figure M.2: Two-dimensional steady state Navier-Stokes simulation of 2D Nozzle
Operating conditions: $P_t=160\text{kPa}$ $T_t=303\text{K}$ $P_{\text{out}}=112-106\text{kPa}$

(a) 2D N-S Mesh No 3

(b) Mach number contours

(c) Steady state pressure on 2D bump

Figure M.3: Two-dimensional steady state Navier-Stokes simulation of 2D Nozzle
Operating conditions: $P_t=160\text{kPa}$ $T_t=303\text{K}$ $P_{\text{out}}=104\text{kPa}$
Figure M.4: Two-dimensional steady state Navier-Stokes simulation of 2D Nozzle
Operating conditions: \( P_t = 160 \text{kPa}, T_t = 303 \text{K}, P_{\text{out}} = 102 \text{kPa} \)
M.3 Experimental and numerical results in 2D nozzle

**EXPERIMENTAL RESULTS**

(a) Schlieren Visualization

**NUMERICAL RESULTS**

(b) Isentropic Mach number at mid-channel

(c) Isentropic Mach number on 2D bump

(d) Isentropic Mach number in 2D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines

Figure M.5: Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈118kPa
M.3. EXPERIMENTAL AND NUMERICAL RESULTS IN 2D NOZZLE

**EXP-EXP COMPARISON**
towards 2D analysis

**EXP-NUM COMPARISON**

(a) Time-averaged pressure distribution
(b) Numerically computed pressure distribution

(c) Experimental results - 2D plot at y=50mm
(d) Exp-Num comparison - 2D plot at y=50mm

(e) Experimental results - 2D plot at y=25mm & 75mm
(f) Exp-Num comparison - 2D plot at y=25mm

(g) Experimental results - 2D plot at y=10mm & 90mm
(h) Exp-Num comparison - 2D plot at y=10mm

Figure M.6: Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{out} \approx 118kPa$
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number at mid-channel

(c) Isentropic Mach number on 2D bump

(d) Isentropic Mach number in 2D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines

(g) Heat transfer coefficient at mid-channel

NUMERICAL RESULTS

Figure M.7: Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Pout≈116kPa
M.3. EXPERIMENTAL AND NUMERICAL RESULTS IN 2D NOZZLE

Figure M.8: Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: $P_t \approx 160$kPa $T_t \approx 303$K $P_{out} \approx 116$kPa
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization
(b) Isentropic Mach number at mid-channel
(c) Isentropic Mach number on 2D bump
(d) Isentropic Mach number in 2D nozzle
(e) Oil visualization over 2D bump
(f) Streamlines
(g) Heat transfer Coefficient over 2D bump

Figure M.9: Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈114kPa
M.3. EXPERIMENTAL AND NUMERICAL RESULTS IN 2D NOZZLE

**(EXP-EXP)**

Comparison towards 2D analysis

**EXP-NUM**

Comparison

---

**Figure M.10:** Experimental and 3D NS numerical results comparison in 2D Nozzle

Operating conditions: $P_t \approx 160kPa$ $T_t \approx 303K$ $Psout \approx 114kPa$
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number at mid-channel

(c) Isentropic Mach number on 2D bump

(d) Isentropic Mach number in 2D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines

(g) Heat transfer coefficient at mid-channel

Figure M.11: Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: \( P_t \approx 160 \text{kPa} \ T_t \approx 303 \text{K} \ P_{\text{out}} \approx 112 \text{kPa} \)
M.3. EXPERIMENTAL AND NUMERICAL RESULTS IN 2D NOZZLE

Figure M.12: Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: $P_t \approx 160$ kPa $T_t \approx 303$ K $P_{out} \approx 112$ kPa

(a) Time-averaged pressure distribution
(b) Numerically computed pressure distribution
(c) Experimental results - 2D plot at $y=50$mm
(d) Exp-Num comparison - 2D plot at $y=50$mm
(e) Experimental results - 2D plot at $y=25$ & 75mm
(f) Exp-Num comparison - 2D plot at $y=25$mm
(g) Experimental results - 2D plot at $y=10$ & 90mm
(h) Exp-Num comparison - 2D plot at $y=10$mm
APPENDIX M. STEADY STATE RESULTS

**EXPERIMENTAL RESULTS**

(a) Schlieren Visualization

(b) Isentropic Mach number at mid-channel

(c) Isentropic Mach number on 2D bump

(d) Isentropic Mach number in 2D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines

(g) Heat transfer coefficient at mid-channel

**NUMERICAL RESULTS**

Figure M.13: Experimental and 3D NS numerical results in 2D Nozzle

Operating conditions: \( P_t \approx 160 \text{kPa} \) \( T_t \approx 303 \text{K} \) \( P_{sout} \approx 110 \text{kPa} \)

362
Figure M.14: Experimental and 3D NS numerical results comparison in 2D Nozzle
Operating conditions: \( P_{\text{t}} \approx 160\text{kPa} \) \( T_{\text{t}} \approx 303\text{K} \) \( P_{\text{out}} \approx 110\text{kPa} \)
APPENDIX M. STEADY STATE RESULTS

**EXPERIMENTAL RESULTS**

(a) Schlieren Visualization

(b) Isentropic Mach number at mid-channel

(c) Isentropic Mach number on 2D bump

(d) Isentropic Mach number in 2D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines

(g) Heat transfer coefficient at mid-channel

**NUMERICAL RESULTS**

Figure M.15: Experimental and 3D NS numerical results in 2D Nozzle

Operating conditions: $P_t \approx 160\, \text{kPa}$ $T_t \approx 303\, \text{K}$ $P_{\text{out}} \approx 108\, \text{kPa}$
M.3. EXPERIMENTAL AND NUMERICAL RESULTS IN 2D NOZZLE

![Figure M.16](image)

**Figure M.16**: Experimental and 3D NS numerical results comparison in 2D Nozzle

Operating conditions: $P_t \approx 160\text{kPa}$ $T_t \approx 303\text{K}$ $P_{out} \approx 108\text{kPa}$
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number at mid-channel

(c) Isentropic Mach number on 2D bump

(d) Isentropic Mach number in 2D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines

(g) Heat transfer coefficient at mid-channel

Figure M.17: Experimental and 3D NS numerical results in 2D Nozzle
Operating conditions: $P_t \approx 160$ kPa $T_t \approx 303$ K $P_{sout} \approx 106$ kPa

NUMERICAL RESULTS
M.3. EXPERIMENTAL AND NUMERICAL RESULTS IN 2D NOZZLE

Figure M.18: Experimental and 3D NS numerical results comparison in 2D Nozzle

Operating conditions: $P_t \approx 160$kPa $T_t \approx 303$K $P_{out} \approx 106$kPa
APPENDIX M. STEADY STATE RESULTS

**EXPERIMENTAL RESULTS**

(a) Schlieren Visualization

(b) Isentropic Mach number at mid-channel

(c) Isentropic Mach number on 2D bump

(d) Isentropic Mach number in 2D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines

(g) Heat transfer coefficient at mid-channel

Figure M.19: Experimental and 3D NS numerical results in 2D Nozzle

Operating conditions: \( P_t \approx 160\,\text{kPa} \), \( T_t \approx 303\,\text{K} \), \( P_{sout} \approx 104\,\text{kPa} \)
**EXP-EXP COMPARISON**

(a) Time-averaged pressure distribution  
(b) Numerically computed pressure distribution

(c) Experimental results - 2D plot at y=50mm  
(d) Exp-Num comparison - 2D plot at y=50mm

(e) Experimental results - 2D plot at y=25&75mm  
(f) Exp-Num comparison - 2D plot at y=25mm

(g) Experimental results - 2D plot at y=10&90mm  
(h) Exp-Num comparison - 2D plot at y=10mm

Figure M.20: Experimental and 3D NS numerical results comparison in 2D Nozzle  
Operating conditions: $P_t \approx 160kPa \ T_t \approx 303K \ Ps_{out} \approx 104kPa$
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number on 2D bump

(c) Oil visualization over 2D bump

(d) Heat transfer coefficient at mid-channel

EXP-EXP COMPARISON towards 2D analysis

(e) Pressure distribution on 2D bump

(f) Experimental results - 2D plot at y=50mm

(g) temp

(h) Experimental results - 2D plot at y=10&90mm

Figure M.21: Experimental steady state results and self comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈102kPa

370
M.3. EXPERIMENTAL AND NUMERICAL RESULTS IN 2D NOZZLE

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number on 2D bump

(c) Oil visualization over 2D bump

(d) Heat transfer coefficient at mid-channel

EXP-EXP COMPARISON
towards 2D analysis

(e) Pressure distribution on 2D bump

(f) Experimental results - 2D plot at y=50mm

(g) Experimental results - 2D plot at y=25&75mm

(h) Experimental results - 2D plot at y=10&90mm

Figure M.22: Experimental steady state results and self comparison in 2D Nozzle
Operating conditions: $P_t \approx 160$ kPa $T_t \approx 303$ K $P_{out} \approx 100$ kPa
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number on 2D bump

(c) Oil visualization over 2D bump

(d) Heat transfer coefficient at mid-channel

EXP-EXP COMPARISON towards 2D analysis

(e) Pressure distribution on 2D bump

(f) Experimental results - 2D plot at y=50mm

(g) Experimental results - 2D plot at y=25&50mm

(h) Experimental results - 2D plot at y=10&90mm

Figure M.23: Experimental steady state results and self comparison in 2D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈098kPa
M.4 Experimental and numerical results on 3D nozzle

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number on 3D bump

(c) Oil visualization over 3D bump

EXP-EXP COMPARISON

(d) Pressure distribution on 3D bump

(e) Experimental results - 2D plot at y=50mm

(f) Experimental results - 2D plot at y=30&70mm

(g) Experimental results - 2D plot at y=10&90mm

Figure M.24: Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: \( P_t \approx 160\text{ kPa} \) \( T_t \approx 303\text{ K} \) \( P_{\text{out}} \approx 128\text{ kPa} \)
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number on 3D bump

(c) Oil visualization over 3D bump

EXP-EXP COMPARISON

(d) Pressure distribution on 3D bump

(e) Experimental results - 2D plot at y=50mm

(f) Experimental results - 2D plot at y=30&70mm

(g) Experimental results - 2D plot at y=10&90mm

Figure M.25: Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: $P_t \approx 160\,\text{kPa}$ $T_t \approx 303\,\text{K}$ $P_{out} \approx 126\,\text{kPa}$
M.4. EXPERIMENTAL AND NUMERICAL RESULTS ON 3D NOZZLE

**EXPERIMENTAL RESULTS**

- (a) Schlieren Visualization
- (b) Isentropic Mach number on 3D bump
- (c) Oil visualization over 3D bump

**EXP-EXP COMPARISON**

- (d) Pressure distribution on 3D bump
- (e) Experimental results - 2D plot at y=50mm
- (f) Experimental results - 2D plot at y=30&70mm
- (g) Experimental results - 2D plot at y=10&90mm

Figure M.26: Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈124kPa

375
Figure M.27: Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: $P_t \approx 160\text{kPa}$ $T_t \approx 303\text{K}$ $P_{out} \approx 122\text{kPa}$
M.4. EXPERIMENTAL AND NUMERICAL RESULTS ON 3D NOZZLE

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number on 3D bump

(c) Oil visualization over 3D bump

EXP-EXP COMPARISON

(d) Pressure distribution on 3D bump

(e) Experimental results - 2D plot at y=50mm

(f) Experimental results - 2D plot at y=30&70mm

(g) Experimental results - 2D plot at y=10&90mm

Figure M.28: Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈120kPa
APPENDIX M. STEADY STATE RESULTS

**EXPERIMENTAL RESULTS**

(a) Schlieren Visualization

(b) Isentropic Mach number at mid-channel

(c) Experimental isentropic Mach number

(d) Isentropic Mach number in 3D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines on 3D bump surface

(g) Numerical streamlines on 3D bump surface and side wall

**NUMERICAL RESULTS**

Figure M.29: Experimental and 3D NS numerical results in 3D Nozzle

Operating conditions: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{out} \approx 118kPa$

378
M.4. EXPERIMENTAL AND NUMERICAL RESULTS ON 3D NOZZLE

Figure M.30: Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: $P_t \approx 160\,\text{kPa}$ $T_t \approx 303\,\text{K}$ $P_{out} \approx 118\,\text{kPa}$
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(b) Isentropic Mach number at mid-channel

(c) Experimental isentropic Mach number

(d) Isentropic Mach number in 3D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines on 3D bump surface

(g) Numerical streamlines on 3D bump surface and side wall

Figure M.31: Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{sout} \approx 116kPa$

NUMERICAL RESULTS
M.4. EXPERIMENTAL AND NUMERICAL RESULTS ON 3D NOZZLE

EXPERIMENTAL-NUMERICAL COMPARISON

(a) Time-averaged pressure distribution  (b) Numerically computed pressure distribution

(c) Exp-Num comparison - 2D plot at y=50mm

(d) Exp-Num comparison - 2D plot at y=10mm  (e) Exp-Num comparison - 2D plot at y=90mm

(f) Exp-Num comparison - 2D plot at y=30mm  (g) Exp-Num comparison - 2D plot at y=70mm

Figure M.32: Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: \( P_t \approx 160 \text{kPa} \) \( T_t \approx 303 \text{K} \) \( P_{out} \approx 116 \text{kPa} \)
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number at mid-channel

(c) Experimental isentropic Mach number

(d) Isentropic Mach number in 3D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines on 3D bump surface

(g) Numerical streamlines on 3D bump surface

Figure M.33: Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: \( P_t \approx 160 \text{kPa} \), \( T_t \approx 303 \text{K} \), \( P_{out} \approx 114 \text{kPa} \)
M.4. EXPERIMENTAL AND NUMERICAL RESULTS ON 3D NOZZLE

EXPERIMENTAL- NUMERICAL COMPARISON

(a) Time-averaged pressure distribution  (b) Numerically computed pressure distribution

(c) Exp-Num comparison - 2D plot at y=50mm

(d) Exp-Num comparison - 2D plot at y=10mm  (e) Exp-Num comparison - 2D plot at y=90mm

(f) Exp-Num comparison - 2D plot at y=30mm  (g) Exp-Num comparison - 2D plot at y=70mm

Figure M.34: Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈114kPa
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(c) Experimental isentropic Mach number

(e) Oil visualization over 2D bump

Figure M.35: Experimental and 3D NS numerical results in 3D Nozzle

Operating conditions: $P_t \approx 160kPa \ T_t \approx 303K \ P_{out} \approx 112kPa$

NUMERICAL RESULTS

(b) Isentropic Mach number at mid-channel

(d) Isentropic Mach number in 3D nozzle

(f) Streamlines on 3D bump surface

(g) Numerical streamlines on 3D bump surface and side wall
M.4. EXPERIMENTAL AND NUMERICAL RESULTS ON 3D NOZZLE

EXPERIMENTAL-NUMERICAL COMPARISON

(a) Time-averaged pressure distribution  
(b) Numerically computed pressure distribution

(c) Exp-Num comparison - 2D plot at y=50mm

(d) Exp-Num comparison - 2D plot at y=10mm  
(e) Exp-Num comparison - 2D plot at y=90mm

(f) Exp-Num comparison - 2D plot at y=30mm  
(g) Exp-Num comparison - 2D plot at y=70mm

Figure M.36: Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈112kPa
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number at mid-channel

Numerical RESULTS

(c) Experimental isentropic Mach number

(d) Isentropic Mach number in 3D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines on 3D bump surface

(g) Numerical streamlines on 3D bump surface and side wall

Figure M.37: Experimental and 3D NS numerical results in 3D Nozzle

Operating conditions: $P_t \approx 160\text{kPa}$ $T_t \approx 303\text{K}$ $P_{out} \approx 110\text{kPa}$
M.4. EXPERIMENTAL AND NUMERICAL RESULTS ON 3D NOZZLE

EXPERIMENTAL-NUMERICAL COMPARISON

(a) Time-averaged pressure distribution
(b) Numerically computed pressure distribution

(c) Exp-Num comparison - 2D plot at y=50mm

(d) Exp-Num comparison - 2D plot at y=10mm
(e) Exp-Num comparison - 2D plot at y=90mm

(f) Exp-Num comparison - 2D plot at y=30mm
(g) Exp-Num comparison - 2D plot at y=70mm

Figure M.38: Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: \( P_t \approx 160 \text{kPa} \), \( T_t \approx 303 \text{K} \), \( P_{out} \approx 110 \text{kPa} \)

387
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(c) Experimental isentropic Mach number

(e) Oil visualization over 2D bump

(g) Numerical streamlines on 3D bump surface and side wall

NUMERICAL RESULTS

(b) Isentropic Mach number at mid-channel

(d) Isentropic Mach number in 3D nozzle

(f) Streamlines on 3D bump surface

Figure M.39: Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈108kPa
Figure M.40: Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Pout≈108kPa
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

NUMERICAL RESULTS

(b) Isentropic Mach number at mid-channel

(c) Experimental isentropic Mach number

(d) Isentropic Mach number in 3D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines on 3D bump surface

(g) Numerical streamlines on 3D bump surface and side wall

Figure M.41: Experimental and 3D NS numerical results in 3D Nozzle

Operating conditions: Pt≈160kPa Tt≈303K Psout≈106kPa
M.4. EXPERIMENTAL AND NUMERICAL RESULTS ON 3D NOZZLE

EXPERIMENTAL-NUMERICAL COMPARISON

(a) Time-averaged pressure distribution  (b) Numerically computed pressure distribution

(c) Exp-Num comparison - 2D plot at y=50mm

(d) Exp-Num comparison - 2D plot at y=10mm  (e) Exp-Num comparison - 2D plot at y=90mm

(f) Exp-Num comparison - 2D plot at y=30mm  (g) Exp-Num comparison - 2D plot at y=70mm

Figure M.42: Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈106kPa
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number at mid-channel

(c) Experimental isentropic Mach number

(d) Isentropic Mach number in 3D nozzle

(e) Oil visualization over 2D bump

(f) Streamlines on 3D bump surface

(g) Numerical streamlines on 3D bump surface and side wall

Figure M.43: Experimental and 3D NS numerical results in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈104kPa
M.4. EXPERIMENTAL AND NUMERICAL RESULTS ON 3D NOZZLE

EXPERIMENTAL-
NUMERICAL
COMPARISON

(a) Time-averaged pressure distribution  (b) Numerically computed pressure distribution

(c) Exp-Num comparison - 2D plot at y=50mm

(d) Exp-Num comparison - 2D plot at y=10mm  (e) Exp-Num comparison - 2D plot at y=90mm

(f) Exp-Num comparison - 2D plot at y=30mm  (g) Exp-Num comparison - 2D plot at y=70mm

Figure M.44: Experimental and 3D NS numerical results comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈104kPa
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

EXP-EXP COMPARISON

(a) Schlieren Visualization

(b) Isentropic Mach number on 3D bump

(c) Oil visualization over 3D bump

(d) Pressure distribution on 3D bump

(e) Experimental results - 2D plot at y=50mm

(f) Experimental results - 2D plot at y=30&70mm

(g) Experimental results - 2D plot at y=10&90mm

Figure M.45: Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈102kPa
Figure M.46: Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈100kPa
APPENDIX M. STEADY STATE RESULTS

EXPERIMENTAL RESULTS

(a) Schlieren Visualization

(b) Isentropic Mach number on 3D bump

(c) Oil visualization over 3D bump

EXP-EXP COMPARISON

(d) Pressure distribution on 3D bump

(e) Experimental results - 2D plot at y=50mm

(f) Experimental results - 2D plot at y=30&70mm

(g) Experimental results - 2D plot at y=10&90mm

Figure M.47: Experimental steady state results and self comparison in 3D Nozzle
Operating conditions: Pt≈160kPa Tt≈303K Psout≈098kPa
Appendix N

Unsteady results

N.1 Introduction

The present appendix gathers all illustrations of the experimental and numerical unsteady results on both the 2D and 3D bump test objects. Results from unsteady pressure measurement and high speed Schlieren visualizations are first presented for different operating conditions on the 2D bump test object. Unsteady results from 2D RANS simulations at similar and new OPs are thereafter presented. Finally, unsteady results from pressure measurements and 3D RANS calculations in the 3D nozzle are presented.

From a global perspective, results have been sorted with the increasing perturbation frequency for each different OP investigated. Tables 5.9 and 7.1 presents the different unsteady operating conditions for both experimental investigation and numerical simulation of the 2D and 3D bump test objects.

N.2 Unsteady results in two-dimensional nozzle

N.2.1 Unsteady pressure measurements
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification - Second harmonic

(h) Unsteady pressure phase angle - Second harmonic

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.1: Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: $P_t \approx 160kPa \ T_t \approx 303K \ P_{out} \approx 106kPa \ F_p=50Hz \ \Delta p = \pm 2.12kPa$
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

Figure N.2: Unsteady pressure measurement results in 2D Nozzle

Expe-OP1: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{out} \approx 106kPa$ $F_p = 50Hz$ $A_p = \pm 2.12kPa$
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution
(b) Comparison Steady state-Time Averaged pressure distribution
(c) Unsteady pressure amplification - Fundamental
(d) Unsteady pressure phase angle - Fundamental
(e) Unsteady pressure amplification - First harmonic
(f) Unsteady pressure phase angle - First harmonic
(g) Unsteady pressure amplification - Second harmonic
(h) Unsteady pressure phase angle - Second harmonic
(i) Shock motion amplitude
(j) Shock motion phase angle

Figure N.3: Unsteady pressure measurement results in 2D Nozzle
Expo-OP1: Pt≈160kPa Tt≈303K Psout≈106kPa Fp=100Hz Ap=±2.12kPa

400
Figure N.4: Unsteady pressure measurement results in 2D Nozzle

Expe-OP1: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{out} \approx 106kPa$ $F_p = 100Hz$ $A_p = \pm 2.12kPa$
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution
(b) Comparison Steady state-Time Averaged pressure distribution
(c) Unsteady pressure amplification - Fundamental
(d) Unsteady pressure phase angle - Fundamental
(e) Unsteady pressure amplification - First harmonic
(f) Unsteady pressure phase angle - First harmonic
(g) Unsteady pressure amplification - Second harmonic
(h) Unsteady pressure phase angle - Second harmonic

(i) Shock motion amplitude
(j) Shock motion phase angle

Figure N.5: Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: $P_t \approx 160kPa, T_t \approx 303K, P_{out} \approx 106kPa, F_p = 250Hz, A_p = \pm 2.12kPa$
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

Figure N.6: Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: \( P_t \approx 160 \text{kPa} \) \( T_t \approx 303 \text{K} \) \( P_{out} \approx 106 \text{kPa} \) \( F_p = 250 \text{Hz} \) \( A_p = \pm 2.12 \text{kPa} \)
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification - Second harmonic

(h) Unsteady pressure phase angle - Second harmonic

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.7: Unsteady pressure measurement results in 2D Nozzle Expe-OP1: \(P_t \approx 160\text{kPa} \quad T_t \approx 303\text{K} \quad P_{out} \approx 106\text{kPa} \quad F_p = 500\text{Hz} \quad A_p = \pm 2.12\text{kPa}\)
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

Figure N.8: Unsteady pressure measurement results in 2D Nozzle
Expe-OP1: \( P_t \approx 160kPa \), \( T_t = 303K \), \( P_{sout} \approx 106kPa \), \( F_p = 500Hz \), \( A_p = \pm 2.12kPa \)
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution
(b) Comparison Steady state-Time Averaged pressure distribution
(c) Unsteady pressure amplification - Fundamental
(d) Unsteady pressure phase angle - Fundamental
(e) Unsteady pressure amplification - First harmonic
(f) Unsteady pressure phase angle - First harmonic
(g) Unsteady pressure amplification - Second harmonic
(h) Unsteady pressure phase angle - Second harmonic
(i) Shock motion amplitude
(j) Shock motion phase angle

Figure N.9: Unsteady pressure measurement results in 2D Nozzle
Expe-OP2: $P_t \approx 160\text{kPa}$ $T_t \approx 303\text{K}$ $P_{out} \approx 112\text{kPa}$ $f_p = 50\text{Hz}$ $A_p = \pm 1.25\text{kPa}$
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

Figure N.10: Unsteady pressure measurement results in 2D Nozzle Expe-OP2: $P_t \approx 160\,\text{kPa}$ $T_t \approx 303\,\text{K}$ $P_{s_{\text{out}}} \approx 112\,\text{kPa}$ $F_p = 50\,\text{Hz}$ $A_p = \pm 1.25\,\text{kPa}$
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification - Second harmonic

(h) Unsteady pressure phase angle - Second harmonic

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.11: Unsteady pressure measurement results in 2D Nozzle
Expo-OP2: Pt\textapprox160kPa Tt\approx303K Psout\approx112kPa F_p=100Hz A_p = \pm 1.25kPa
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

Figure N.12: Unsteady pressure measurement results in 2D Nozzle

Expe-OP2: $P_t \approx 160\text{kPa}$ $T_t \approx 303\text{K}$ $P_{\text{out}} \approx 112\text{kPa}$ $f_p = 100\text{Hz}$ $A_p = \pm 1.25\text{kPa}$
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution
(b) Comparison Steady state-Time Averaged pressure distribution
(c) Unsteady pressure amplification - Fundamental
(d) Unsteady pressure phase angle - Fundamental
(e) Unsteady pressure amplification - First harmonic
(f) Unsteady pressure phase angle - First harmonic
(g) Unsteady pressure amplification - Second harmonic
(h) Unsteady pressure phase angle - Second harmonic
(i) Shock motion amplitude
(j) Shock motion phase angle

Figure N.13: Unsteady pressure measurement results in 2D Nozzle
Expo-OP2: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{out} \approx 112kPa$ $F_p = 250Hz$ $A_p = \pm 1.25kPa$
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

Experiment - 2D Nozzle KTH

(a) Unsteady pressure amplification at y=10mm

(b) Unsteady pressure phase angle at y=10mm

(c) Unsteady pressure amplification at y=25mm

(d) Unsteady pressure phase angle at y=25mm

(e) Unsteady pressure amplification at mid-channel with steady state pressure distribution for extreme rod positions

(f) Unsteady pressure phase angle at mid-channel with steady state pressure distribution for extreme rod positions

(g) Unsteady pressure amplification at y=75mm

(h) Unsteady pressure phase angle at y=75mm

(i) Unsteady pressure amplification at y=90mm

(j) Unsteady pressure phase angle at y=90mm

Figure N.14: Unsteady pressure measurement results in 2D Nozzle

Expe-OP2: Pt≈160kPa Tt≈303K Psout≈112kPa Fp=250Hz Ap = ±1.25kPa
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification - Second harmonic

(h) Unsteady pressure phase angle - Second harmonic

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.15: Unsteady pressure measurement results in 2D Nozzle
Expo-OP2: Pt≈160kPa Tt≈303K Psout≈112kPa Fp=500Hz Ap=±1.25kPa
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

![Graphs showing unsteady pressure measurement results in 2D Nozzle](image)

Figure N.16: Unsteady pressure measurement results in 2D Nozzle
Expe-OP2: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{sout} \approx 112kPa$ $F_p = 500Hz$ $A_p = \pm 1.25kPa$
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification - Second harmonic

(h) Unsteady pressure phase angle - Second harmonic

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.17: Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{out} \approx 106kPa$ $F_p=50Hz$ $A_p = \pm 1.14kPa$
**N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE**

Figure N.18: Unsteady pressure measurement results in 2D Nozzle

Experiments - 2D Nozzle KTH

Unsteady flow conditions: \( F_p = 50 \text{Hz} \), \( A_p = \pm 1.20 \text{kPa} \)

Y=10mm

Steady state OP: \( P_{t}^{\text{in}} \approx 159.96 \text{kPa} \)

\( P_{s}^{\text{out}} = 106.00 \text{kPa} \)

Vert. rod

Horiz. rod

\( X[\text{mm}] \)

Amplitude Fundamental

Amplitude 1st Harmonic

T-A pressure

Phase 1st Harm.

Phase Fund.

Harmonic

Phase

\( T-A \) pressure

\( X[\text{mm}] \)

(1) Unsteady pressure amplification at \( y=10\text{mm} \)

(2) Unsteady pressure phase angle at \( y=10\text{mm} \)

(3) Unsteady pressure amplification at \( y=25\text{mm} \)

(4) Unsteady pressure phase angle at \( y=25\text{mm} \)

(5) Unsteady pressure amplification at mid-channel with steady state pressure distribution for extreme rod positions

(6) Unsteady pressure phase angle at mid-channel with steady state pressure distribution for extreme rod positions

(7) Unsteady pressure amplification at \( y=75\text{mm} \)

(8) Unsteady pressure phase angle at \( y=75\text{mm} \)

(9) Unsteady pressure amplification at \( y=90\text{mm} \)

(10) Unsteady pressure phase angle at \( y=90\text{mm} \)

\( F_{p} = 50 \text{Hz} \), \( A_{p} = \pm 1.14 \text{kPa} \)
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution
(b) Comparison Steady state-Time Averaged pressure distribution

c) Unsteady pressure amplification - Fundamental
(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic
(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification - Second harmonic
(h) Unsteady pressure phase angle - Second harmonic

(i) Shock motion amplitude
(j) Shock motion phase angle

Figure N.19: Unsteady pressure measurement results in 2D Nozzle
Expo-OP3: $P_t \approx 160kPa$, $T_t \approx 303K$, $P_{out} \approx 106kPa$, $F_p = 100Hz$, $A_p = \pm 1.14kPa$
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

Figure N.20: Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: Pt≈160kPa Tt≈303K P'sout≈106kPa Fp=100Hz A_p = ±1.14kPa
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification - Second harmonic

(h) Unsteady pressure phase angle - Second harmonic

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.21: Unsteady pressure measurement results in 2D Nozzle
Expo-OP3: $P_t \approx 160kPa$, $T_t \approx 303K$, $P_{sout}=106kPa$, $F_p=250Hz$, $A_p = \pm 1.14kPa$
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

![Diagrams](Images)

(a) Unsteady pressure amplification at y=10mm
(b) Unsteady pressure phase angle at y=10mm
(c) Unsteady pressure amplification at y=25mm
(d) Unsteady pressure phase angle at y=25mm
(e) Unsteady pressure amplification at mid-channel with steady state pressure distribution for extreme rod positions
(f) Unsteady pressure phase angle at mid-channel with steady state pressure distribution for extreme rod positions
(g) Unsteady pressure amplification at y=75mm
(h) Unsteady pressure phase angle at y=75mm
(i) Unsteady pressure amplification at y=90mm
(j) Unsteady pressure phase angle at y=90mm

Figure N.22: Unsteady pressure measurement results in 2D Nozzle
Expe-OP3: $P_t \approx 160\text{kPa} \quad T_t \approx 303\text{K} \quad P_{out} \approx 106\text{kPa} \quad F_p = 250\text{Hz} \quad A_p = \pm 1.14\text{kPa}$
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification - Second harmonic

(h) Unsteady pressure phase angle - Second harmonic

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.23: Unsteady pressure measurement results in 2D Nozzle

Expo-OP3: Pt≈160kPa Tt≈303K Psout≈106kPa F_p=500Hz A_p=±1.14kPa
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

Figure N.24: Unsteady pressure measurement results in 2D Nozzle

Expe-OP3: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{sout} \approx 106kPa$ $F_p = 500Hz$ $A_p = \pm 1.14kPa$
APPENDIX N. UNSTEADY RESULTS

N.2.2 Unsteady simulations

Figure N.25: Unsteady 2D RANS numerical results in 2D Nozzle
Num-OP1: \( P_{t} \approx 160 \text{kPa} \quad T \approx 303 \text{K} \quad P_{\text{out}} \approx 120 \text{kPa} \quad F_{p} = 100 \text{Hz} \quad A_{p} = \pm 2.4 \text{kPa} \)
Figure N.26: Unsteady 2D RANS numerical results in 2D Nozzle

Num-OP1: $P_i \approx 160\text{kPa}$ $T_t \approx 303\text{K}$ $P_{sout} \approx 120\text{kPa}$ $F_p = 500\text{Hz}$ $A_p = \pm 2.4\text{kPa}$
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification on bump surface - Fundamental

(h) Unsteady pressure phase angle on bump surface - Fundamental

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.27: Unsteady 2D RANS numerical results in 2D Nozzle

Num-OP1: $P_t \approx 160\text{kPa}$, $T_t \approx 303\text{K}$, $P_{out} \approx 120\text{kPa}$, $F_p = 1000\text{Hz}$, $A_p = \pm 2.4\text{kPa}$
Figure N.28: Unsteady 2D RANS numerical results in 2D Nozzle
Num-OP2: $P_t \approx 160\text{kPa}$ $T_t \approx 303\text{K}$ $P_{out} \approx 116\text{kPa}$ $F_p = 100\text{Hz}$ $A_p = \pm 2.32\text{kPa}$
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification on bump surface - Fundamental

(h) Unsteady pressure phase angle on bump surface - Fundamental

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.29: Unsteady 2D RANS numerical results in 2D Nozzle

Num-OP2: Pt≈160kPa Tt≈303K Psout=116kPa Fp=500Hz Ap = ±2.32kPa
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

Figure N.30: Unsteady 2D RANS numerical results in 2D Nozzle
Num-OP2: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{out} \approx 116kPa$ $F_p = 1000Hz$ $A_p = \pm 2.32kPa$
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification on bump surface - Fundamental

(h) Unsteady pressure phase angle on bump surface - Fundamental

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.31: Unsteady 2D RANS numerical results in 2D Nozzle

Num-OP3: $P_t \approx 160\text{kPa}$ $T_t \approx 303\text{K}$ $P_{out} \approx 110\text{kPa}$ $F_p = 100\text{Hz}$ $A_p = \pm 2.2\text{kPa}$
Figure N.32: Unsteady 2D RANS numerical results in 2D Nozzle
Num-OP3: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{out} \approx 110kPa$ $F_p = 500Hz$ $A_p = \pm 2.2kPa$
APPENDIX N. UNSTEADY RESULTS

(a) Time-Averaged pressure distribution

(b) Comparison Steady state-Time Averaged pressure distribution

(c) Unsteady pressure amplification - Fundamental

(d) Unsteady pressure phase angle - Fundamental

(e) Unsteady pressure amplification - First harmonic

(f) Unsteady pressure phase angle - First harmonic

(g) Unsteady pressure amplification on bump surface - Fundamental

(h) Unsteady pressure phase angle on bump surface - Fundamental

(i) Shock motion amplitude

(j) Shock motion phase angle

Figure N.33: Unsteady 2D RANS numerical results in 2D Nozzle

Num-OP3: P_t≈160kPa T_t≈303K P_out≈110kPa \( F_p = 1000\text{Hz} \) \( A_p = \pm 2.2\text{kPa} \)
Figure N.34: Unsteady 2D RANS numerical results in 2D Nozzle

Num-OP4: \( P_t \approx 160\text{kPa} \ T_t \approx 303\text{K} \ P_{\text{out}} \approx 102\text{kPa} \ F_p = 100\text{Hz} \ A_p = \pm 2.04\text{kPa} \)
APPENDIX N. UNSTEADY RESULTS

Figure N.35: Unsteady 2D RANS numerical results in 2D Nozzle
Num-OP4: $P_t \approx 160\text{kPa}$ $T_t \approx 303\text{K}$ $P_{out} \approx 102\text{kPa}$ $F_p = 500\text{Hz}$ $A_p = \pm 2.04\text{kPa}$
N.2. UNSTEADY RESULTS IN TWO-DIMENSIONAL NOZZLE

Figure N.36: Unsteady 2D RANS numerical results in 2D Nozzle
Num-OP4: Pt≈160kPa Tt≈303K Psout≈102kPa F_p=1000Hz A_p = ±2.04kPa
N.3 Unsteady results on three-dimensional nozzle

N.3.1 Unsteady pressure measurements

Figure N.37: Unsteady pressure measurement results in 3D Nozzle
Exp-OP: \( P_t \approx 160kPa \) \( T_t \approx 303K \) \( P_{out} \approx 116kPa \) \( F_p = 50Hz \) \( A_p = \pm 2.10kPa \)
N.3. UNSTEADY RESULTS ON THREE-DIMENSIONAL NOZZLE

(a) Unsteady pressure amplification at y=20mm

(b) Unsteady pressure phase angle at y=20mm

(c) Unsteady pressure amplification at y=35mm

(d) Unsteady pressure phase angle at y=35mm

(e) Unsteady pressure amplification at y=50mm

(f) Unsteady pressure phase angle at y=50mm

(g) Unsteady pressure amplification at y=65mm

(h) Unsteady pressure phase angle at y=65mm

(i) Unsteady pressure amplification at y=80mm

(j) Unsteady pressure phase angle at y=80mm

Figure N.38: Unsteady pressure measurement results in 3D Nozzle
Expe-OP: Pt≈160kPa Tt≈303K Psout≈116kPa Fp=50Hz Ap = ±2.10kPa
APPENDIX N. UNSTEADY RESULTS

Figure N.39: Unsteady pressure measurement results in 3D Nozzle
Expe-OP: Pt=160kPa Tt≈303K Pout≈116kPa F_p=100Hz A_p = ±2.10kPa
N.3. UNSTEADY RESULTS ON THREE-DIMENSIONAL NOZZLE

Experiments - 3D Nozzle KTH DFSD on Bump Surface Pressure

(a) Unsteady pressure amplification at y=20mm

(b) Unsteady pressure phase angle at y=20mm

(c) Unsteady pressure amplification at y=35mm

(d) Unsteady pressure phase angle at y=35mm

(e) Unsteady pressure amplification at y=50mm

(f) Unsteady pressure phase angle at y=50mm

(g) Unsteady pressure amplification at y=65mm

(h) Unsteady pressure phase angle at y=65mm

(i) Unsteady pressure amplification at y=80mm

(j) Unsteady pressure phase angle at y=80mm

Figure N.40: Unsteady pressure measurement results in 3D Nozzle
Exp-OP: Pt≈160kPa Tt≈303K Psout≈116kPa Fp=100Hz Ap=±2.10kPa
Figure N.41: Unsteady pressure measurement results in 3D Nozzle
Expe-OP: Pt=160kPa Tt≈303K Psout≈116kPa F_p=250Hz A_p=±2.10kPa
Figure N.42: Unsteady pressure measurement results in 3D Nozzle
Expe-OP: $P_t=160\text{kPa}$ $T_t=303\text{K}$ $P_{out}=116\text{kPa}$ $F_p=250\text{Hz}$ $A_p = \pm 2.10\text{kPa}$
Figure N.43: Unsteady pressure measurement results in 3D Nozzle
Expe-OP: Pt=160kPa Tt≈303K Psout≈116kPa \( F_p=500Hz \) \( A_p = \pm 2.10kPa \)
Experiments - 3D Nozzle KTH

Figure N.44: Unsteady pressure measurement results in 3D Nozzle

Expe-OP: Pt\(\approx\)160kPa \(T_{t}\approx\)303K \(P_{\text{out}}\approx\)116kPa \(F_{\text{p}}\approx\)500Hz \(A_{\text{p}}\approx\)\(\pm\)2.10kPa
N.3.2 Three-dimensional RANS calculation

(a) Time-Averaged pressure distribution

(b) Unsteady pressure amplification - Fundamental

(c) Unsteady pressure phase angle - Fundamental

(d) Unsteady pressure amplification - First harmonic

(e) Unsteady pressure phase angle - First harmonic

Figure N.45: Unsteady 3D RANS numerical results in 3D Nozzle

Unsteady pressure distribution on 3D bump

Num-OP: $P_t \approx 160\text{kPa}$ $T_t \approx 303\text{K}$ $P_{\text{out}} \approx 116\text{kPa}$ $f_p = 500\text{Hz}$ $A_p = \pm 2.32\text{kPa}$
Figure N.46: Unsteady 3D RANS numerical results in 3D Nozzle

Unsteady pressure distribution on 3D bump

Num-OP: Pt≈160kPa Tt≈303K Psout≈116kPa Fp=500Hz Ap=±2.32kPa
APPENDIX N. UNSTEADY RESULTS

(a) Unsteady pressure amplification at y=20mm
(b) Unsteady pressure phase angle at y=20mm
(c) Unsteady pressure amplification at y=35mm
(d) Unsteady pressure phase angle at y=35mm
(e) Unsteady pressure amplification at y=50mm
(f) Unsteady pressure phase angle at y=50mm
(g) Unsteady pressure amplification at y=65mm
(h) Unsteady pressure phase angle at y=65mm
(i) Unsteady pressure amplification at y=80mm
(j) Unsteady pressure phase angle at y=80mm

Figure N.47: Unsteady 3D RANS numerical results in 3D Nozzle
Unsteady pressure distribution in 3D nozzle channel
Num-OP: $P_t \approx 160kPa$ $T_t \approx 303K$ $P_{sout} \approx 116kPa$ $F_p = 500Hz$ $A_p = \pm 2.32kPa$
N.3. UNSTEADY RESULTS ON THREE-DIMENSIONAL NOZZLE

Figure N.48: Unsteady 3D RANS numerical results in 3D Nozzle

Unsteady shock motion in 3D nozzle channel
Num-OP: Pt≈160kPa Tt≈303K Psout≈116kPa F_p=500Hz A_p = ±2.32kPa
APPENDIX N. UNSTEADY RESULTS
Bibliography


BIBLIOGRAPHY


BIBLIOGRAPHY


BIBLIOGRAPHY


BIBLIOGRAPHY


BIBLIOGRAPHY


Pearcey, H. (1968). The interaction between local effects at the shock and rear separation - a source of significant scale effect in wind tunnel tests on airfoils and wings. In *AGARD CP-35*.


