Numerical Investigation of Blade Flutter at or Near Stall in Axial Turbomachines

Doctoral Thesis

by

Wolfgang Höhn
Abstract

During the design of the compressor and turbine stages of today's aeroengines aerodynamically induced vibrations become increasingly important since higher blade load and better efficiency are desired. Aerodynamically induced vibrations in turbomachines can be classified into two general categories, i.e. selfexcited vibrations, usually denoted as flutter, and forced response. In the first case the aerodynamic forces acting on the structure are dependent on the motion of the structure. In the latter case the aerodynamic forces can be considered to be independent of the structural motion. In this thesis the development of a method based on the unsteady, compressible Navier-Stokes equations in two dimensions is described in order to study the physics of flutter for unsteady viscous flow around cascaded vibrating blades at stall.

The governing equations are solved by a finite difference technique in boundary fitted coordinates. The numerical scheme uses the Advection Upstream Splitting Method to discretise the convective terms and central differences discretizing the diffusive terms of the fully non-linear Navier-Stokes equations on a moving H-type mesh. The unsteady governing equations are explicitly and implicitly marched in time in a time-accurate way using a four stage Runge-Kutta scheme on a parallel computer or an implicit scheme of the Beam-Warming type on a single processor. Turbulence is modeled using the Baldwin-Lomax turbulence model. The blade flutter phenomenon is simulated by imposing a harmonic motion on the blade, which consists of harmonic body translation in two directions and a rotation, allowing an interblade phase angle between neighboring blades. An aerodynamic instability is given which can lead to a flutter problem, if the computed unsteady pressure forces amplify the imposed blade motion. Nonreflecting boundary conditions are used for the unsteady analysis at inlet and outlet of the computational domain. The computations are performed on multiple blade passages in order to account for nonlinear effects. Unsteady boundary conditions are developed considering primary and secondary gust effects towards the investigation of the forced response problem with the present method.

Subsonic massively stalled and transonic separated unsteady flow cases in compressor and turbine cascades are studied. The results, compared with experiments and the predictions of other researchers, show good agreement for inviscid and viscous flow cases for the investigated flow situations with respect to the steady and unsteady pressure distribution on the blade in the vicinity of shocks and in separated flow areas. The results show the applicability of the new scheme for stalled flow around cascaded blades. As expected the viscous and inviscid methods show different results in areas where viscous effects are important, i.e. separated flow and shock waves. In particular, different predictions for inviscid and viscous flow for the aerodynamic damping for the investigated flow cases are found.

Keywords: turbomachinery, flutter, forced response, gust, unsteady aerodynamics, Navier-Stokes equations, Advection Upstream Splitting Method, implicit scheme, nonreflecting boundary conditions, gust boundary conditions, parallel computing
Preface

This thesis is partly based on the Teknologi Licenciat thesis [Höhn, 1996] entitled “Unsteady Viscous Flow Around Cascaded Vibrating Blades at or Near Stall” presented in November 1996 and the papers, technical notes and reports listed below.

The following documents are enclosed in the appendix.

- Paper A:

- Paper B:
  Höhn, W. and Fransson, T.H.; 1998 a

- Paper C:
  Höhn, W. and Fransson, T.H.; 1998 b
  "Flutter Analysis in Turbomachines Using a Modified Flux Vector Splitting Scheme.", AIAA Paper AIAA 98-3430.

- Paper D:
  Höhn, W. and Fransson, T.H.; 1999

- Paper E:
  Höhn, W.; 1999

The following documents are not enclosed in the appendix.

- Paper F:
  Höhn, W.; 1996

- Paper G:
  Höhn, W.; 1997 a
97/1, Chair of Heat and Power Technology, Royal Institute of Technology, Stockholm, Sweden, 1997.

- Paper H:
  Hölm, W.; 1997 c

- Paper I:

- Paper J:
  Hölm, W.; 1998
Acknowledgments

I would like to express my gratitude to Professor Torsten H. Fransson at the Chair of Heat and Power Technology at the Royal Institute of Technology of Stockholm, who made this work possible.

The study was carried out in the beginning within the NUTEK project "Unsteady Flow Around Cascaded Blades at or Near Stall". Later the study was supported by the NFFP project "Aeromechanical Vibrations in Cascades". The financial support of ASEA Brown Boveri (CH), Volvo Aero Corporation AB (S), Rolls Royce plc. (UK) and NUTEK¹ (S) during the first project and FMV² (S) and VOLVO Aero Corporation (S) in the second project is gratefully acknowledged.

My thank goes to the opponent of the thesis Dr. Joseph Verdon, United Technologies Research Center, East Hartford (United States) and to the members of the jury committee: Prof. Dr.-Ing. Heinz Gallus, Technical University of Aachen (Germany), Dr. rer. nat. Volker Carstens from the German Aerospace Center DLR in Göttingen (Germany) and Mr. Hans Mårtensson from VOLVO Aero Corporation in Trolähättan (Sweden).

Moreover, the author would like to thank Dr. Peter Ott, "Laboratoire de Thermique Appliquée et de Turbomachine - École Polytechnique Fédérale de Lausanne", for the discussions about the original code and his comments as opponent on the "Teknologi Licenci" thesis.

Very special thanks go to Mr. Hans Mårtensson, Dr. Gabor Csaba, Dr. Pieter Groth, Dr. David Lindström and Dr. Lenard Helmers at VOLVO Aero Corporation (VAC) for their discussions and assistance in the second project especially during my stays at VAC.

My colleague Jiasen Hu deserves many thanks for the fruitful discussions about turbulent flow in engineering applications. I thank Mrs. Virginie Casanovas, Mr. Markus Jöcker, Mr. Kai Freundenreich and Mr. Francois Moyroud for the interesting collaboration in the field of forced response in axial turbomachines.

Thanks go to Dr. Volker Carstens for allowing me to stay at the German Aerospace Center DLR in Göttingen and Mr. Björn Grüber for his hospitality during this stay and his assistance in vectorizing the presented numerical method at the NEC supercomputer at DLR in Braunschweig, Germany.

Henrik Enbuske, Tomas Börjesson, Andreas Gauch, Johan Linnarsson, Joachim Ahlm, Christian Trapp and Arif Khalifa did a very good job here during their diploma thesis within the two projects in the field of flutter and forced response.

Moreover, I would like to thank the “Flygtekniska Försöksanstalten” (FFA) in Stockholm for providing me with mesh generator FFANET and Mr. Lars Tysell from FFA helping me to do the first steps with this program.

I would like to thank the "Ecole Polytechnique Fédérale de Lausanne (EPFL)" (CH), the "National Supercomputer Center" at the University of Linköping (S), the "Department of Numerical Analysis and Computing Science (NADA/KTH)" (S) especially the "Parallel

¹Project manager: Dr. Wiktør Raklow, Project number: 91-00448P
²Project manager: Mr. Dag Brising, Project number NFFP-304
Dator Centrum (PDC/KTH), the Supercomputing Center in Umeå (S), the German Aerospace Center DLR for granting me innumerable cpu hours without which this work would not have been possible.

Finally, I would like to thank my family and all my friends in “good old” Germany, in France and in Belgium and the new ones I meet in Sweden at the institution and in my leisure time.
“...Da saß der fleißige junge Mensch an seinen Abenden und stocherte eine solche Maschine (Radioapparat) zusammen, hingerissen von der Idee der Drahtlosigkeit, anbetend auf frommen Knien vor dem Gott der Technik, welcher es fertiggebracht hat, nach Jahrtausenden Dinge zu entdecken und höchst unvollkommen darzustellen, welche jeder Denker schon immer gewußt und klüger benutzt hat ...”

source: Hermann Hesse, “Der Steppenwolf”
Contents

Abstract .................................................................................. I
Preface ................................................................................... II
Acknowledgments .................................................................. IV
List of Figures .......................................................................... X
List of Tables ........................................................................ XIV
Nomenclature .......................................................................... XV
Abbreviations .......................................................................... XVIII

1 Introduction ........................................................................ 1
  1.1 Flutter in turbomachines ............................................... 1
  1.2 Previous work ............................................................... 5
  1.3 Present work ................................................................. 7
    1.3.1 Objectives ............................................................... 7
    1.3.2 Approach ............................................................... 7

2 Governing Equations ............................................................. 10
  2.1 Differential form .......................................................... 10
  2.2 Transformation of the governing equations ...................... 12
    2.2.1 Thin layer approximation ....................................... 12
  2.3 Algebraic turbulence model ........................................... 13
  2.4 Initial and boundary conditions ...................................... 15
    2.4.1 Initial conditions .................................................. 15
    2.4.2 Boundary conditions ............................................. 15

3 Numerical Discretisation .......................................................... 24
  3.1 Discretisation of the convective terms ........................... 24
3.1.1  Van Leer's Flux Vector Splitting Scheme for viscous flow  ..........  26
3.1.2  AUSM - Mach number splitting compared to AUSM V/D ...............  27
3.2  Discretisation of the diffusive terms  ......................................  27
3.3  Mesh movement  ...........................................................................  30
3.4  Time discretisation of the 2D Navier-Stokes equations .......... ............  30
   3.4.1  Explicit time integration  .......................................................  30
   3.4.2  Implicit time integration  .......................................................  34
3.5  Discretisation of the Baldwin-Lomax turbulence model .......................  40
3.6  Stability criteria  ........................................................................  40
   3.6.1  Steady state  ........................................................................  40
   3.6.2  Unsteady flow  .....................................................................  40
3.7  Convergence criteria  .................................................................  41
   3.7.1  Steady state  ........................................................................  41
   3.7.2  Unsteady flow  .....................................................................  41

4  Results  ..................................................................................  43
4.1  Steady state results  .................................................................  44
   4.1.1  Basic test cases  ...................................................................  44
   4.1.2  Cascade flow  ......................................................................  47
   4.1.3  Conclusions  ......................................................................  64
4.2  Unsteady results  .......................................................................  65
   4.2.1  Flat plate  ...........................................................................  65
   4.2.2  Stcf 5  ..............................................................................  66
   4.2.3  Stcf 10 ...............................................................................  74
   4.2.4  Stcf 11 ...............................................................................  79
   4.2.5  Conclusions  ......................................................................  82
4.3  Comparison of INSTHPT with other codes  .......................... ...............  83
4.4  Summary of the papers in the appendix  .......................................  83
   4.4.1  Ecomas 96  .........................................................................  84
   4.4.2  ISUAAT 97  .........................................................................  84
   4.4.3  AIAA 98  ............................................................................  84
CONTENTS

4.4.4 AIAA 99 .......................................................... 85

5 Conclusions and Future Work 86
  5.1 Summary of the work .............................................. 86
  5.2 Conclusions .......................................................... 87
  5.3 Future work .......................................................... 88

Literature ............................................................. 90

Appendices ........................................................... 102

A Paper A ............................................................... 103

B Paper B ............................................................... 111

C Paper C ............................................................... 128

D Paper D ............................................................... 139

E Paper E ............................................................... 153

F Boundary conditions used in the parallel code 173

G Convective terms $\eta$ - direction 174

H Relationships for reference values 177
  H.1 Reynolds number .................................................. 177
  H.2 Wall shear stress .................................................. 178
List of Figures

1.1 Trent 800, [Rolls-Royce plc., 1986] ................................................. 1
1.2 Schematic of common types of fan/compressor blade flutter according to Nixon, [1989] ................................................................. 2
1.3 Turbine characteristic [Cohen et al., 1990] ........................................... 4
1.4 Two dimensional cascade according [Dorney and Verdon, 1994] ........... 8

3.1 Computational cell ................................................................. 28
3.2 Sketch of the parallel code of INSTHPT ....................................... 32
3.3 Computational subdomains .......................................................... 33
3.4 Speed up against the serial code (1 node) as a function of the number of nodes of the parallel code ..................................................... 34

4.1 Velocity distribution on a flat plate Re = 1000000, 100 x 100 mesh-points 44
4.2 Skin friction distribution on a flat plate Re = 1000000, 100 x 100 mesh-points ................................................................. 45
4.3 $u^+ = f(y^+)$ on a flat plate Re = 1000000 in comparison with the “log law” of the wall and the law for the “laminar” sublayer, 100 x 100 mesh-points 46
4.4 Flow around a cylinder in 2D, geometry ........................................ 46
4.5 H-type mesh used for the computations, 106 x 100 mesh-points ............ 47
4.6 Separation angle of the separation bubble in comparison with other authors, 106 x 100 mesh-points ..................................................... 47
4.7 Length of the separation bubble in comparison with other authors, 106 x 100 mesh-points ................................................................. 48
4.8 H-type mesh used for standard configuration 4, 106 x 91 meshpoints ....... 49
4.9 Steady state surface pressure distribution of standard configuration 4, case 3 50
4.10 Mesh used for standard configuration 5, 120 x 100 meshpoints ............ 50
4.11 Steady state surface pressure distribution of standard configuration 5, case 2 51
4.12 Steady state surface pressure distribution of standard configuration 5, case 3, \(i = 5\degree\) .................................................. 52
4.13 Length of the separation bubble against the angle of incidence, standard configuration 5 .................................................. 53
4.14 Steady state surface pressure distribution of standard configuration 5, case 12, \(i = 7.5\degree\) .................................................. 54
4.15 Steady state surface pressure distribution of standard configuration 5, case 18, \(i = 10\degree\) .................................................. 56
4.16 H-type mesh used for standard configuration 10, 140 x 100 meshpoints .................................................. 58
4.17 Steady state surface pressure distribution for STCF 10, \(M_1 = 0.7\) .................................................. 58
4.18 Skin friction for STCF 10, \(M_1 = 0.7\) .................................................. 60
4.19 Mesh used for standard configuration 10, \(M_1 = 0.8\) .................................................. 60
4.20 Steady state Mach number distribution of standard configuration 10, \(M_1 = 0.8\) .................................................. 61
4.21 Skin friction of standard configuration 10, \(M_1 = 0.8\) .................................................. 61
4.22 Mesh used for standard configuration 11, subsonic case .................................................. 62
4.23 Steady state Mach number distribution of standard configuration 11, subsonic case .................................................. 63
4.24 Mesh used for standard configuration 11, transonic case .................................................. 64
4.25 Steady state Mach number distribution of standard configuration 11, off-design case .................................................. 65
4.26 Amplitude of unsteady surface pressure distribution flat plate .................................................. 66
4.27 Phase of unsteady surface pressure distribution flat plate .................................................. 66
4.28 Amplitude of the unsteady pressure over chord for standard configuration 5 case 7, \(i = 4\degree\), reflecting boundary conditions applied in INSTHPT .................................................. 67
4.29 Phase of the unsteady pressure over chord for standard configuration 5 case 7, \(i = 4\degree\), reflecting boundary conditions applied in INSTHPT .................................................. 68
4.30 Amplitude of the unsteady pressure over chord for standard configuration 5 case 7, \(i = 4\degree\), nonreflecting boundary conditions applied in INSTHPT .................................................. 68
4.31 Phase of the unsteady pressure over chord for standard configuration 5 case 7, \(i = 4\degree\), nonreflecting boundary conditions applied in INSTHPT .................................................. 69
4.32 Amplitude of unsteady surface pressure distribution of standard configuration 5, case 11, \(i = 6\degree\) .................................................. 70
4.33 Phase of unsteady surface pressure distribution of standard configuration 5, case 11, \(i = 6\degree\) .................................................. 70
4.34 Amplitude of unsteady surface pressure distribution of standard configuration 5, case 12, \( i = 8^\circ \) .................................................. 71
4.35 Phase of unsteady surface pressure distribution of standard configuration 5, case 12, \( i = 8^\circ \) .................................................. 72
4.36 Amplitude of unsteady surface pressure distribution of standard configuration 5, case 18, \( i = 10^\circ \) .................................................. 73
4.37 Phase of unsteady surface pressure distribution of standard configuration 5, case 18, \( i = 10^\circ \) .................................................. 73
4.38 Damping as a function of incidence angle standard configuration 5 .... 74
4.39 Amplitude of unsteady surface pressure distribution of standard configuration 10, case 23 .................................................. 75
4.40 Phase of unsteady surface pressure distribution of standard configuration 10, case 23 .................................................. 75
4.41 Amplitude of unsteady surface pressure distribution of standard configuration 10, case 24 .................................................. 76
4.42 Phase of unsteady surface pressure distribution of standard configuration 10, case 24 .................................................. 77
4.43 Amplitude of unsteady surface pressure distribution of standard configuration 10, case 31 .................................................. 79
4.44 Phase of unsteady surface pressure distribution of standard configuration 10, case 31 .................................................. 79
4.45 Amplitude of unsteady surface pressure distribution of standard configuration 11, subsonic case 656, \( \sigma = 180.0 \) .................................................. 80
4.46 Phase of unsteady surface pressure distribution of standard configuration 11, subsonic case 656, \( \sigma = 180.0 \) .................................................. 80
4.47 Amplitude of unsteady surface pressure distribution of standard configuration 11, off-design case 134 .................................................. 82
4.48 Phase of unsteady surface pressure distribution of standard configuration 11, off-design case 134 .................................................. 82
List of Tables


4.1 Flow parameters, flat plate for \( Re_\infty = 1 \, 000 \, 000 \) ........................................ 44

4.2 Flow parameters, flat plate for \( Re_\infty = 1 \, 000 \, 000 \), turbulent calculation . 45

4.3 Flow parameters, 2D cylinder .................................................. 46

4.4 Flow parameters, standard configuration 4, case 3 ......................... 48

4.5 Flow parameters, international standard configuration 4, case 3 .... 49

4.6 Flow parameters, standard configuration 5, case 2 ...................... 50

4.7 Flow parameters, international standard configuration 5, case 2 .... 51

4.8 Flow parameters, standard configuration 5, case 3 ...................... 52

4.9 Flow parameters, international standard configuration 5, case 3 .... 52

4.10 Flow parameters, standard configuration 5, case 12 .................. 53

4.11 Flow parameters, international standard configuration 5, case 12 .... 54

4.12 Flow parameters, standard configuration 5, case 18 .................. 55

4.13 Flow parameters, international standard configuration 5, case 18 .... 55

4.14 Flow parameters, standard configuration 10, \( M_1 = 0.7 \) .............. 57

4.15 Flow parameters, international standard configuration 10, \( M_1 = 0.7 \) . 57

4.16 Flow parameters, standard configuration 10, \( M_1 = 0.8 \) .............. 59

4.17 Flow parameters, international standard configuration 10, \( M_1 = 0.8 \) . 59

4.18 Flow parameters, standard configuration 11, subsonic case ............. 62

4.19 Flow parameters, international standard configuration 11, subsonic case . 63

4.20 Flow parameters, standard configuration 11, transonic case .......... 63

4.21 Flow parameters, international standard configuration 11, transonic case . 64

4.22 Acoustic resonance conditions standard configuration 5 .............. 73
<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>4.23</td>
<td>Damping, international standard configuration 10, case 23</td>
</tr>
<tr>
<td>4.24</td>
<td>Damping, international standard configuration 10, case 24</td>
</tr>
<tr>
<td>4.25</td>
<td>Damping, international standard configuration 10, case 31</td>
</tr>
<tr>
<td>4.26</td>
<td>Damping, international standard configuration 11, subsonic case</td>
</tr>
<tr>
<td>4.27</td>
<td>Damping, international standard configuration 11, transonic case</td>
</tr>
<tr>
<td>4.28</td>
<td>Acoustic resonance conditions standard configuration 11</td>
</tr>
<tr>
<td>F.1</td>
<td>Types of boundary conditions used for the parallel code</td>
</tr>
</tbody>
</table>
## Nomenclature

### Variable | Meaning | Unit
---|---|---

#### Latin alphabet, lower case letters

- $a$: speed of sound
- $c$: chord length
- $c_p$: specific heat at constant pressure
- $c_v$: specific heat at constant volume
- $c_f$: local skin friction coefficient
- $c_p$: steady pressure coefficient
- $\tilde{c}_p$: unsteady pressure coefficient
- $d$: blade thickness
- $e_c$: total energy per unit mass
- $f$: frequency
- $h$: specific enthalpy
- $h$: dimensionless (with chord) bending- or torsion-amplitude
- $i$: angle of incidence
- $k$: specific heat conductivity
- $k$: reduced frequency $k = \frac{c_o}{\bar{\omega}_e}$
- $\bar{n}$: normal unit vector
- $p$: static pressure
- $t$: time
- $\bar{u}$: fluid velocity vector (with the velocity components $u$ in x-direction and $v$ in y-direction)
- $u, v$: fluid velocity components in the x, y-direction
- $u_r$: friction velocity
- $x$: coordinate in axial direction
- $y$: coordinate in circumferential direction

#### Latin alphabet, upper case letters

- $A^+$: constant in the Baldwin-Lomax turbulence model
- $C_{CR}$: constant in the Baldwin-Lomax turbulence model
- $C_{Kleb}$: Klebanoff constant in the Baldwin-Lomax turbulence model
- $C_{MUTM}$: constant in the Baldwin-Lomax turbulence model
- $C_{WK}$: wake constant in the Baldwin-Lomax turbulence model
- $H$: total enthalpy
- $J$: Jacobi determinant
- $K$: Klauser constant in the Baldwin-Lomax turbulence model
- $L$: characteristic length scale
- $M$: Mach number
- $P$: pitch
- $Pr$: Prandtl number
- $Re$: Reynolds number
- $R$: specific gas constant
\( R^+ \) Riemann invariant
\( R^- \) Riemann invariant
\( T \) temperature \([K]\)
\( T \) period time of the blade motion \([s]\)
\( Y \) normal distance form the wall or the wake centerline \([m]\)

**Greek alphabet**

\( \alpha \) angle of incidence \([\text{deg}]\)
\( \beta \) flow angle in the moving coordinate system \([\text{deg}]\)
\( \delta \) partial derivative \([-]\)
\( \delta \) boundary layer thickness \([m]\)
\( \delta \) bending vibration direction = \( tan^{-1}(h_y/h_x) \) \([\text{deg}]\)
\( \delta_1 \) displacement thickness \([m]\)
\( \delta_2 \) momentum thickness \([m]\)
\( \gamma \) stagger angle \([\text{deg}]\)
\( \gamma \) ratio of specific heats \([-]\)
\( \kappa \) von Karman constant \([-]\)
\( \mu \) dynamic viscosity \([\text{kg} \cdot \text{m}^{-1}]\)
\( \lambda \) mixing length \([m]\)
\( \nu \) kinematic viscosity \([\text{m}^2 \cdot \text{s}^{-1}]\)
\( \rho \) density \([\text{kg} \cdot \text{m}^{-3}]\)
\( \tau_{x,x,y,y} \) shear stress tensor
\( \eta \) circumferential coordinate in the computational space \([-]\)
\( \epsilon \) computational error \([-]\)
\( \xi \) axial coordinate in the computational space \([-]\)
\( \sigma \) interblade phase angle, positive in positive \([\text{deg}]\)
\( y \) (circumferential) direction, see figure 1.4
\( \omega \) circular frequence, \( \omega = 2\pi f \) \([\text{rad} \cdot \text{s}^{-1}]\)
\( \Omega \) vorticity \([\text{rad} \cdot \text{s}^{-1}]\)
\( \Xi \) aerodynamic work, see equation 1.1
\( \Phi \) flux limiter \([-]\)

**Superscripts**

\(-\) mean value
\(\sim\) perturbation value, unsteady quantity
\(\cdot\) dimensionless value, \(-\infty\) used
\(\ast\) dimensionless value
\(+\) dimensionless turbulent value
\(\dagger,\ddagger\) vector notation

**Subscripts**

e boundary layer edge
is isentropic
eff effective
ref reference value
\[ L \quad \text{left} \\
R \quad \text{right} \\
T \quad \text{total value} \\
t \quad \text{turbulent value} \\
w \quad \text{total value} \\
w \quad \text{value at the wall} \\
1 \quad \text{values at the inlet in physical space} \\
2 \quad \text{values at the outlet in physical space} \\
-\infty \quad \text{reference values at the middle in the inlet in physical space} \\
t \quad \text{partial derivative with respect to } t \\
x \quad \text{partial derivative with respect to } x \\
y \quad \text{partial derivative with respect to } y \\
\eta \quad \text{partial derivative with respect to } \eta \\
\tau \quad \text{partial derivative with respect to } \tau \\
\xi \quad \text{partial derivative with respect to } \xi \]
Abbreviations

### Variable

<table>
<thead>
<tr>
<th>Latin alphabet</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>( a = \sqrt{\frac{\gamma p}{\rho}} )</td>
<td>speed of sound</td>
</tr>
<tr>
<td>( c_f = \frac{2\mu (\frac{\partial u}{\partial y})<em>{w}}{q</em>{\infty} \Re_{\infty}} )</td>
<td>local skin friction coefficient</td>
</tr>
<tr>
<td>( c_p = \frac{\rho_{\infty} \gamma_{m} - p_{\infty}}{p_{\infty} - p_{1}} )</td>
<td>pressure coefficient</td>
</tr>
<tr>
<td>( e_{c} = \frac{1}{\gamma-1} p + \frac{q^2}{2} )</td>
<td>total energy per unit mass</td>
</tr>
<tr>
<td>( h = T \gamma_{m} / (\gamma_{m} - 1) )</td>
<td>enthalpy</td>
</tr>
<tr>
<td>( H = h + \frac{q^2}{2} )</td>
<td>total enthalpy</td>
</tr>
<tr>
<td>( M_{eff} = \sqrt{\frac{2}{\gamma-1} \left[ \left( \frac{p_{\infty}}{p_{1}} \right)^{\frac{\gamma_{m}}{\gamma_{m} - 1}} - 1 \right]} )</td>
<td>effective Mach number</td>
</tr>
<tr>
<td>( M_{is} = \sqrt{\frac{2}{\gamma-1} \left[ \left( \frac{p_{\infty}}{p_{1}} \right)^{\frac{\gamma_{m}}{\gamma_{m} - 1}} - 1 \right]} )</td>
<td>isentropic Mach number, inlet</td>
</tr>
<tr>
<td>( p = (\gamma - 1) \rho \left( e_{c} - \frac{q^2}{2} \right) )</td>
<td>static pressure</td>
</tr>
<tr>
<td>( p = \rho T )</td>
<td>static pressure</td>
</tr>
<tr>
<td>( Pr = \frac{\mu_{w}}{k} )</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>( p_{1} = p_{1} \left( \frac{\gamma_{m}}{2} M_{is}^2 + 1 \right)^{\frac{2}{\gamma_{m}-1}} )</td>
<td>total pressure, inlet</td>
</tr>
<tr>
<td>( Re = \frac{\rho_{\infty} u_{w}}{\mu_{w}} )</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>( q = \sqrt{u^2 + v^2} )</td>
<td>magnitude of the velocity vector</td>
</tr>
<tr>
<td>( u^{+} = \sqrt{\frac{Re_{\infty} \gamma_{m}^{2}}{\gamma_{m}-1} \left( \frac{\mu_{w}}{\rho_{\infty} u_{w}} \right) \left( 1 - \frac{u}{u_{w}} \right)} )</td>
<td>turbulent velocity, see appendix H</td>
</tr>
</tbody>
</table>

### Greek alphabet

\[
\delta_1 (x) = \int_{0}^{\delta} \left( 1 - \frac{\rho u}{\rho_{\infty} u_{w}} \right) dy \quad \text{displacement thickness}
\]
\[
\delta_2 (x) = \int_{0}^{\delta} \left( \frac{\rho u}{\rho_{\infty} u_{w}} \right) \left( 1 - \frac{u}{u_{w}} \right) dy \quad \text{momentum thickness}
\]

\(^{3}\text{dimensionless, see chapter 2 for reference values}\)
Chapter 1

Introduction

1.1 Flutter in turbomachines

Aerodynamically induced vibrations in turbomachines can be classified into two general categories, namely flutter and forced vibrations [Försching, 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.

Figure 1.1: Trent 800, [Rolls-Royce plc., 1986]

The phenomenon of flutter was observed firstly in civil engineering and aircrafts. Due to wind gusts skyscrapers, chimneys or bridges were exited. The most famous example was the suspension bridge of Tacoma, USA, which was destroyed by aerodynamic oscillations
in 1940 [Försching, 1974]. The efforts to control aircraft wing flutter go back to World War I. At that time one tried to avoid wires which stabilized the wings.

The theory of flutter was developed by Theodosoren [1935], Glauert [1928] and Küssner [1929]. An excellent review of the research of flutter and the later development is done by Ashley [1986], Platzer and Carta [1987, 1988] and Srinivasan [1997].

In the design of turbomachines flutter occurred when the first steam turbines where developed [Mellanby and Kerr, 1923]. At that time damping wires were used in order to get rid of the blade vibration problem. Flow induced fatigue failures of compressor and turbine blades in jet engines have occurred since the advent of the aircraft gas turbine (Shannon, [1945]; and Pherson, [1953]). Bendiksen [1993] concluded that this type of blade failures often happened in engines that incorporated a novel structural or aerodynamic design. Two well documented cases involving the P & W F100 engine fan stall flutter problem (Jeffers and Meece [1975]) and the TF43-A 100 engine (Troha and Swain [1976]) are presented in the literature. Important experimental research and contributions on blade flutter has been done by Carta [1982], Fleeter and Capece [1984]. Bölcs and Fransson [1986], Fransson [1990] and Fransson and Verdon [1993] investigated systematically cascade flutter and the available flutter prediction methods.

Figure 1.1 shows a modern jet engine. The air passes first through the compressor part, enters the combustion chamber and thereafter the turbine section.

![Figure 1.2: Schematic of common types of fan/compressor blade flutter according to Nixon, [1989]](image-url)
1.1. FLUTTER IN TURBOMACHINES

Figure 1.2 shows schematically a compressor map with the boundaries for the most common types of flutter. Subsonic/transonic positive incidence flutter and the classical supersonic flutter are considered to be the most important types [Nixon, 1989], 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 part 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 [Bölcs and Siter, 1986]. Classical flutter can happen at the operating point.

Negative incidence flutter, also referred 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 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.

Supersonic shock flutter (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 at fan blades. The dominating parameters are Mach number, reduced frequency, interblade phase angle and shock position. The vibratory modes are bending and pitching.

Supersonic high-incidence or stalled flutter (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 the fan blades of the compressor section of a gas turbine. The blades are highly loaded and strong shocks occurs. The important parameters are the Mach number and the reduced frequency.

Figure 1.3 shows a turbine map, where different speeds in comparison with the design speed are given.

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 (values of \( \frac{p_{03}}{p_{04}} \approx 1.5 \)).

In turbines shock flutter can be reached for high outlet Mach numbers (values of

---

1 Here, a system with a single degree of freedom is assumed.
$p_{03}/p_{04} \approx 2.0$). For such flow conditions shock boundary layer interaction can induce separation. For supersonic flutter the turbine is choked (values of $p_{03}/p_{04} \approx 3.0$ or higher).

When flutter is addressed in turbomachines, it is often argued that due to the 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. The stability of the motion is determined by calculating the aerodynamic damping. The aerodynamic damping by the fluid on the blade during one period of the blade motion is given for a system with a single degree of freedom by:

$$
\Xi = \frac{1}{A_0 c_h} \int_0^T \left[ \int_{\Gamma} c_p (\vec{u} \cdot \vec{n}) \, ds \right] dt. \tag{1.1}
$$

Here, $A_0$, $c_h$, $c_p$, $\vec{u}$ and $\vec{n}$ are the blade motion amplitude, chord, surface pressure coefficient, local blade velocity and the unit normal vector on the blade surface directed outwards. The surface integral is calculated around the blade boundary $\Gamma$ and the time integral is evaluated for one flutter period $T$. The aerodynamic damping $\Xi$ can be negative as well as positive. Practically, if the aerodynamic damping plus the structural damping is negative then flutter will occur.
1.2 Previous work

Since the beginning of the 1970's a number of methods for computing unsteady compressible turbomachinery flow have been developed. The first methods investigated the 2D flow through a linear cascade. A further development was the introduction of a stream tube thickness leading to quasi 3D methods. Since about 1994 codes have been developed tackling the flutter problem of a complete blade, i.e. 3D flutter codes, and very recently first methods investigating several blade passages appeared.

The simplest methods investigate the flow through a cascade of flat plate airfoils. In this analysis small harmonic perturbations to a uniform inviscid mean flow at zero incidence and zero time-mean load are studied. The obtained perturbation equation for the primitive variables, i.e. the density, the pressure and the velocity, satisfies the Helmholtz equation \(^2\), which can be derived from the conservation of mass, momentum and an isentropic relation. It was solved by Smith [1971] for subsonic flow replacing the airfoils by vortices or using pressure dipoles [Kaji and Okazaki, 1970; Ni, 1979]. The program LINSUB published by Whitehead [1987] based on the thesis of Smith [1971] does this very efficiently for subsonic flow. The extension to supersonic flow is described by Whitehead [1987]. The method has been found very useful in prediction of supersonic fan flutter [Verdon, 1986]. Besides these 2-D methods Namba [1977] and Salain [1987] developed 3-D codes based on the same idea.

Linear methods which superimpose unsteady linear perturbations on a nonlinear steady solution are an efficient way to compute realistic flows through cascades. In such approaches the unsteady equations of motion are linearized in time with respect to small harmonic perturbation. Consequently, the unsteady surface pressure distribution is independent of the magnitude of the small harmonic motion. These equations are linear differential equations with variable coefficients depending on the steady flow field. For instance, the linear potential equation can be solved with a finite element method [Whitehead, 1990] on a fixed grid and assume the flow to be adiabatic, reversible and irrotational, so the equations are those for the velocity potential in 2-D and quasi 3-D flow. These methods are approximately ten times more expensive considering the CPU-time than the flat plate analysis. With these methods for potential flow it is possible to study effects of the airfoil thickness and aerodynamic loading also due to shock motion, provided the shocks are weak. This is due to the assumption of an irrotational and isentropic flow. As Giles [1989] pointed out the Mach number ahead of the shock should be lower than 1.3 in order to keep the assumption of irrotational and isentropic flow.

The linear Euler methods are not restricted to isentropic and irrotational flow. The linear perturbations can be expressed as linear differential equations, which are coupled to the steady flow field through the variable coefficients. The first method was introduced by Ni and Siso [1976] for 2D flow. The idea was picked up in the 1980's by Hall [1987], who published a scheme with shock fitting. Contributions have been made also by Hall and Lawrence [1992] for 3D flow and by Kahl and Klose [1993] for quasi 3D flow situations. Montgomery and Verdon [1998] developed a 3D linearized Euler analysis and investigated the unsteady 3D flow field around a compressor blade. Three dimensional linearized Navier-Stokes calculations for flutter were performed by Holmes et al. [1998] among

\(^2\)In the present context, the Helmholtz equation can be derived from the wave equation by the method of separation of variables.
others.

Some fluid dynamic problems cannot be linearized. For instance, in the vicinity of strong shocks as well as for large amplitudes of the blade motion, therefore, the fully (nonlinear) Euler equations are solved by time marching methods instead by different authors (Joubert [1984], Fransson and Pandolfi [1986], Giles [1986], Gerolymos [1988], He [1989] and Kau [1990]). It requires at least an order of magnitude more computer time and computer storage than the linearized methods. In the first step the solution for the steady state flow is calculated, which is then used as initial condition for the unsteady analysis, in which the Euler equations are solved in a time accurate manner. Groth et al. [1996] and Carstens [1994] developed among others a 3D Euler method. In order to simulate the blade motion the computational mesh has to be deformed such that the computational blade boundary at all points in time coincides with the actual physical blade boundary. To account for this mesh motion extra terms arise in the governing equations.

Due to the fact that Euler codes are restricted by the inviscid assumption to flow without viscous induced separation time-accurate, nonlinear, Reynolds averaged Navier-Stokes codes were developed (Rai [1989], Sidén [1991] and Giles and Haines [1991]). The main limitations of the Navier-Stokes codes are due to the commonly used Boussinesq assumption for the turbulence model, which is required to make computations possible on current computers. The above defined aerodynamic damping, equation 1.1, can be strongly influenced by separated flow situations and shocks in compressors and in turbines. At high Reynolds numbers viscous effects are present in a thin boundary layer close to the blade. By a coupled approach between an Euler method and an integral boundary layer method accurate predictions of cases with boundary layer separations can be achieved (He and Denton, [1993]). Recently Grüber and Carstens [1996] and Ayer and Verdon [1996] developed 2D Navier-Stokes methods based on the Baldwin-Lomax turbulence model. Weber [1997] and Isomura [1996] established methods based on the quasi 3D Navier-Stokes equations, where Weber used the Badwin-Lomax turbulence model and Isomura implemented the Johnson-King turbulence model. Finally, first 3D unsteady Navier-Stokes methods appear, i.e. Chew et al. [1998] among others, which are very limited in their physical prediction capabilities, since the computational mesh is to coarse to resolve viscous effects like boundary layers and 3D vortex structures. More mesh points in order to resolve these viscous phenomena would lead to computational performance requirements, i.e. number of cpu-hours and memory, which are not available at today’s supercomputers.

The present models have led to acceptable prediction capabilities of unsteady inviscid and viscous flow through vibrating two- and three dimensional cascades and around blades at attached flow conditions without shock waves. To date, however, the most dangerous flow condition, namely when a blade operates at high angle of attack, i.e. dynamic stall, is not possible to predict and has been only tackled for moderately separated flow. Dynamic stall describes the complex phenomena that results in the dynamic delay of stall on airfoils and around oscillating blades in turbomachines, when the angle of incidence where stall appears is significantly higher than for the static (steady state) stall case. So far it has been extensively investigated for airfoil wings by Ekaterinaris [1995]. Eguchi and Wiedermann [1995] studied some basic phenomena for stalled flow around cascaded blades. Weber [1997] investigated moderately separated flow. It seems to be an important effect in turbomachinery and it has not been fully understood yet. Isomura [1996] pointed out the importance of a transition point model during the investigation of stall flutter.
1.3 Present work

1.3.1 Objectives

The main objective of this study is to predict 2D viscous flow around vibrating blades at or near stall with special emphasis to find out when viscous effects are important. In particular flow cases representing subsonic and transonic stall flutter are investigated by choosing the international standard configuration 5, 10 and 11 [Fransson et al., 1998]. The obtained results are compared with experimental data [Fransson and Verdon, 1993] and results obtained by other flow solvers for unsteady subsonic and transonic flow cases, i.e. Hodberg [1994] and Ahlinder [1994] and results from Grüber and Carstens [1996], Groth et al. [1996], Ayer and Verdon [1996], Huff [Huff and Reddy, 1990] and Hall [Hall and Clark, 1991].

The second objective emphasize the numerical method, i.e. the applicability of a modified flux vector splitting scheme for steady and unsteady, viscous, separated flow. Moreover, the influence of different boundary conditions is investigated with respect to unsteady pressure on the blade.

1.3.2 Approach

The developed numerical method accounts for unsteady, viscous, fully turbulent flow around cascaded blades towards a better understanding of the above mentioned flow situations, i.e. viscous and nonlinear effects for blade flutter in turbomachines. It uses the "Advection Upstream Splitting Method" (AUSM) for the discretisation of the convective terms [Wada and Liou, 1994] and central differences for the diffusive terms [Hänel et al., 1987] in the Navier-Stokes equations. The AUSM is a flux vector splitting scheme as the van Leer's flux vector splitting method [van Leer, 1982]. In contrast to the latter the pressure term is separated from the convective terms of the Navier-Stokes equations and treated differently. Advantages of the method are the simple implementation of the method in comparison to, for instance, flux difference splitting methods [Roe, 1981] and that there is no need to specify numerical dissipation parameters as, for example, for "central schemes" [Jameson, 1982]. On the other hand improved prediction capabilities in the viscous boundary layer compared with the van Leer splitting [van Leer, 1982] were found. Turbulence is modeled by using the Baldwin-Lomax turbulence model [Baldwin and Lomax, 1978]. Despite its weakness for largely separated flow, see discussion in chapter 4.1, the model is used since it is easy to implement and computationally robust. Algebraic turbulence models are widely used for steady state calculations. The application of such a model for unsteady flow is based on the assumption that the flutter period is much longer than the time it takes for the turbulent viscosity to adapt to a certain velocity profile.

The time accurate integration of the governing equations is performed by applying an explicit 4 stage Runge-Kutta scheme [Leyland et al., 1994] on a parallel computer (IBM SP2) as well as an implicit factorized time integrating scheme [Beam and Warming, 1978] using implicit non-reflecting boundary conditions [Chakravarthy, 1983], which makes the code feasible for single processor use.

The analysis is limited to one blade row, which is assumed to be infinitely long in the
circumferential direction, see figure 1.4, i.e. the flow conditions at the inlet and outlet are given. Hence, the influence of neighboring blade rows as well as 3D flow effects are not considered. At the farfield boundary, viscous effects are assumed to be negligible and for steady state capacitive boundary conditions based on the theory of characteristics for the locally one-dimensional problem are used [Ott, 1991]. Unsteady calculations are performed with the same code using locally non-reflecting boundary conditions [Chakravarthy, 1983]. No investigations have been made, if these boundary conditions allow small reflections or are really highly non-reflecting.

![Diagram of a two-dimensional cascade](image)

Figure 1.4: Two dimensional cascade according [Dorney and Verdon, 1994]

The blade flutter phenomena is simulated by imposing a motion on the blade, which consists of harmonic body translation in two directions and rotation, allowing an interblade phase angle between two neighboring blades. The studied test cases apply either a bending or a torsional motion. No combined motion, i.e. bending and torsional motion, has been investigated. However, the present method allows to do so. The mechanical system is assumed to be very stiff, i.e. the rigid blade motion is imposed with the eigen-frequencies of the structural system.

In the first step the solution for the steady state flow is calculated, which is then used as initial condition for the unsteady analysis, in which the Navier-Stokes equations are solved in a time accurate manner. In order to simulate the blade motion the computational mesh has to be deformed. To account for this mesh motion extra terms arise in the governing equations. The simulation of non-zero interblade phase angles is handled by storing the
computed flow variables at the periodic boundary nodes and applying the stored quantities at a later time on the opposite periodic boundary [Erdos et al., 1977]. In this way the computational domain only has to include one blade passage. Alternatively, multiple blade passages can be used.

The present method is validated by flat plate cases for laminar and turbulent flow at different Reynolds numbers, laminar flow around a 2-D cylinder, different turbine and a compressor cascade test cases for fully turbulent flow for steady state flow conditions.

Thereafter, emphasize is made on the investigation/validation of the unsteady inviscid and viscous flow situations in cascaded flow for three standard configurations (see [Böllcs and Fransson, 1986]), i.e. a turbine and two compressor test cases, and is performed on the parallel computer IBM/SP2 at KTH. In particular, flow situations are studied where viscous effects are important, namely the viscous induced separation (standard configuration 5) and shock-boundary layer interaction (standard configuration 10), as well as the new standard configuration 11 [Fransson et al., 1998], which is a transonic turbine test case.
Chapter 2

Governing Equations

The program INSTHPT [Ott, 1991; Hambraeus, 1995] is based on the Euler equations for ideal gases. In this work the given program is changed with respect to the overall numerical scheme for the Euler equations. Moreover, the code is extended to viscous turbulent flow as well as several new time integration schemes and boundary conditions are implemented in the present method in comparison with former existing code versions of INSTHPT.

2.1 Differential form

The Navier-Stokes equations (NSE) are derived with the following assumptions:

- 2 D
- compressible flow
- unsteady
- without gravitational forces
- Newtonian liquid (Cauchy-Poisson equation)
- with Stokes’ hypothesis
- ideal gas
- adiabatic flow conditions

The equation for the conservation of mass, momentum and energy in non-dimensional form reads:

\[
\frac{\partial \tilde{U}}{\partial t} + \frac{\partial \tilde{F}}{\partial x} + \frac{\partial \tilde{G}}{\partial y} = \frac{\rho_{-\infty} q_{-\infty}}{Re_{-\infty}} \left( \frac{\partial \tilde{R}}{\partial x} + \frac{\partial \tilde{S}}{\partial y} \right)
\]

\(^1\)For convenience, all following equations are given in non-dimensional form without any special notation.
2.1. DIFFERENTIAL FORM

with

\[
\vec{U} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho e \end{pmatrix}, \quad \vec{F} = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho u H \end{pmatrix},
\]

\[
\vec{G} = \begin{pmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ \rho v H \end{pmatrix},
\]

\[
\vec{R} = \begin{pmatrix} 0 \\ t_{xx} \\ t_{xy} \\ u t_{xx} + v t_{xy} + \frac{\mu \partial h}{Pr \partial x} \end{pmatrix},
\]

\[
\vec{S} = \begin{pmatrix} 0 \\ t_{xy} \\ t_{yy} \\ u t_{xy} + v t_{yy} + \frac{\mu \partial h}{Pr \partial y} \end{pmatrix}
\]

(2.1)

using the following reference constants:

\[
p_{ref}; u_{ref} = v_{ref} = q_{ref} = \sqrt{RT_{ref}}; x_{ref} = y_{ref};
\]

\[
t_{ref} = \frac{x_{ref}}{u_{ref}}; T_{ref}; e_{ref} = h_{ref} = q_{ref}^2;
\]

\[
\mu_{ref}; \rho_{ref} = \frac{p_{ref}}{RT_{ref}}
\]

(2.2)

Note that \( q_{ref} = \frac{a_{ref}}{\sqrt{\gamma}} \), where \( a \) denotes the speed of sound and \( \gamma \) denotes the ratio of the constant specific heats \( c_p \) and \( c_v \). The Reynolds number is based on chord length and inlet flow conditions \(-\infty\), see appendix H. The quantities \( t_{xx}, t_{xy} \) and \( t_{yy} \) denote the shear stress tensor and are defined as:

\[
t_{xx} = \mu \left( \frac{4 \partial u}{3 \partial x} - \frac{2 \partial v}{3 \partial y} \right)
\]

\[
t_{xy} = t_{yx} = \mu' \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)
\]

\[
t_{yy} = \mu \left( \frac{4 \partial v}{3 \partial y} - \frac{2 \partial u}{3 \partial x} \right)
\]

(2.3) - (2.5)

The non-dimensional dynamic viscosity \( \mu' \) in the equations (2.3) - (2.5) is derived in appendix H.
2.2 Transformation of the governing equations

If new independent variables are introduced, i.e. a boundary - fitted moving coordinate system, a strong conservation law form can be maintained as shown, for example by Viviand [1974]. Subject to the general transformation \( \xi = \xi(x,y,t), \eta = \eta(x,y,t), \tau = t \) the equation (2.1) can be written as

\[
\frac{\partial \tilde{U}}{\partial \tau} + \frac{\partial \tilde{F}}{\partial \xi} + \frac{\partial \tilde{G}}{\partial \eta} = \frac{\rho^*}{Re} \frac{\partial^2 \tilde{F} + \partial^2 \tilde{G}}{\rho^*} \left( \frac{\partial (y_n \tilde{R} - x_n \tilde{S})}{\partial \eta} \right) + \left( \frac{\partial (x_\xi \tilde{R} + x_\xi \tilde{S})}{\partial \eta} \right)
\]

(2.6)

with

\[
\tilde{U} = \frac{\tilde{U}}{J},
\]

\[
\tilde{F} = \left( (-x_\eta y_n + y_\eta x_n) \tilde{U} + y_n \tilde{F} - x_n \tilde{G} \right),
\]

\[
\tilde{G} = \left( (-x_\xi y_\xi - y_\xi x_\xi) \tilde{U} - y_\xi \tilde{F} + x_\xi \tilde{G} \right),
\]

(2.7)

Here \( J \) is the transformation Jacobian

\[
J = \frac{1}{(x_\xi y_n - x_\eta y_\xi)}
\]

(2.8)

2.2.1 Thin layer approximation

Frequently, one assumes that the boundary layer is thin in comparison with the length of the profile. This is valid if the pressure gradient in flow direction is small or vanishes.

\[
\frac{\partial p}{\partial \xi} = 0
\]

(2.9)

Then, one can neglect the derivatives in the \( \xi \)-direction of the computational domain:

\[
\frac{\partial}{\partial \xi} = 0
\]

(2.10)

It is possible to switch from the 'full' Navier-Stokes-equations to the thin layer approximation in the present method. This option can be used for flat plate computations with zero pressure gradient in the streamwise direction.
2.3 Algebraic turbulence model

The extension to the turbulent flow is done by the implementation of the algebraic turbulence model of Baldwin and Lomax [1978].

The Baldwin and Lomax algebraic turbulence model, patterned after the two-layer eddy viscosity model of Cebeci [1970], is used for the Reynolds averaged Navier-Stokes equations. The effects of turbulence are simulated in terms of an eddy viscosity coefficient $\mu_t$. Thus, in stress terms of the laminar Navier-Stokes equations, the molecular coefficient of viscosity $\mu$ is replaced by $\mu + \mu_t$. In heat flux the term $\mu/Pr$ is replaced by $\mu/Pr + \mu_t/Pr_t$.

The eddy viscosity is determined from:

$$\mu_t' = \begin{cases} 
\mu_t'_{inner} & \text{for } 0 \leq Y \leq Y_c \\
\mu_t'_{outer} & \text{for } Y_c \leq Y 
\end{cases} \tag{2.11}$$

In the program this is accomplished by the following formulation, see e.g. [Cebeci and Bradshaw, 1977]:

$$\mu_t' = \min \left\{ \mu_t'_{inner}, \mu_t'_{outer} \right\} \tag{2.12}$$

$Y$ denotes the normal distance from the wall or wake centerline. $Y_c$ is the crossover distance, i.e. the smallest value of $Y$ for which the values of the inner and the outer formulas are equal. The Prandtl-van Driest formulation [Baldwin and Lomax, 1978] is used for the inner region:

$$\mu_t'_{inner} = \frac{Re_{-\infty}}{\nu_{-\infty}^2} \rho \lambda^2 |\Omega| \tag{2.13}$$

with the mixing length:

$$\lambda = \kappa Y \left\{ 1 - \exp \left( \frac{Y^+}{4} \right) \right\} \tag{2.14}$$

$|\Omega|$ is the magnitude of the vorticity with

$$|\Omega| = \sqrt{\left( \frac{\partial u}{\partial y} - \frac{\partial v}{\partial x} \right)^2} \tag{2.15}$$

and the law-of-the-wall co-ordinate

$$Y^+ = \sqrt{\frac{Re_{-\infty}}{\nu_{-\infty}^2 \rho_{-\infty} \mu_t}} \left[ \frac{\partial u}{\partial y} \right]_{W, Y} \tag{2.16}$$

The Clauser formulation [Baldwin and Lomax, 1978] is modified in the outer region according to
\[ \mu'_{outer} = \frac{Re_{-\infty}}{q_{-\infty}} KC_{CP} \rho F_{Wake} F_{Kleb}(Y) \]  

(2.17)

where \( K \) is the Clauser constant, \( C_{CP} \) is an additional constant, and

\[ F_{Wake} = \min \left\{ \frac{Y_{Max} F_{Max}}{C_{WK} Y_{Max} u_{Diff}^2 / F_{Max}} \right\} \]  

(2.18)

The quantities \( Y_{Max} \) and \( F_{Max} \) are determined from the function

\[ F(Y) = Y |\Omega| \left\{ 1 - \exp \left( \frac{Y}{\lambda^+} \right) \right\} \]  

(2.19)

In wakes, the exponential term in the equation above is set equal to zero. The quantity \( F_{Max} \) is the maximum value of \( F(Y) \) that occurs in a velocity profile and \( Y_{Max} \) is the value of \( Y \) at which \( F_{Max} \) occurs. The function \( F_{Kleb}(Y) \) is the Klebanoff intermittency factor [Baldwin and Lomax, 1978] given by

\[ F_{Kleb}(Y) = \left[ 1 + 5.5 \left( \frac{C_{Kleb} Y}{Y_{Max}} \right)^6 \right]^{-1} \]  

(2.20)

The quantity \( u_{Diff} \) is the difference between maximum and minimum total velocity in the velocity profile (i.e. at a fixed x-station):

\[ u_{Diff} = \left( \sqrt{u^2 + v^2} \right)_{Max} - \left( \sqrt{u^2 + v^2} \right)_{Min} \]  

(2.21)

The second term in \( u_{Diff} \) is taken to be zero, except in wakes.

The outer formulation can be used in wakes as well as in attached and separated boundary layers. The effect of transition to turbulence can be simulated by setting \( \mu' \) equal to zero everywhere in velocity profile of the boundary layer for which the maximum tentatively computed value of \( \mu' \) from the foregoing relations is less than a specified value, that is,

\[ \mu' = 0 \quad \text{for} \quad \left( \mu' \right)_{Max} \leq C_{MUTM} \]  

(2.22)

The constants appearing in the foregoing relations have been determined by requiring agreement with the Cebeci [1970] formulation for constant pressure boundary layers at transonic speeds. The values determined are given in table 2.1. These are the standard values used by Baldwin and Lomax [1978].

Note that the Baldwin-Lomax turbulence model can be used for attached flows and weakly separated flows, but is not designed for large separation. A next step can be the replacement of the model by a second order model, for instance the \( k-\epsilon \) turbulence model. The turbulence model is formulated in a generalized approach for curvilinear co-ordinates.

Presently, an onset of the transition point at the leading edge of the particular profile is used in the code. It would be possible to implement a function in order to switch from
2.4. \textit{Initial and boundary conditions}

The type of the Navier-Stokes equations can be characterized as hybrid hyperbolic - parabolic or as incomplete parabolic. Therefore, the equations have to be supplemented by appropriate initial and boundary conditions to define a well-posed initial boundary value problem. Since a well-posedness theory for the Navier-Stokes equations is lacking, the initial and boundary conditions discussed below are more based on physical considerations, and are not guaranteed to be well-posed. They have been proven useful in computations of sub-sonic and transonic flow at moderate to large Reynolds numbers ($O(10^2 - 10^7)$) [Rizzi, 1994].

2.4.1 Initial conditions

The initial condition should be reasonable and compatible with the boundary conditions. Steady state Euler and laminar viscous solutions are generated from an inviscid free stream solution. For the steady state turbulent calculations a not converged laminar viscous solution is used. This is due to the fact that a viscous boundary layer is required around the blade to start with the Baldwin-Lomax turbulence model.

2.4.2 Boundary conditions

The boundary conditions for the calculations are the following:

a) Solid wall

i) No-slip condition

For the viscous calculations the no-slip condition is applied at the blade.

$\vec{u} = \vec{0}$ at the wall (stationary wall)
If the blade is moving the fluid velocity is equal to the blade velocity in the absolute frame of reference.

\[ \vec{u} = \vec{u}_w \text{ at a moving wall} \]

ii) Temperature condition

The adiabatic wall boundary condition is applied at the blade wall, which corresponds to a heat flux which is zero.

\[ \left. \frac{\partial T}{\partial n} \right|_w = 0 \]

which is the same as

\[ \left. \frac{\partial T}{\partial \eta} \right|_w = 0 \]

for orthogonal mesh lines at the wall.

iii) Pressure

Steady State: The pressure is obtained from the wall momentum equation, which can be derived from momentum equation by multiplying with the outer normal unit vector using the no-slip condition [Hirsch, 1990], or from the boundary layer approximation to that, i.e.

\[ \left. \frac{\partial p}{\partial n} \right|_w = 0 \]

This assumption is valid for high Reynolds numbers and if the surface curvature is not too large. For orthogonal meshes \(^1\) the equation can be written as

\[ \left. \frac{\partial p}{\partial \eta} \right|_w = 0. \]

Unsteady Flow: The assumption of a vanishing pressure gradient normal to the wall is approximately satisfied for an unsteady (accelerated) blade if the reduced frequency is small enough which is assumed in the present work.

Once the velocity vector \( \vec{u} \), \( T \) and \( p \) are known, the dependent flow variables can be determined from them by using the equation of state for ideal gas in dimension-less form:

\[ p = \rho T \]

and the equation of the total energy for ideal gas:

\[ \frac{\partial}{\partial \eta} \begin{pmatrix} \nabla d \xi \\ \nabla d \eta \end{pmatrix} \]

\[ = \begin{pmatrix} \nabla d \xi \\ \nabla d \eta \end{pmatrix} \] \tag{2.23}

Thus, the normal derivative of a variable \( a \), e.g. \( a = T \) or \( p \) reads in transformed co-ordinates:

\[ \frac{\partial a}{\partial n} = \nabla d \cdot \nabla da = \frac{\xi_x \eta_x + \xi_y \eta_y \partial a}{\sqrt{\eta_x^2 + \eta_y^2}} + \sqrt{\eta_x^2 + \eta_y^2} \partial a \eta \right|_w = 0 \]

\[ \left. \frac{\partial a}{\partial \eta} \right|_w = 0 \]

\[ \left. \frac{\partial a}{\partial \eta} \right|_w = 0 \]

If the mesh is orthogonal, equation (2.24) simplifies to

\[ \frac{\partial a}{\partial n} = \frac{\nabla d \eta \partial a}{\nabla d \eta} \]

\[ \frac{\partial a}{\partial \eta} \]

\[ \frac{\partial a}{\partial \eta} \]

due to \( \xi_x \eta_x + \xi_y \eta_y = J^2 (\sigma_x \sigma_y + \gamma \partial u \partial \xi) = 0 \)
\[
\frac{1}{(\gamma-1)} \rho \frac{D}{Dt} \beta + \frac{q^2}{2} = e
\]

The code is suited to orthogonal meshes at the blade due to the fact that the above mentioned simplified boundary conditions for the pressure and temperature are used.

b) Farfield, steady state

At the farfield boundary, viscous effects are assumed to be negligible. Therefore, the same boundary conditions as for the Euler equations can be used [Rizzi, 1994]. These are based on the theory of characteristics for the locally one-dimensional problem normal to the boundary. The signal propagation from the interior to the boundary is modeled by spatial extrapolation. The ingoing information is set by the derivation of the characteristic variables, which are calculated from the independent variables of the Navier-Stokes equations. The flow variables to be given from outside or inside the region of interest can be the characteristic variables or the locally 1D Riemann invariants. Basically, the steady state boundary conditions are used from the code given by the École Fédérale Polytechnique de Lausanne [Ott, 1991], which are briefly discussed below.

Farfield, steady state, inlet boundary condition  From the theory of characteristics it is known that the number of variables which can be set freely at any boundary is equal to the number of characteristics entering through the boundary. For 2D subsonic flow there are 3 incoming characteristics, whereas one goes out of the flow field. Therefore, three variables are set at the inlet and the fourth one is determined from the interior flow field. In INSTHPT the stagnation pressure, the stagnation density and the inlet flow angle are fixed. In order to obtain the fourth value the flow is locally treated in a 1-D manner along each grid line going in the computational domain. Along these grid-lines the outgoing Riemann invariant \( R^- \) is studied. For the 1D problem in space the three characteristics are

\[
\begin{align*}
C_0 : \frac{dx}{dt} & = u \\
C_+ : \frac{dx}{dt} & = u + a \\
C_- : \frac{dx}{dt} & = u - a
\end{align*}
\]  

For isentropic flows, on the characteristics \( C_+ \) and \( C_- \) two independent variables can be derived [Hirsch, 1990]:

\[
\begin{align*}
C_1 = 2H_0 &= \frac{2}{\gamma - 1} a^2 + q^2 = \frac{2}{\gamma - 1} \frac{\rho_0}{P_0} \\
C_2 = R^- &= \frac{2}{\gamma - 1} a - q
\end{align*}
\]

\footnote{At the farfield boundary the inviscid propagation of the pressure, vorticity and entropy waves are more dominant than the diffusive, viscous properties of the flow field, which are a local phenomenon.}
Furthermore, the reference entropy which propagates along the $C_0$ characteristic is set to zero at the entrance in the channel.

Solving the system (2.27) for the speed of sound $a$, one obtains a quadratic equation for $a$:

$$a^2 - 2\frac{\gamma - 1}{\gamma + 1}R^- a + \frac{(\gamma - 1)^2}{2(\gamma + 1)}(R^2 - 2H_0) = 0$$  \hspace{1cm} (2.28)

The two roots for equation (2.28) are:

$$a_{1,2} = \frac{\gamma - 1}{\gamma + 1}R^- \pm \sqrt{\frac{1}{2} \frac{(\gamma - 1)^3}{(\gamma + 1)^2} R^- R + \frac{(\gamma - 1)^2}{(\gamma + 1)} H_0}$$  \hspace{1cm} (2.29)

For the calculations the largest, positive root is chosen [Ott, 1991; Fransson, 1986]. From equation (2.27) follows the amplitude of the velocity and $u$ and $v$ is found to be:

$$u = q \cos (\alpha_1) \quad v = q \sin (\alpha_1)$$  \hspace{1cm} (2.30)

The density is found to be

$$\rho = \left(\frac{a^2}{\gamma}\right)^{\frac{1}{\gamma - 1}}$$  \hspace{1cm} (2.31)

using the isentropic relation for an ideal gas, the relation for the speed of sound and the assumption that $p_0 = 1$ and $\rho_0 = 1$.

**Farfield, steady state, outlet boundary condition**  At the subsonic outlet three values have to be determined from the interior and only one value is prescribed, i.e. the static pressure at the outlet.

The three variables taken from the inlet are the flow angle $\beta_1$, the density $\rho$ and the Riemann invariant $R^+$ given by

$$R^+ = \frac{2}{\gamma - 1} a + q$$  \hspace{1cm} (2.32)

The flow angle and density are extrapolated with zero order accuracy, whereas the Riemann invariant $R^+$ is taken to be constant along the right running characteristic $C_+ = \frac{dx}{dr} = u + a$ in a 1D manner along each outrunning grid line.

For the length of the velocity vector $|\vec{u}| = q = \sqrt{u^2 + v^2}$ it follows:

$$q_2 = R^+ - \frac{2}{\gamma - 1} a_2$$  \hspace{1cm} (2.33)
2.4. INITIAL AND BOUNDARY CONDITIONS

The speed of sound at the exit \( a_2 \) is calculated from the extrapolated density and the prescribed static pressure. Thereafter, the velocity components are obtained with equation (2.33) and the extrapolated flow angle.

c) Farfield, unsteady

For an implicit time integrating scheme the boundary condition should be given in the same form as the governing equations, which reads for inviscid flow (inviscid part of equation (2.6):

\[
\frac{\partial \tilde{U}}{\partial \tau} + L \left( \frac{\partial \tilde{F}}{\partial \xi} + \frac{\partial \tilde{G}}{\partial \eta} \right) = 0
\]  

(2.34)

The difference to equation (2.6) is the influence matrix \( L \) [Chakravarthy, 1983], which is the identity matrix for the field, but includes the boundary conditions at the boundary of the computational domain.

Equation (2.34) has to be linearized in time and can be finally written in the following form at the boundary

\[
\left( I + \Delta \tau \left( L^n \delta_\xi \left( \tilde{A}^n \right) + L^n \delta_\eta \left( \tilde{B}^n \right) + L^n_F + L^n_G \right) \right) \Delta \tilde{U}^n 
= -\Delta \tau \left( L^n \delta_\xi \left( \tilde{F}^n \right) + L^n \delta_\eta \left( \tilde{G}^n \right) \right)
\]  

(2.35)

with

\[
L^n_F = \frac{\partial L^n}{\partial \tilde{U}} \frac{\partial \tilde{F}}{\partial \xi}, \quad L^n_G = \frac{\partial L^n}{\partial \tilde{U}} \frac{\partial \tilde{G}}{\partial \eta}.
\]  

(2.36)

\( \Delta \tilde{U}^n \) denotes the change of the conservative variables from the present to the next time level \( n + 1 \). Formally the matrices \( L^n_G \) and \( L^n_F \) arise by the linearization in time. Since these matrices are very complicated the matrices are skipped and the influence matrix is used at time level \( n \).

The solution of equation (2.35) becomes much easier if it is written in factorized form [Beam and Warming, 1978]:

\[
\left( I + \Delta \tau \left( L^n \delta_\xi \left( \tilde{A}^n \right) + L^n_F \right) \right) \left( I + \Delta \tau \left( L^n \delta_\eta \left( \tilde{B}^n \right) + L^n_G \right) \right) \Delta \tilde{U}^n 
= -\Delta \tau \left( L^n \delta_\xi \left( \tilde{F}^n \right) + L^n \delta_\eta \left( \tilde{G}^n \right) \right) + O \left( \Delta \tau^3 \right)
\]  

(2.37)

The matrices \( \tilde{A}^n \), \( \tilde{B}^n \) denote the Jacobians for the unsplitted fluxes \( \tilde{F}^n \), \( \tilde{G}^n \) in generalized coordinates.

\[
\tilde{A} = \frac{\partial \tilde{F}}{\partial \tilde{U}} = J \frac{\partial \tilde{F}_i}{\partial \tilde{U}^k} \frac{\partial \tilde{U}^k}{\partial u_k}.
\]  

(2.38)
with

\[
\bar{F} = \xi_i \bar{U} + y_i \bar{F} - x_n \bar{G} \tag{2.39}
\]

\[
\bar{G} = -\eta_i \bar{U} - y \xi \bar{F} + x \xi \bar{G} \tag{2.40}
\]

A similar expression is valid for \( \bar{B} \).

Equation (2.37) can be solved in two steps as explained in chapter 3 for the field algorithm \[\text{Beam and Warming, 1978}\].

The boundary conditions can be written in the characteristic form of the Euler equations for the case that waves pass perpendicularly at \( \xi = \text{const.} \) as:

\[
S_A \frac{\partial \bar{U}}{\partial t} + \Lambda_A S_A \frac{\partial \bar{U}}{\partial \xi} + S_A \bar{B} \frac{\partial \bar{U}}{\partial \eta} = 0 \tag{2.41}
\]

\( S_A \) denote the left eigenvector matrix for the Jacobian matrix \( \bar{A} \) with the matrix for the eigenvalues \( \Lambda_A \).

The general form of steady and unsteady boundary conditions is given in equation (2.42) as

\[
B_i(\bar{U}) = 0, \ i = 1, p \tag{2.42}
\]

where \( B_i \) denotes the \( i \)'s boundary condition for the four characteristics in two-dimensional flow which yields:

\[
\frac{\partial B_i}{\partial t} = \frac{\partial B_i}{\partial \bar{U}} \frac{\partial \bar{U}}{\partial t} = 0 \tag{2.43}
\]

Finally after some algebra the boundary conditions in equation (2.41) can be written as

\[
\bar{S}_{A_i} \frac{\partial \bar{U}}{\partial t} + \Lambda_{A_i} \bar{S}_{A_i} \frac{\partial \bar{U}}{\partial \xi} + \bar{S}_{A_i} \frac{\partial \bar{G}}{\partial \eta} = 0 \quad i = 1, \ldots, m - p \tag{2.44}
\]

\[
\frac{\partial B_i}{\partial \bar{U}} \frac{\partial \bar{U}}{\partial t} = 0 \quad i = m - p + 1, \ldots, m \tag{2.45}
\]

Outrunning characteristics are given by equation (2.44) where incoming waves are calculated with equation (2.45).

For nonreflecting boundary conditions equation (2.45) takes the following form \[\text{Chakravarthy, 1983}\]:

\[
\frac{\partial R_i}{\partial t} = \bar{S}_{A_i} \frac{\partial \bar{U}}{\partial t} = 0 \quad i = m - p + 1, \ldots, m \tag{2.46}
\]
where $R_i$ defines the $i$’s characteristic variable.

It follows

$$\frac{\partial B_i}{\partial U} = S_{A_i}$$  \hspace{1cm} (2.47)

Introducing the vector

$$L_2 = (S_1, ..., S_{m-p}, 0, ..., 0)^T$$  \hspace{1cm} (2.48)

equation (2.44) can be written

$$L_2 \frac{\partial \tilde{U}}{\partial t} + \Lambda_A L_2 \frac{\partial \tilde{G}}{\partial \xi} + L_2 \frac{\partial \tilde{F}}{\partial \eta} = 0, \quad i = 1, ..., m-p$$  \hspace{1cm} (2.49)

with $\Lambda_A L_2 = L_2 \bar{\Lambda}$ and $\bar{\Lambda} \frac{\partial \tilde{U}}{\partial \xi} = \frac{\partial \tilde{F}}{\partial \xi}$

it follows

$$L_2 \frac{\partial \tilde{U}}{\partial t} + L_2 \left( \frac{\partial \tilde{F}}{\partial \xi} + \frac{\partial \tilde{G}}{\partial \eta} \right) = 0.$$  \hspace{1cm} (2.50)

Using

$$L_1 = L_2 + \left( 0, ..., \partial B_{m-p+1}/\partial \tilde{U}, ..., \partial B_{m}/\partial \tilde{U} \right)^T$$

$$= \left( \bar{S}_1, ..., \bar{S}_{m-p}, \partial B_{m-p+1}/\partial \tilde{U}, ..., \partial B_{m}/\partial \tilde{U} \right)^T$$  \hspace{1cm} (2.51)

Equation (2.50) reads

$$L_1 \frac{\partial \tilde{U}}{\partial t} + L_2 \left( \frac{\partial \tilde{F}}{\partial \xi} + \frac{\partial \tilde{G}}{\partial \eta} \right) = 0$$  \hspace{1cm} (2.52)

and

$$\frac{\partial \tilde{U}}{\partial t} + L \left( \frac{\partial \tilde{F}}{\partial \xi} + \frac{\partial \tilde{G}}{\partial \eta} \right) = 0$$  \hspace{1cm} (2.53)

with

$L = L_1^{-1} L_2$

For purely nonreflecting boundary conditions $L_1 = S$ and $L = S^{-1} L_2$. $S$ denotes the matrix for the right (column-)eigenvectors of the Jacobian matrix $\bar{\Lambda}$. 
In the matrix $L_2$ only these eigenvectors are considered which are connected to outrunning characteristics. Consequently, at the inlet of the computational domain one eigenvector is considered and at the outlet three eigenvectors are considered. The other eigenvectors are set equal to zero for axially subsonic flow.

d) Gust boundary conditions

Gust boundary conditions are developed for the explicit version of the code INSTHPT. So far these boundary conditions have been applied to the phenomenon of forced response, see appendix E. However, the application of these boundary conditions towards flutter is straightforward. In a future work it should be investigated how neighboring blade rows simulated as gust boundary conditions effect the stability of oscillating blade rows with respect to flutter.

e) Periodic boundary condition

In order to calculate a cascade flow, two different periodic boundary conditions are used. The time lagged periodic boundary condition or direct store method was used by Erdos et al. [1977] for the first time. Then it is possible to use one blade passage. An alternative concept is the use of multi-blade passages, see e.g. Grüber and Carstens [1996].

Steady state  The steady state boundary condition reads:

\[ \tilde{U}(x, y) = \tilde{U}(x, y + P) \]  \hspace{1cm} (2.54)

where $P$ denotes the pitch of the cascade. Only one passage is needed for steady state calculations.

The standard method to accomplish this is to introduce dummy cells at the lower and the upper boundary of the inlet and outlet duct of the computational domain.

The values of the flow variables in the lowest two cells in the flow channel of the computational domain are used in the dummy cells outside the upper boundary of the channel and vice versa. Two rows of dummy cells are needed for the viscous discretisation.

Unsteady flow  All blades are assumed to oscillate in the same mode and motion and only differ in phase by a constant interblade phase angle between adjacent blades.

For INSTHPT the unsteady flow periodicity can be written as

\[ \tilde{U}(x, y, t) = \tilde{U}(x, y + P, t + \Delta T) \]  \hspace{1cm} (2.55)

with $\Delta T = \sigma/360^\circ T$, $T$ the period length of the prescribed blade motion and $\sigma$ the interblade phase angle.

Direct store method
Computationally, the direct store method of Gerolymos [1988] and Erdos et al. [1977] is used for the explicit unsteady code. The computed flow variables at the upper periodic boundary are stored in an array and applied at a later time, corresponding to the phase shift \( \sigma \) or \( \Delta T \) at the lower periodic boundary. In the same manner, the flow variables on the lower boundary are stored and applied at a later time, corresponding to \( 2\pi \) minus the phase shift on the upper periodic boundary:

\[
\vec{U}_{\text{lower dummy}}(t) = \vec{U}_{\text{upper interior}}(t - \Delta T) \\
\vec{U}_{\text{upper dummy}}(t) = \vec{U}_{\text{lower interior}}(t - (T - \Delta T))
\]  

(2.56)

**Multiple blade passages**

The fully implicit formulation of the code requires multiple blade passages.

The condition for the periodic boundary reads then:

\[
\vec{U}(x, y, t) = \vec{U}(x, y + n_p P, t)
\]  

(2.57)

where \( n_p \) denotes the number of passages.

Equation (2.56) changes to

\[
\vec{U}_{\text{lower dummy}}(t) = \vec{U}_{\text{upper interior}}(t) \\
\vec{U}_{\text{upper dummy}}(t) = \vec{U}_{\text{lower interior}}(t)
\]  

(2.58)
Chapter 3

Numerical Discretisation

3.1 Discretisation of the convective terms

Flux vector splitting schemes such as the Van Leer splitting [Van Leer, 1982] have been proven to be a simple and useful technique for arriving at upwind differencing. Unfortunately, the simplicity comes at a price of reduced accuracy due to the numerical diffusion. As a result, significant errors appear in the viscous region. One possibility to improve the modeling of boundary layers is the Advection Upstream Splitting Method (AUSM), see Wada and Liou [1994].

The main step is to recognize that the inviscid flux vector, for instance $\vec{F}$, consists of two physically distinct parts, namely the convective and the pressure term:

$$\vec{F} = u \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho H \end{pmatrix} + \begin{pmatrix} p \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

(3.1)

In a first step the vector $\vec{U}$ is evaluated at the cell face. Depending on the propagation, the independent flow variables are discretised in the following way, for instance at the cell-face $(i+1/2, j)$ (see figure 3.1) (MUSCL-Approach, (see Anderson et al. [1987])):

$$\vec{U}_{i+1/2,j}^+ = \vec{U}_{i,j} + \Phi \frac{\vec{U}_{i,j} - \vec{U}_{i-1,j}}{2}$$

$$\vec{U}_{i+1/2,j}^- = \vec{U}_{i+1,j} + \Phi \frac{\vec{U}_{i+2,j} - \vec{U}_{i+1,j}}{2}$$

(3.2)

$\Phi$ denotes a "Flux-Limiter" which is switched from 1 to 0 where a shock occurs, which leads to 2nd or 1st order accuracy, respectively. Thereafter, one has to calculate the fluxes $\vec{F}^+$ and $\vec{F}^-$ at the cell faces. Therefore, one can write the $\vec{F}$-flux, for example, in the following way:
\[ \bar{F}_{1/2} = \bar{F}^{(+/−)} = \frac{1}{2} \left( \left( \rho \bar{u}^{\xi} \right)_{1/2} \left( \Psi_L + \Psi_R \right) - \left( \rho \bar{u}^{\xi} \right)_{1/2} \left( \Psi_R - \Psi_L \right) \right) + p_{1/2} \left( \begin{array}{c}
\frac{y_{\eta}}{y_{\eta}} \\
-x_{\eta} \\
-x_{\xi}
\end{array} \right) \] (3.4)

with \( \Psi^T = (1, u, v, H) \).

In the present method the so-called AUSMD/V (see [Wada and Liou, 1994]) splitting is used, it reads

\[ \left( \rho \bar{u}^{\xi} \right)_{1/2} = \bar{u}^{\xi+}_{L} \rho_{L} + \bar{u}^{\xi−}_{R} \rho_{R} \] (3.5)

for the mass flux.

The only difference between AUSMV and AUSMD is the splitting of the x-momentum flux \( (\rho u^2)_{1/2} \) which reads

\[ \left( \rho u \bar{u}^{\xi} \right)_{1/2} = \bar{u}^{\xi+}_{L} (\rho u)_L + \bar{u}^{\xi−}_{R} (\rho u)_R \] (3.6)

for the AUSMV scheme and

\[ \left( \rho u \bar{u}^{\xi} \right)_{1/2} = \left( \rho \bar{u}^{\xi} \right)_{1/2} \bar{u}^{\xi+}_{L} (\rho u_L + u_R) + \text{abs} \left( \left( \rho \bar{u}^{\xi} \right)_{1/2} \right) \bar{u}^{\xi−}_{R} (\rho u_L - u_R) \] (3.7)

for the AUSMD scheme. The results presented in the thesis are obtained with the AUSMD scheme.

The splittings for \( \bar{u}^{\xi+}_{L} \) and \( \bar{u}^{\xi−}_{R} \) are given as

\[
\bar{u}^{\xi+}_{L} = \begin{cases} 
\alpha_L \left( \frac{(\bar{u}^{\xi+}_{L} + c_m)^2}{4c_m} - \frac{\bar{u}^{\xi+}_{L} + |\bar{u}^{\xi+}_{L}|}{2} \right) + \\
\frac{\bar{u}^{\xi+}_{L} + |\bar{u}^{\xi+}_{L}|}{2}, & \text{if } \frac{\bar{u}^{\xi+}_{L}}{c_m} \leq 1 \\
\frac{\bar{u}^{\xi+}_{L} + |\bar{u}^{\xi+}_{L}|}{2}, & \text{otherwise}
\end{cases}
\] (3.8)

\[
\bar{u}^{\xi−}_{R} = \begin{cases} 
\alpha_R \left( \frac{(\bar{u}^{\xi−}_{R} - c_m)^2}{4c_m} - \frac{\bar{u}^{\xi−}_{R} - |\bar{u}^{\xi−}_{R}|}{2} \right) + \\
\frac{\bar{u}^{\xi−}_{R} - |\bar{u}^{\xi−}_{R}|}{2}, & \text{if } \frac{\bar{u}^{\xi−}_{R}}{c_m} \leq 1 \\
\frac{\bar{u}^{\xi−}_{R} - |\bar{u}^{\xi−}_{R}|}{2}, & \text{otherwise}
\end{cases}
\] (3.9)
with

\[ \alpha_L = \frac{2 (p/\rho)_L}{(p/\rho)_L + (p/\rho)_R} \]  

(3.10)

\[ \alpha_R = \frac{2 (p/\rho)_R}{(p/\rho)_R + (p/\rho)_L} \]  

(3.11)

and \( c_m = \max(a_L, a_R) \). Secondly, the pressure flux is

\[ p_{1/2} = p^+_L + p^-_R \]  

(3.12)

where

\[ p^\pm_{L,R} = \begin{cases} 
\frac{1}{4} p_{L,R} \left( \frac{\hat{p}^\pm_{L,R}}{c_m} \pm 1 \right) \left( 2 \mp \frac{\hat{p}^\pm_{L,R}}{2 c_m} \right), & if \frac{\hat{p}^\pm_{L,R}}{c_m} \leq 1 \\
p_{L,R} \frac{\hat{p}^\pm_{L,R}}{2 c_m}, & otherwise \end{cases} \]  

(3.13)

The discretisation of the convective terms in the \( \eta \)-direction can be found in appendix G.

3.1.1 Van Leer’s Flux Vector Splitting Scheme for viscous flow

Hänel and Schwane [1989] introduced a modification of the scheme of van Leer [1982]. This is done due to the bad accuracy of the scheme in boundary layers, which is also found by Hölm [1995 a]. The main reason for that is the splitting error of the tangential momentum equation in the original van Leer flux vector splitting scheme. In boundary layer flows the tangential velocity \( u \) grows from zero at the wall to its large outer value \( u_e \) over a short distance \( \delta \), whereby the normal velocity is small. The flux component \( \rho u v \) normal to the wall of the tangential momentum equation is splitted according van Leer’s scheme in

\[ \rho u v = \left[ \rho a/4 \left( 1 + \frac{v}{a} \right)^2 u \right]^+ - \left[ \rho a/4 \left( 1 - \frac{v}{a} \right)^2 u \right]^\pm \]  

(3.14)

Assuming \( \rho a = \text{const.} \) and \( v/a \ll 1 \) an error remains

\[ \rho u v \rightarrow \rho a/4 \left[ u^+ - u^- \right] \]  

(3.15)

Since the curvature of \( u \) especially for high Reynolds numbers is large, this error is found to be responsible for the strong smearing using flux vector splitting in viscous layers. The error can be completely removed if the tangential velocity \( u \) extrapolated
from one direction only. 1 This idea was initiated by H"anel and was further developed by Wada and Liou [1994].

3.1.2 AUSM - Mach number splitting compared to AUSM V/D

Different splittings are possible, Kroll and Radespiel [1993] reported good results for viscous flow using the AUSM-"Mach number splitting":

\[(\rho u)_{1/2} = \frac{1}{2} \left[ M_{1/2} (\rho L a_L + \rho R a_R) - \left| M_{1/2} \right| (\rho R a_R - \rho L a_L) \right] \]  

(3.16)

where

\[ M_{1/2} = \frac{u_L}{a_L} + \frac{u_R}{a_R} \]  

(3.17)

which can be easily transformed to generalized coordinates. The reason why the AUSMD/V - scheme is used is the better prediction of shock phenomena like the carbuncle phenomenon [Wada and Liou, 1994]. The carbuncle phenomenon denotes a numerically induced instability in the vicinity of shocks.

3.2 Discretisation of the diffusive terms

The extension of the given code to viscous flow is accomplished by using an idea of H"anel et al. [1987]. According to this paper the viscous terms are centrally discretised without any artificial viscosity.

The first derivatives are approximated for example for the middle of the left cell face of the computational cell (see figure 3.1) as shown in the equations (3.18) to (3.21) and which can be found for example in Peyret and Taylor [1986].

\[ \delta u_{\xi_{i+1/2,j}} = \frac{\partial u}{\partial \xi} \bigg|_{i+\frac{1}{2},j} = \frac{u_{i,j} - u_{i-1,j}}{\Delta \xi} + O \left( \Delta \xi^2 \right) \]  

(3.18)

\[ \delta u_{\eta_{i+1/2,j}} = \frac{\partial u}{\partial \eta} \bigg|_{i+\frac{1}{2},j} = \frac{1}{4} \left( u_{i,j+1} + u_{i-1,j+1} - (u_{i,j-1} + u_{i-1,j-1}) \right) \frac{1}{\Delta \eta} + O \left( \Delta \eta^2 \right) \]  

(3.19)

\[ \delta u_{\eta_{i,j-1/2}} = \frac{\partial u}{\partial \eta} \bigg|_{i,j-\frac{1}{2}} = \frac{u_{i,j} - u_{i,j-1}}{\Delta \eta} + O \left( \Delta \eta^2 \right) \]  

(3.20)

1Alternatively to equation (3.15), equation (3.14) can be rewritten as \((\rho u)_{1/2} = \frac{1}{4} \left[ M_{1/2} (\rho L a_L + \rho R a_R) - (M_{1/2}^2 - M_{1/2}^2) (\rho R a_R - \rho L a_L) \right]. With \((M_L^2 - M_R^2) = \frac{1}{2} + \frac{1}{4} (M_L - M_R) + \frac{1}{4} (M_L^2 - M_R^2)\) follows that terms remain in the boundary layer where \(M_L, M_R\) approaches zero in contrast to the AUSM.
\[ \delta u_{\xi_{i,j-1/2}} = \left. \frac{\partial u}{\partial \xi} \right|_{\xi_{i,j-1/2}} = \frac{1}{4} \left( u_{i+1,j} + u_{i+1,j-1} - (u_{i-1,j} + u_{i-1,j-1}) \right) \frac{1}{\Delta \xi} + O \left( \Delta \xi^2 \right) \] (3.21)

The velocity component in the y-direction \( v \) and the enthalpy \( h \) is discretised in the same way.

\[
\begin{array}{ccc}
  +i-1,j+1 & +i,j+1 & +i+1,j+1 \\
  i,j+1 & i+1,j+1 & \\
  +i-1,j & +i,j & +i+1,j \\
  i,j & i+1,j & \\
  +i-1,j-1 & +i,j-1 & +i+1,j-1 \\
\end{array}
\]

Figure 3.1: Computational cell

The coordinates are stored in the cell edges and the flow variables are stored in the cell center.

The metric terms are computed according to the equations (3.22) to (3.25) and the equations (3.26) to (3.29).

\[ \frac{\partial \xi}{\partial x} = y_n J \] (3.22)

\[ \frac{\partial \xi}{\partial y} = -x_n J \] (3.23)

\[ \frac{\partial \eta}{\partial x} = -y_n J \] (3.24)

\[ \frac{\partial \eta}{\partial y} = x_n J \] (3.25)

\[ x_{\xi,1/2,j} = x_{i+1,j} - x_{i,j} \] (3.26)
\[ y_{i,j}^{n} = y_{i,j}^{n-1} - \Delta \xi \]

\[ x_{i,j}^{n} = x_{i,j}^{n-1} \]

\[ y_{i,j}^{n} = y_{i,j}^{n-1} - \Delta \eta \]

In the equations (3.26) to (3.29) it is assumed that \( \Delta \xi = \Delta \eta = 1 \) in the computational domain.

The time derivatives are computed according to equation (3.30) to (3.33):

\[ \frac{\partial u}{\partial x} = \frac{\partial u}{\partial \xi} \frac{\partial \xi}{\partial x} + \frac{\partial u}{\partial \eta} \frac{\partial \eta}{\partial x} \]

\[ \frac{\partial u}{\partial y} = \frac{\partial u}{\partial \xi} \frac{\partial \xi}{\partial y} + \frac{\partial u}{\partial \eta} \frac{\partial \eta}{\partial y} \]

The derivatives are built at \( i - \frac{1}{2}, j \) and at \( i, j - \frac{1}{2} \), i.e. the middle of the left cell face and the low cell face respectively. In case the velocities or the metric terms have to be averaged to these points this is accomplished by averaging neighbor points. For example, the Jacobian is calculated for the cell center. So it has to be interpolated for every derivative to the cell face with second order accuracy.

The needed metric terms are either calculated at these points or interpolated. The differencing for the diffusion terms representing shear stress and heat-transfer effects corresponds to a second order accurate central difference in which second derivatives are treated as differences across cell interfaces of first derivative terms, as

\[ \partial \xi S \approx \delta \xi S_{i,j} = \left( S_{i+1/2,j} - S_{i-1/2,j} \right) \]

For example, the term
\[ \tau_{xx} = \frac{2}{3} \mu \left( 2 \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} \right) \]  
(3.37)

Applying the chain rule becomes

\[ \tau_{xx} = \frac{2}{3} \mu J \left( 2 \left( \frac{\partial u}{\partial \xi} y_\eta - \frac{\partial u}{\partial \eta} y_\xi \right) - \left( \frac{\partial v}{\partial \xi} x_\eta + \frac{\partial v}{\partial \eta} x_\xi \right) \right) \]  
(3.38)

And is differenced in \( S_{i+1/2,j} \) as

\[ \tau_{xx} = \frac{2}{3} \mu_{i+1/2,j} J_{i+1/2,j} \left[ 2 \left( \delta_\xi u_{i+1/2,j} y_{n+1/2,j} - \delta_\eta u_{i+1/2,j} y_{i+1/2,j} \right) \right. \]  

\[ \left. - \left( \delta_\xi v_{i+1/2,j} x_{n+1/2,j} + \delta_\eta v_{i+1/2,j} x_{i+1/2,j} \right) \right] \]  
(3.39)

Where derivatives of the velocities are defined as in the equations (3.18) to (3.21).

### 3.3 Mesh movement

The mesh movement is performed by an algebraic interpolation of the mesh lines in axial and tangential direction. The requirements are that the vertical lines at the inlet and outlet are fixed as well as that the motion of the mesh points on the blade coincide with the rigid body motion of the blade surface [Hambraeus 1995]. Enbuske [1996] performed an extensive study on unsteady predictions with the present method for inviscid flow, which showed that the mesh movement works even for high amplitudes. An alternative approach using Fourier coefficients for the harmonic mesh motion of each point was studied but was not found to be superior compared to the algebraic method.

### 3.4 Time discretisation of the 2D Navier-Stokes equations

#### 3.4.1 Explicit time integration

Two integration schemes are available in the code.

- The two step scheme according to Grossman [1986], which is 2nd order accurate in time [Ott, 1991], reads:

\[ \bar{U}^1 = \bar{U}^n - \bar{R} \left( \bar{U}^n \right) \]  
(3.40)

\[ \bar{U}^{n+1} = \frac{1}{2} \left( \bar{U}^n - \bar{R} \left( \bar{U}^1 \right) + \bar{U}^1 \right) \]
A Runge-Kutta 4 stage integration scheme is used, which is second order accurate in time and can be found in Leyland et al. [1994] and reads:

\[
\begin{align*}
\tilde{U}^1 &= \tilde{U}^n - \alpha_1 \tilde{R} (\tilde{U}^n) \\
\tilde{U}^2 &= \tilde{U}^n - \alpha_2 \tilde{R} (\tilde{U}^1) \\
\tilde{U}^3 &= \tilde{U}^n - \alpha_3 \tilde{R} (\tilde{U}^2) \\
\tilde{U}^{n+1} &= \tilde{U}^n - \alpha_4 \tilde{R} (\tilde{U}^3)
\end{align*}
\] (3.41)

with

\[
\tilde{U} = (\rho, \rho u, \rho v, \rho e)^T
\] (3.42)

and

\[
\tilde{R} (\tilde{U}^n) = J \Delta T \left( \frac{\Delta \bar{F}^n}{\Delta \xi} + \frac{\Delta \bar{G}^n}{\Delta \eta} \right) \\
- \left( \frac{\Delta (y_i \bar{R} - x_i \bar{S})^n}{\Delta \xi} + \frac{\Delta (-y_i \bar{R} + x_i \bar{S})^n}{\Delta \eta} \right) \left( \frac{\rho u_{\infty}^2 q_{-\infty}}{R e_{-\infty}} \right)
\] (3.43)

\(\bar{F}\) and \(\bar{G}\) are defined as in chapter 2. The factors for the 4 stage Runge-Kutta scheme are chosen according to Leyland et al. [1994] to

\[\alpha_1 = 0.15, \quad \alpha_2 = 0.20, \quad \alpha_3 = 0.5, \quad \alpha_4 = 1.0\] (3.44)

Parallelization of the numerical method

This section describes the parallelization of the method INSTHPT using the explicit time integration on the parallel super computer IBM SP2 at the parallel computer center (PDC) at KTH. This makes the explicit code feasible for unsteady viscous calculations.

Implementation The parallel program of INSTHPT consists of different executable codes as inferred by the parent with child paradigm in figure 3.2 [Hustad and Jennions, 1995]. The parallelization can be divided into four distinct phases.

First, the run data files are read, i.e. the input file containing the values specified at the boundary of the computational domain, the file with the data to decompose the computational domain and the mesh- and flow data to start with. Thereafter, the mesh and the flow data is decomposed. Presently, the code can be decomposed into 4 up to 16 computational domains. The code is parallelized on a IBM/SP2 using the message passing system MPI. In the second phase each child (processor) receives its input mesh-
and flow data. For phase III the children calculate their local problem while message passing is performed to update the flow variables across the dummy cells around each local sub domain. Finally, in phase IV, the parent receives and reconstructs the flow data to the original format for the full problem to ensure the compatibility with the used post processing routines.

**Domain decomposition** The main criteria for the mesh and flow decomposition is that it should be reasonably fast (compared with the time it takes to do the CFD analysis) and minimizes the number of dummy cells around the local sub-domains so that the communication between the children (processors) does not destroy the gain of cpu-time due to the smaller local problem.

The parallel programming model is the “Master-Worker” - model [Fristedt and Tengwall, 1995]. The programs executed by the master and the worker are different. The master controls the execution of the program whereas the workers perform the computation on smaller domains. The data is static, i.e. the size and the kind of computations doesn’t change during the whole calculations. In this code the one dimensional data distribution [Fristedt and Tengwall, 1995], namely the block distribution is applied.

INSTHPT is divided in subdomains. Therefore, artificial boundaries, i.e. dummy cells must be introduced. Table F.1 in appendix F shows the type of boundary conditions used in the code. According to this value for the different sides of the computational domain the computation of the boundary condition is chosen in the code.

As an example the decomposition for 4 subdomains is shown in figure 3.3. Starting
from the initial problem, the domain in the computational space ($\xi, \eta$ - plane) is divided in subdomains. In figure 3.3 the subdomains are surrounded with thick lines. Each subdomain with a row of two dummy cells around it, is computed on a different node. After each time step the information in the dummy cells is exchanged with the neighboring blocks.

![Diagram of computational subdomains](image)

Figure 3.3: *Computational subdomains*

Following this example the subdomain 1, i.e. the subdomain in the lower left, has different boundaries around it. At the entrance the boundary is of the type of reflecting or non-reflecting boundary conditions. The type number is 4 according to table F.1. The lower boundary is of a mixed type namely periodic and solid wall boundary condition, i.e. type 9 in table F.1. The right boundary condition to the neighbor subdomain 2 and the upper boundary condition to subdomain 3 (dashed lines) are a dummy boundary conditions (type 1). These cells are needed to compute the difference operators in chapter 3 for all cells in the subdomain itself, which is surrounded by the thick line.

Depending on the boundary type determined by table F.1 different subroutines of the FORTRAN code are applied. For instance, the dummy cells at the left boundary condition
are updated after a time step by the first inner two vertical lines of subdomain 2 and four edge cells of subdomain 4. Actually, this is a shift of the flow information from the inner cells of subdomain 2 to the dummy cells of subdomain 1 and vice versa.

**Performance of the parallel version of the numerical method** Figure 3.4 shows the achieved speedup against the ideal one. It can be seen that the achieved speedup is getting lower for larger number of nodes. This is due to the increasing communication between the nodes.

![Graph showing speedup against number of processors](image)

Figure 3.4: *Speed up against the serial code (1 node) as a function of the number of nodes of the parallel code*

**Computational time for particular test cases** The steady state flat plate computation for a Reynolds number of 1 000 000 took about 1 hour on eight nodes on the IBM SP2 to achieve full convergence using local time stepping.

An inviscid unsteady computation conducted on standard configuration 4 running 14 cycles to obtain a periodic unsteady solution took 3 hours on 16 nodes on the IBM SP2.

The viscous unsteady computations on standard configuration 5 with an incidence angle of 4 degrees running 12 cycles, presented in section 4.2 were computed in approximately 5 days on 16 nodes of the IBM SP2.

With the help of figure 3.4 the needed time for a single node can be calculated.

### 3.4.2 Implicit time integration

The used implicit method is first order accurate in time and reads [Beam and Warming, 1978]:

\[2\text{60 MFLOPS theoretical performance per node}\]
\[ (\tilde{U}^{n+1} - \tilde{U}^n) = \Delta \tau \left( \frac{\partial \tilde{U}}{\partial \tau} \right)^{n+1} \]  

(3.45)

Inserting the right hand side of equation (2.6) results in:

\[ (\tilde{U}^{n+1} - \tilde{U}^n) = \Delta \tau \left( -\frac{\partial \tilde{F}}{\partial \xi} - \frac{\partial \tilde{G}}{\partial \eta} + \rho_s^s q_s^s \frac{\partial \tilde{R}}{\partial \xi} + \frac{\partial \tilde{S}}{\partial \eta} \right)^{n+1} \]  

(3.46)

After linearising the flux terms in time and performing a factorisation the implicit scheme follow to:

\[
\begin{align*}
(I + \Delta \tau \frac{\partial}{\partial \xi} \left( \frac{\partial \tilde{F}}{\partial U} \right)^n) - \Delta \tau \rho_s^s q_s^s \frac{\partial \tilde{R}}{\partial \xi} \\
(I + \Delta \tau \frac{\partial}{\partial \eta} \left( \frac{\partial \tilde{G}}{\partial U} \right)^n) - \Delta \tau \rho_s^s q_s^s \frac{\partial \tilde{S}}{\partial \eta} \\
= -\Delta \tau \left( \frac{\partial}{\partial \xi} (\tilde{F})^n + \frac{\partial}{\partial \eta} (\tilde{G})^n \\
- \frac{\rho_s^s q_s^s}{Re_{\infty}} \left( \frac{\partial}{\partial \xi} (\tilde{R})^n + \frac{\partial}{\partial \eta} (\tilde{S})^n \\
+ \frac{\partial}{\partial \xi} \left( \Delta \tilde{R}_2 \right)^n + \frac{\partial}{\partial \eta} \left( \Delta \tilde{S}_1 \right)^n \right) \right) + O (\Delta \tau^3)
\end{align*}
\]  

(3.47)

The product \( \frac{\partial}{\partial \xi} \left( \frac{\partial \tilde{F}}{\partial U} \right)^n \) \((\tilde{U}^{n+1} - \tilde{U}^n)\) is discretised as \( \frac{\partial}{\partial \xi} \left( \frac{\partial \tilde{F}}{\partial U} \right) \tilde{U}^{n+1} \) as shown in equation (3.48). Note that the viscous derivatives in equation (2.6) can be written as \( \tilde{R} = \tilde{R}_1 + \tilde{R}_2 \), where \( \tilde{R}_1 \) contains all derivatives in \( \xi \) and \( \tilde{R}_2 \) is built with the mixed derivatives. On the other hand \( \tilde{S}_1 \) is consisting of the mixed derivatives and \( \tilde{S}_2 \) contains the term in \( \eta \) - direction. The terms \( \left( \Delta \tilde{R}_2 \right)^n \) and \( \left( \Delta \tilde{S}_1 \right)^n \) denote the differences of the viscous terms which contain cross derivatives and cannot be factorised and are lagged in time, i.e. \( \left( \Delta \tilde{R}_2 \right)^n = \tilde{R}_2^n - \tilde{R}_2^{n-1} \).

**Numerical Discretisation**

The discretisation of the code is based on the Advection Upstream Splitting Method [Wada and Liou, 1994].

Applying a linearisation in time based on the extrapolated variables \( \tilde{U}_L \) and \( \tilde{U}_R \) for the inviscid and the viscous flux terms equation (3.46) can be written as

\[
\left( \Delta \tilde{U}^n + \Delta \tau \left( \delta_\xi \left( \tilde{A}_L^n \Delta \tilde{U}_L^n \right) + \delta_\xi \left( \tilde{A}_R^n \Delta \tilde{U}_R^n \right) + \\
- \rho_s^s q_s^s \delta_\xi \left( \tilde{R}_{1L} \Delta \tilde{U}_L^n \right) - \rho_s^s q_s^s \delta_\xi \left( \tilde{R}_{1R} \Delta \tilde{U}_R^n \right) \right) \right)
\]

(3.48)
\[
\begin{align*}
\Delta \tau \left( \delta_\eta \left( \tilde{B}_L^n \Delta \tilde{U}_L^n \right) + \delta_\eta \left( \tilde{B}_R^n \Delta \tilde{U}_R^n \right) \\
- \frac{\rho^* \alpha^* q^*}{Re_{-\infty}} \delta_\eta \left( \tilde{S}_{2L}^n \Delta \tilde{U}_L^n \right) - \frac{\rho^* \alpha^* q^*}{Re_{-\infty}} \delta_\eta \left( \tilde{S}_{2R}^n \Delta \tilde{U}_R^n \right) \right) \\
= - \Delta \tau \left( \delta_\xi \left( \tilde{F}^{n+1} - \tilde{F}_i^n - \tilde{F}_i^n - \left( \tilde{F}_i^n - \tilde{F}_i^{n-1} \right) \right) + \delta_\eta \left( \tilde{G}^{n+1} - \tilde{G}_i^n - \tilde{G}_i^n - \left( \tilde{G}_i^n - \tilde{G}_i^{n-1} \right) \right) \right). \\
(3.48)
\end{align*}
\]

The Jacobian matrices $\tilde{A}$, $\tilde{B}$, $\tilde{R}_1$ and $\tilde{S}_2$ are explained below.

The bar over the vectors on the right hand side of equation (3.48) denotes the vectors in the computational space. The vectors containing cross derivatives in $\xi$ and $\eta$ ($\tilde{R}_2$, $\tilde{S}_1$) of the diffusive terms were transferred already to the right hand side. The reason for this is that these terms cannot be factorised on the left hand side of equation (3.48). The mixed viscous terms are computed from the present and the previous time step (n - 1).

The discretised form of equation (3.48) in factorised form reads:

\[
\begin{align*}
\left( \Delta \tau \left( \left( -\tilde{A}_{i-1/2,j}^n + \frac{\rho^* \alpha^* q^*}{Re_{-\infty}} \tilde{R}_{1,i-1/2,j}^n \right) \Delta \tilde{U}_{i-1,j}^n \right) \\
+ \left( \frac{I}{\Delta \tau} + \tilde{A}_{i+1/2,j}^n \right) \Delta \tilde{U}_{i+1,j}^n \\
+ \left( \tilde{A}_{R,i+1/2,j}^n - \frac{\rho^* \alpha^* q^*}{Re_{-\infty}} \tilde{R}_{1,R,i+1/2,j}^n \right) \Delta \tilde{U}_{i+1,j}^n \right) \\
&+ \left( \Delta \tau \left( \left( -\tilde{B}_{L,i,j-1/2}^n + \frac{\rho^* \alpha^* q^*}{Re_{-\infty}} \tilde{S}_{2,L,i,j-1/2}^n \right) \Delta \tilde{U}_{i,j-1}^n \right) \\
+ \left( \frac{I}{\Delta \tau} + \tilde{B}_{L,i,j+1/2}^n \right) \Delta \tilde{U}_{i,j+1}^n \\
+ \left( \tilde{B}_{R,i,j+1/2}^n - \frac{\rho^* \alpha^* q^*}{Re_{-\infty}} \tilde{S}_{2,R,i,j+1/2}^n \right) \Delta \tilde{U}_{i,j+1}^n \right) \\
= - \Delta \tau \left( \tilde{F}_{i+1/2,j}^{n+1} - \tilde{F}_{i-1/2,j}^{n+1} + \left( \tilde{G}_{i+1/2,j+1/2}^{n+1} - \tilde{G}_{i+1/2,j-1/2}^{n+1} \right) \\
- \left( \tilde{F}_{i+1/2,j}^{n+1} - \tilde{F}_{i-1/2,j}^{n+1} - \left( \tilde{S}_{i+1/2,j+1/2}^{n+1} - \tilde{S}_{i+1/2,j-1/2}^{n+1} \right) \\
- \left( \Delta \tilde{F}_{21,i+1/2,j}^n - \Delta \tilde{F}_{21,i-1/2,j}^n \right) - \Delta \tilde{S}_{1,i+1/2,j}^n - \Delta \tilde{S}_{1,i-1/2,j}^n \right) = RHS \\
(3.49)
\end{align*}
\]

For the orientation of the index in equation (3.49) see the figure 3.1. Note that $\tilde{R}_1$ and $\tilde{S}_2$ denote here the viscous Jacobians. The system (3.49) is solved in two steps:

\[
\begin{align*}
\text{(first bracket in } (3.49)) \Delta \tilde{U}_{i,j}^{n+1} = RHS \\
(3.50)
\end{align*}
\]

\[
\begin{align*}
\text{(second bracket in } (3.49)) \Delta \tilde{U}_{i,j}^{n+1} = \Delta \tilde{U}_{i,j}^{n} \\
(3.50)
\end{align*}
\]

First the equations are solved in the $\xi-$ direction for a certain $\eta-$ position in equation (3.50). Thereafter, the equations are solved in the $\eta-$ direction for a certain $\xi$. The
3.4. TIME DISCRETISATION OF THE 2D NAVIER-STOKES EQUATIONS

solution $\Delta \vec{U}_{i,j}^n$ denotes the difference vector of the conservative variables between the new time level $(n+1)$ and the previous time level $n$.

**Definition of the inviscid Jacobians**

The vector of the conservative variables can be written as:

$$\vec{U} = \frac{1}{f} U$$

with

$$U = \begin{pmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{pmatrix} = \rho \begin{pmatrix} u \\ v \\ e \end{pmatrix}$$

and

$$\vec{p} = \begin{pmatrix} \rho \\ u \\ v \\ e \end{pmatrix}$$

and $\vec{p}$ denotes the vector of the primitive variables.

The chain rule gives a relationship between the primitive and the conservative variables,

$$\frac{\partial \vec{F}^\pm}{\partial \vec{U}} = \frac{\partial \vec{F}^\pm}{\partial \vec{p}_l} \frac{\partial \vec{p}_l}{\partial \vec{U}} = \frac{\partial \vec{F}^\pm}{\partial \vec{U}}$$

with the matrix $\frac{\partial \vec{p}_l}{\partial \vec{U}}$.

The derivative in equation (3.53) can be written applying a flux vector splitting as

$$\frac{\partial \vec{F}^\pm}{\partial \vec{U}} = \frac{\partial \vec{F}^\pm}{\partial \vec{U}} = \frac{\partial \vec{F}^\pm}{\partial \vec{U}} = \frac{\partial \vec{F}^\pm}{\partial \vec{U}} = \frac{\partial \vec{F}^\pm}{\partial \vec{U}}$$

using equation (3.53) as

$$\frac{\partial \vec{F}^\pm_i}{\partial \vec{U}} = \frac{\partial \vec{F}^\pm_i}{\partial \vec{p}_l} \frac{\partial \vec{p}_l}{\partial \vec{U}}$$

Equation (3.55) is valid for the van Leer flux vector splitting. For the AUSM scheme this equation has to be modified:

$$\vec{A}^\pm_{L,R} = \frac{\partial \vec{F}^+ \pm \vec{F}^-}{\partial \vec{U}_{k,L,R}}$$

Mathematically this corresponds to a linearisation of the convective flux with respect to the extrapolated variables $U_R$ and $U_L$:

$$\frac{\partial F_{i/2}^{1/2}}{\partial U_{i/2}} \Delta U_{i/2} = \frac{\partial F_{i/2}^{1/2}}{\partial U_{i/2}} \Delta U_L + \frac{\partial F_{i/2}^{1/2}}{\partial U_{i/2}} \Delta U_R$$
Since the flux at the cell face cannot be separated the Jacobians are obtained by deriving the sum of the both fluxes \( \tilde{F}^+ + \tilde{F}^- \), for the upwinded conservative variables \( u^\pm_{k,l,r} \).

In the same way the Jacobians \( \tilde{B}^\pm \) in the \( \eta^- \) direction are derived.

**Definition of the viscous Jacobians**

Formally the viscous Jacobians are obtained by the derivative with respect to the conservative variables:

\[
\tilde{R}^n_{1;L,R}, \tilde{S}^n_{2;L,R} = \frac{\partial \tilde{P}_1, \tilde{S}_2}{\partial u_{k,l,r}} 
\]  
(3.58)

The partial derivatives of the viscous terms \( \tilde{P}_1, \tilde{S}_2 \) are written in discrete form with first order accuracy. Thereafter, the partial derivative with respect to the upwinded variable with first order accuracy at a certain cell face \( i,j \) according to equation (3.2) is taken.

**Jacobian matrix at the wall**

**Inviscid Jacobians** The Jacobian at the wall is defined as:

\[
\tilde{B}_{Wall;L,R} = \frac{\partial \tilde{G}_{Wall}}{\partial \tilde{U}_{L,R}} 
\]  
(3.59)

with

\[
\tilde{G}_{Wall} = \begin{pmatrix} \rho, -u \xi, -v \xi, -\eta \end{pmatrix}^T . 
\]  
(3.60)

Introducing the vector \( \tilde{p}_{Wall} \) of the primitive variables at the wall

\[
\tilde{p}_{Wall} = (\rho, u, v, e_c)^T_{Wall} 
\]  
(3.61)

\( \tilde{B}_{Wall;L,R} \) follows to

\[
\tilde{B}_{Wall_{Lor,R}} = \frac{\partial \tilde{c}_{Wall}}{\partial \tilde{U}_k} = \frac{\partial \tilde{G}_{Wall} \partial \tilde{p}_{Wall}}{\partial \tilde{U}_{k,l,r}} 
\]  
(3.62)

Note that the Jacobian is calculated with the unsplit fluxes at the wall. The values on the wall are extrapolated with 0 order accuracy to the blade. Depending if this is the right \( U_R \) or left \( U_L \) upwinded value the partial derivative is performed. Consequently, only one Jacobian in the \( \eta^- \) direction is different from zero.

**Viscous Jacobians** Formally, the viscous Jacobian at the wall is obtained in the same way as in the field, see equation (3.58). However, at the wall the derivative is obtained with the wall velocity \( u_w, v_w \) in the x- and y-direction which is zero for the partial derivative with respect to the upwinded variables \( U_L, U_R \). Moreover, the derivatives for the heat flux are zero, since the adiabatic wall condition is applied.

\[
\tilde{R}^n_{1;L,R}, \tilde{S}^n_{2;L,R} = \frac{\partial \tilde{P}_1, \tilde{S}_2}{\partial u_{k,l,r}} 
\]  
(3.63)
3.4. TIME DISCRETISATION OF THE 2D NAVIER-STOKES EQUATIONS

Numerical discretisation implicit boundary condition

The boundary conditions in a discrete form follow from equation (2.37). The partial derivatives are discretised by central differences in $\eta -$ direction and by forward or backward differences in $\xi -$ direction.

**inlet, $\xi -$ direction**

\[
(I - \Delta \tau L^E_{2,j} \bar{A}_{2,j}) \Delta \bar{U}^n_{2,j} + \Delta \tau \Delta L^E_{2,j} \bar{A}_{3,j} \Delta \bar{U}^n_{3,j} = \\
- \Delta \tau L^E_{2,j} \left( \bar{F}_{3,j} - \bar{F}_{2,j} + \frac{1}{2} \left( \bar{G}_{2,j+1} - \bar{G}_{2,j-1} \right) \right)
\]  

(3.64)

**outlet, $\xi -$ direction**

\[
(I + \Delta \tau L^A_{IC,j} \bar{A}_{IC,j}) \Delta \bar{U}^n_{IC,j} - \Delta \tau \Delta L^A_{IC,j-1} \bar{A}_{IC-1,j} \Delta \bar{U}^n_{IC-1,j} = \\
- \Delta \tau L^A_{IC,j} \left( \bar{F}_{IC,j} - \bar{F}_{IC-1,j} + \frac{1}{2} \left( \bar{G}_{IC,j+1} - \bar{G}_{IC,j-1} \right) \right)
\]  

(3.65)

**inlet, $\eta -$ direction**

\[
\Delta \bar{U}^m_{2,j} + \frac{1}{2} \Delta \tau L^E_{2,j} \bar{B}_{2,j+1} \Delta \bar{U}^m_{2,j+1} - \frac{1}{2} \Delta \tau L^E_{2,j} \bar{B}_{2,j-1} \Delta \bar{U}^m_{2,j-1} = \bar{U}_{2,j}
\]  

(3.66)

**outlet, $\eta -$ direction**

\[
\Delta \bar{U}^n_{IC,j} + \frac{1}{2} \Delta \tau L^A_{IC,j} \bar{B}_{IC,j+1} \Delta \bar{U}^n_{IC,j+1} - \frac{1}{2} \Delta \tau L^A_{IC,j} \bar{B}_{IC,j-1} \Delta \bar{U}^n_{IC,j-1} = \bar{U}_{IC,j}
\]  

(3.67)

The Jacobians $\bar{A}$, $\bar{B}$ are computed with the unsplit inviscid fluxes $\bar{F}$, $\bar{G}$ in the transformed computational space [Pulliam and Chaussee, 1981]. The computation of the influence matrix $L$ is described in the previous chapter.

**Computational time for particular test cases**

The viscous unsteady computations on standard configuration 5 with an incidence angle of 4 degrees running 12 cycles, presented in section 4.2 were computed in approximately 5 days on 16 nodes of the IBM SP2 with the parallel version of the code. On the other hand using the implicit code the same computation can be performed on one node in approximately 6 days. In general CFL numbers between 20 and 40 can be used for the implicit calculations.
3.5 Discretisation of the Baldwin-Lomax turbulence model

The skin friction at the wall is computed according to Hölm [1996]. The vorticity is computed on the nodes of the computational mesh with the derivatives used as in the section 3.2. Therefore, the metric terms are extrapolated to the nodes, and the derivatives of the velocity are approximated as central differences with 2nd order accuracy on the nodes. The Baldwin-Lomax turbulence model is used for the unsteady computations without any changes. Algebraic turbulence models are widely used for steady state calculations. The application of such a model to time dependent flow is based on the assumption that the flutter period is much longer than the time it takes for the turbulent eddy viscosity to adapt to a certain velocity profile.

3.6 Stability criteria

3.6.1 Steady state

For steady state explicit inviscid and viscous computations a local time step $\Delta t_{\text{inviscid,viscous},i,j}$ is applied [Anderson et al., 1985]:

$$\Delta t_{\text{inviscid,viscous},i,j} =$$

$$CFL \left( J \left[ |uy_n - vx_n| + |ux_\xi - vy_\xi| + a \left( \sqrt{x_n^2 + y_n^2} + \sqrt{x_\xi^2 + y_\xi^2} \right) \right] \right)^{-1}_{i,j},$$

In the practical computations the CFL number is prescribed and depending on the flow and the mesh the local time step for each computational cell is computed and used.

3.6.2 Unsteady flow

The unsteady implicit inviscid and viscous computations are performed using a global CFL number defined by Anderson et al. [1985].

$$\Delta t_{\text{inviscid,viscous}} =$$

$$CFL \left( J \left[ |uy_n - vx_n| + |ux_\xi - vy_\xi| + a \left( \sqrt{x_n^2 + y_n^2} + \sqrt{x_\xi^2 + y_\xi^2} \right) \right] \right)^{-1},$$

In the practical computations the CFL number is prescribed and depending on the flow and the mesh the minimum time step is computed.

In order to perform time accurate computations the period time of the blade motion $T$ is divided by the max. overall time step $dt$ obtained by the steady state analysis (equation (3.69)) in order to obtain the number of time steps $k_{cycle}$ per cycle.
\[ k_{cycle} = \frac{T}{\Delta t_{viscous/inviscid}} \]  

(3.70)

### 3.7 Convergence criteria

#### 3.7.1 Steady state

In order to ensure that the steady state cases computed during the tests are converged to the residual to the \(L_2\) norm [Törnig and Spallucci, 1988] is studied for the independent variables \(\phi\). It is calculated as

\[ \epsilon_{L_2} = \log_e \left\{ \frac{\sum_{i=2,j=2}^{I_{dim},J_{dim}} \left| \phi_{i,j}^{n+1} - \phi_{i,j}^n \right|}{\Delta t \cdot IC \cdot JC} \right\} \]  

(3.71)

where \(\phi = \rho, u, v, e_c\).

Furthermore, the maximum norm of the density \(L_\infty\) is calculated for the steady state convergence.

\[ \epsilon_{L_\infty} = \max_{i=2,j=2}^{I_{dim},J_{dim}} \left| \rho_{i,j}^{n+1} - \rho_{i,j}^n \right| \]  

(3.72)

Equation (3.72) is proportional to the change of density in time in a control volume, i.e. the computational cell during one time step.

The values obtained for equation (3.72) are given in chapter 4 for the different calculations.

#### 3.7.2 Unsteady flow

In order to check the unsteady results for convergence four different criteria can be used.

- The time history of the maximum residuum as defined in equation (3.72) is plotted against the number of time steps. If the amplitude does not change anymore and the period length coincides with the period of the blade motion a periodic solution is obtained.

- In the same way the unsteady pressure close to the blade can be studied.

- In the far-field, i.e. close to the inlet- and outlet- boundary, the pressure probably don’t change very much. Therefore, it is checked if the unsteady periodic boundary condition is fulfilled, i.e. the flow variables at the upper period boundary leads the lower periodic boundary by the interblade phase angle.

- The amplitude of the integrated force or moment coefficient for one period of the bending or pitching vibration is studied for consecutive periods. If the value changes from one to the next period lesser than 0.1 per cent a periodic solution is assumed.
For most of the unsteady computations the last criterion is used in order to check for unsteady periodic flow.
Chapter 4

Results

In the beginning of this chapter the laminar flow for a flat plate is investigated. This is done in order to study the behavior of the code in the viscous boundary layer. It is found [Höhn, 1996] that the van Leer splitting for the convective terms of the Navier-Stokes equation is not able to resolve the boundary layer with high accuracy. The reason for that is the high artificial viscosity which is produced in the viscous boundary layer. Therefore, the implementation of the "Advection Upstream Splitting Method" (AUSM) for the convective terms is done, which uses one sided differences which produce less artificial viscosity. The AUSM - scheme is able to resolve the strong gradients in the u-velocity component close to the wall with the same accuracy as a finite difference splitting method [Bergamini and Cinnella, 1994], which is shown in section 4.1.1.

Besides, the viscous turbulent boundary layer is investigated for a flat plate. It is shown in section 4.1.1 that the AUSM for the convective terms and central differences for the diffusive terms together with the Baldwin-Lomax turbulence model gives reasonable results in turbulent viscous boundary layer on a flat plate.

In order to check the separational behavior of the code the steady state laminar flow around a 2D cylinder is studied in section 4.1.1. Furthermore, this is an excellent test case to prove the discretisation of the finite difference scheme in body fitted coordinates.

After that, steady state viscous calculations for turbulent flow on different international standard configurations are conducted. These are studied in order to check the separational behavior for viscous turbulent flow around cascaded blades of a compressor blade (STCF 5), the pressure distribution on two turbine blades (STCF 4 and 11), the capability of the scheme for shocks in cascades (STCF 10) and the prediction of the skin friction (STCF 10) as shown in section 4.1.2.

The final goal namely the prediction of unsteady viscous flow at or near stall and in the vicinity of shocks is studied in section 4.2 by the standard configuration 5, the standard configuration 10 and standard configuration 11, respectively. Standard configuration 5 is an excellent test case to study the prediction capabilities of the present method for unsteady stalled flow. Standard configuration 10 is a subsonic and transonic test case where the flow weakly separates. The transonic case shows unsteady viscous shock boundary layer interaction with shock induced boundary layer separation. Standard configuration 11 provides a test case for subsonic and transonic flow, where the transonic case shows large separation at high angle of attack.
<table>
<thead>
<tr>
<th>$Ma_{\infty}$</th>
<th>$Re_{\infty}$</th>
<th>Pr</th>
<th>no. of mesh-points</th>
<th>$\Delta y_{\text{min}}$</th>
<th>pitch</th>
<th>drop of $c_{\text{L}_{\infty}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>1 000 000</td>
<td>0.72</td>
<td>100 x 100</td>
<td>0.0004</td>
<td>10 * chord</td>
<td>$10^6$</td>
</tr>
</tbody>
</table>

Table 4.1: Flow parameters, flat plate for $Re_{\infty} = 1 000 000$

The results obtained by the present method are always denoted with INSTHPT.

4.1 Steady state results

4.1.1 Basic test cases

Flat plate for laminar and turbulent flow

The figure 4.1 shows the velocity distribution in comparison with the Blasius solution for a Reynolds number of 1 000 000 at a certain position on the plate. This flow case is characterized by sharp gradients perpendicular to the stream-wise velocity direction near the wall. Nevertheless, the AUSM-scheme is able to resolve these gradients with a reasonable accuracy. The inlet Mach number is 0.3 in order to have no compressible effects, due to the fact that the Blasius solution is only valid for incompressible flow. The most important flow data for these calculations are given in table 4.1.

Figure 4.2 shows the calculated local skin friction in comparison with the Blasius solution. The agreement is found to be reasonably good.

![Velocity distribution](image)

Figure 4.1: Velocity distribution on a flat plate $Re = 1000000$, 100 x 100 mesh-points

The extension to the turbulent flow case is done by the implementation of the turbulence model of Baldwin and Lomax [1978]. The Reynolds averaged Navier-Stokes equations are
4.1. STEADY STATE RESULTS

Figure 4.2: Skin friction distribution on a flat plate $Re = 1000000$, 100 x 100 mesh-points

<table>
<thead>
<tr>
<th>$Ma_{-\infty}$</th>
<th>$Re_{-\infty}$</th>
<th>$Pr$, $Pr_t$</th>
<th>no. of mesh-points</th>
<th>$\Delta y_{min}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.675</td>
<td>1 000 000</td>
<td>0.72, 0.9</td>
<td>100 x 100</td>
<td>0.00004</td>
</tr>
<tr>
<td>turbulence model</td>
<td>$Y_{min}^{+}$</td>
<td>drop of $\epsilon_{\infty}$</td>
<td>pitch</td>
<td></td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>1.5</td>
<td>$10^7$</td>
<td>1 ° chord</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.2: Flow parameters, flat plate for $Re_{-\infty} = 1 000 000$, turbulent calculation

solved with a grid consisting of 100 x 100 mesh points in the whole domain. Important flow parameters are given in table 4.2.

The smallest dimensionless coordinate from the wall $y_{min}$ is approximately 1.5 close to the leading edge and therefore sufficiently small in order to resolve the 'laminar' sublayer [Fletcher, 1991].

Finally, figure 4.3 shows the velocity profile for the turbulent boundary layer close to the wall against the 'laminar' sublayer and the law of the wall. Again, the agreement is reasonably good.

A detailed study of the used numerical scheme for laminar and turbulent flow for the flat plate, e.g. the prediction of thermal boundary layers, can be found in Höhn [1996].

Laminar flow around a cylinder

In order to study the separational behavior of the laminar code, this test case is chosen (see figure 4.4). The angle of separation which appears in the wake of the cylinder is denoted by $\theta$, whereas the length of the separation bubble is $L$. This capability of a code is essential for the main goal of the project, i.e. stall flutter. Moreover, it is an excellent test case to check the discretisation and the metric terms of the used scheme. In contrast to the flat plate calculations all metric terms $(x_\xi, y_\xi, x_\eta, y_\eta)$ are different from zero.

Figure 4.5 shows the H-mesh used for the calculations. The pitch is chosen to be 25
Figure 4.3: $u^+ = f(y^+)$ on a flat plate $Re = 1000000$ in comparison with the "log law" of the wall and the law for the "laminar" sublayer, 100 x 100 mesh-points

<table>
<thead>
<tr>
<th>$Ma_{\infty}$</th>
<th>$Re_{\infty}$</th>
<th>Pr</th>
<th>no. of mesh-points</th>
<th>$\Delta y_{\text{min}}$</th>
<th>pitch</th>
<th>drop of $\epsilon L_{\infty}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>10, 30</td>
<td>0.72</td>
<td>106 x 100</td>
<td>0.06</td>
<td>25 * D</td>
<td>$10^6$</td>
</tr>
</tbody>
</table>

Table 4.3: Flow parameters, 2D cylinder

times the diameter of the cylinder, so that free stream conditions are achieved in the middle of the channel, see also table 4.3.

Figure 4.6 and figure 4.7 show the separation angle $\theta$ and the length of the separation bubble $L/D$ for different Reynolds numbers $Re_D$ against results obtained by other authors, here Dennis and Chang [1970] and Bouard and Coutanceau [1980]. Both, the separation point and the length of the separation bubble are reasonably predicted.

Figure 4.4: Flow around a cylinder in 2D, geometry
4.1. STEADY STATE RESULTS

Figure 4.5: H-type mesh used for the computations, 106 x 100 mesh-points

Figure 4.6: Separation angle of the separation bubble in comparison with other authors, 106 x 100 mesh-points

The test case proves that the used numerical scheme is able to capture the laminar separation well.

4.1.2 Cascade flow

After these very basic test cases the results for viscous flow around cascades in two dimensions are studied.
Figure 4.7: Length of the separation bubble in comparison with other authors, 106 x 100 mesh-points

<table>
<thead>
<tr>
<th>$Ma_{-\infty}$</th>
<th>$Re_{-\infty}$</th>
<th>$Pr, Pr_t$</th>
<th>no. of mesh-points</th>
<th>$\Delta h_{\text{min}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.26</td>
<td>820 000</td>
<td>0.72, 0.9</td>
<td>106 x 91</td>
<td></td>
</tr>
<tr>
<td>turbulence model $Y^+_{\text{min}}$</td>
<td>drop of $e\lambda_{\infty}$</td>
<td>pitch</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>2.0</td>
<td>$6 \times 10^8$</td>
<td>0.76 * chord</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.4: Flow parameters, standard configuration 4, case 3

Section 4

Stcf 4

This annular turbine cascade test case was experimentally investigated at Lausanne Institute of Technology [Bölcs et al., 1984]. This cascade represents a typical section of a turbine airfoil with a flutter instability in the first bending mode. The Reynolds number is 820 000 referred to chord and inlet velocity with an inlet Mach number of 0.28. Table 4.4 and table 4.5 show the flow parameters of the calculations performed with INSTHPT, other used codes and the experimental data.

The results are shown in figure 4.9. The steady state pressure distribution, agree with experimental data (see [Bölcs et al., 1984]) and calculations of other viscous flow solvers, here [Grüber and Carstens, 1996], who used the same mesh, which is show in figure 4.8, and has a $y^+$ of about 2.0. The better prediction of the inviscid results is probably due to 3D effects in the annular cascade experiments. The slightly overshoot in the velocity on the suction side, which corresponds to a lower surface pressure $c_p$, was also observed by Abhari and Giles [1995]. This overshoot is due to the blockage effect caused by the boundary layer for viscous flow which leads to an over-acceleration (Mach number bump) on the suction side of the blade in comparison with the inviscid results.
### Table 4.5: Flow parameters, international standard configuration 4, case 3

<table>
<thead>
<tr>
<th></th>
<th>Exp.</th>
<th>INSTHPT/viscous</th>
<th>INSTHPT/inviscid</th>
<th>DLR/viscous</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_{is_1}$</td>
<td>0.28</td>
<td>0.25</td>
<td>0.26</td>
<td>0.28</td>
</tr>
<tr>
<td>$\beta_1$</td>
<td>45</td>
<td>44.5</td>
<td>44.5</td>
<td>45.0</td>
</tr>
<tr>
<td>$M_{is_2}$</td>
<td>0.90</td>
<td>0.90</td>
<td>0.90</td>
<td>0.90</td>
</tr>
<tr>
<td>$\beta_2$</td>
<td>71.0</td>
<td>71.6</td>
<td>71.5</td>
<td>72.0</td>
</tr>
<tr>
<td>Turbulence Model</td>
<td></td>
<td>Baldwin-Lomax (B-L)</td>
<td>-</td>
<td>B-L</td>
</tr>
<tr>
<td>Governing equations</td>
<td></td>
<td>Navier-Stokes (2D)</td>
<td>Euler (2D)</td>
<td>Navier-Stokes (2D) [Grüber and Carstens, 1996]</td>
</tr>
</tbody>
</table>

### Figure 4.8: H-type mesh used for standard configuration 4, 106 x 91 meshpoints

**Stcf 5**

This two-dimensional cascade was investigated at the Office National d’Études et de Recherche Aérospatiales (ONERA). The configuration consists of six fan stage tip sections [Bölcs and Fransson, 1986].

**4 degrees incidence** Figure 4.10 shows the cascade geometry and the mesh used for the calculations of the program INSTHPT with 80 points on the blade on the upper and the lower side, respectively and 120x100 meshpoints in total. The necessary flow information is summarized in table 4.6 and table 4.7.

Table 4.7 shows the flow parameters of the calculations for INSTHPT, other numerical codes and the experimental data. The Reynolds number based on the inlet velocity and the chord length is 1400000 for this flow case [Fransson, 1993]. $Y^+$ is about 1.5 close to the leading edge for the calculations conducted with INSTHPT.

The results shown in figure 4.11, the steady state pressure distribution, agree with...
VOLFAP [Sidén, 1991]. Both calculations are conducted with an incidence angle of 4 degrees. The results show good agreement. However, a difference to the experimental

<table>
<thead>
<tr>
<th>$Ma_{\infty}$</th>
<th>$Re_{\infty}$</th>
<th>$Pr, Pr_t$</th>
<th>no. of meshpoints</th>
<th>$\Delta h_{\text{min}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.51</td>
<td>1 400 000</td>
<td>0.72, 0.9</td>
<td>120 x 100</td>
<td>0.00003</td>
</tr>
<tr>
<td>turbulence model</td>
<td>$Y_{\text{min}}^+$</td>
<td>drop of $\epsilon_{\infty}$</td>
<td>pitch</td>
<td></td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>1.5</td>
<td>$10^6$</td>
<td></td>
<td>0.95 * chord</td>
</tr>
</tbody>
</table>

Table 4.6: Flow parameters, standard configuration 5, case 2
4.1. STEADY STATE RESULTS

Figure 4.11: Steady state surface pressure distribution of standard configuration 5, case 2

<table>
<thead>
<tr>
<th></th>
<th>Exp.</th>
<th>INSTHPT</th>
<th>VOLSOL</th>
<th>VOLFAP</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Mis_1$</td>
<td>0.5</td>
<td>0.51</td>
<td>0.51</td>
<td>0.52</td>
</tr>
<tr>
<td>$\beta_1$</td>
<td>63.3</td>
<td>63.3</td>
<td>62.4</td>
<td>63.3</td>
</tr>
<tr>
<td>$Mis_2$</td>
<td></td>
<td>0.47</td>
<td>0.45</td>
<td>0.47</td>
</tr>
<tr>
<td>$\beta_2$</td>
<td></td>
<td>60.7</td>
<td>60.3</td>
<td>60.2</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>4.0</td>
<td>4.0</td>
<td>3.1</td>
<td>4.0</td>
</tr>
<tr>
<td>Turbulence</td>
<td></td>
<td>Baldwin-Lomax (B-L)</td>
<td>B-L</td>
<td>B-L</td>
</tr>
<tr>
<td>Model</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Governing</td>
<td></td>
<td>Navier-Stokes (2D)</td>
<td>Navier-Stokes (2D)</td>
<td>Navier-Stokes (2D)</td>
</tr>
<tr>
<td>equations</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 4.7: Flow parameters, international standard configuration 5, case 2

data (see Bölcs and Fransson [1986]) is found. Predictions of another viscous flow solver VOLSOL (see Groth et al. [1996]) conducted with an angle of incidence of 3.1 degrees show a better agreement with the experimental data. This is confirmed by other researchers as well. The reason for this is not clear [Fransson and Verdon, 1993].

6 degrees incidence The calculations are carried out according to the experimental flow conditions with an inlet Mach number of 0.53 and a Reynolds number of 1400000 on a mesh shown in figure 4.10. The geometrical properties of the mesh are kept the same (see table 4.6). However, the angle of incidence is set to five degrees in the computations. Former investigations [Hedberg, 1994] on this test case showed that better agreement with the experiments is achieved by decreasing the angle of incidence in the computations to five degrees. The necessary information is summarized in the tables 4.8 and 4.9.

The $y^+$ close to the leading edge on the suction side of the blade is about 1.5. The computed $c_p$ values, using the definition established by Fransson and Verdon [1993], are compared with results obtained by Hu [1995] and the experimental values, see figure 4.12.
$$Ma_{\infty} \quad Re_{\infty} \quad Pr, Pr_t \quad no. \text{ of mesh-points} \quad \Delta h_{\min}$$

<table>
<thead>
<tr>
<th>turbulence model</th>
<th>$Y^+_{\text{min}}$</th>
<th>drop of $\epsilon_{L,\infty}$</th>
<th>pitch</th>
<th>0.00003</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baldwin-Lomax</td>
<td>2.1</td>
<td>$4 \times 10^{5}$</td>
<td>0.95 * chord</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.8: Flow parameters, standard configuration 5, case 3

<table>
<thead>
<tr>
<th>$M_i$</th>
<th>$M_b$</th>
<th>$\beta_1$</th>
<th>$\beta_2$</th>
<th>$\alpha$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>0.5</td>
<td>65.3</td>
<td>60.6</td>
<td>6.0</td>
</tr>
<tr>
<td>0.45</td>
<td>0.45</td>
<td>64.3</td>
<td>60.2</td>
<td>5.0</td>
</tr>
<tr>
<td>45</td>
<td>45</td>
<td>64.0</td>
<td>4.7</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.9: Flow parameters, international standard configuration 5, case 3

The numerical results obtained on the same mesh show fairly good agreement for the present method INSTHPT and Navier-Stokes solver VOLSOL [Hu, 1995].

![Graph of Cp vs Chordwise Distance](image)

Figure 4.12: Steady state surface pressure distribution of standard configuration 5, case 3, $i = 5^\circ$

The pressure side is in perfect agreement comparing the numerical results and the experiments. On the suction side the present method INSTHPT indicate separation which
is shown by a moderate increase of the pressure close to the leading edge, whereas VOLSOL does not show separation in the surface pressure plot. Neither of the codes follow the experiments in the separated area. Downstream the separation where the flow reattaches again to the blade surface the computational results are again in good agreement with the experiments.

Figure 4.13: Length of the separation bubble against the angle of incidence, standard configuration 5

The size of the separation bubble is about twenty percent of chord length for the present method INSTIPT, whereas VOLSOL shows a separated area of approximately fifteen percent of chord length (fig. 4.13).

8 degrees incidence The third test case performed on standard configuration 5 is conducted with a nominal angle of incidence of eight degrees. The computations are performed with VOLSOL [Groth et al., 1996] and the present method INSTIPT on the mesh shown in figure 4.10, giving a $y^+$ lower than 2 on the whole blade surface for the computations. The angle of incidence used is 7.5 degrees, because somewhat better agreement to the experiments is found in general with this lower angle. Numerical results from other researchers at lower incidence confirm this circumstance [Fransson and Verdon, 1993]. The necessary information is summarized in the tables 4.10 and 4.11.

A good agreement between the numerical predictions and the experiments, see figure 4.14, is found on the pressure side of the blade. On the suction side the numerical predictions differ significantly for twenty five up to seventy percent of chordwise distance

<table>
<thead>
<tr>
<th>$Ma_{\infty}$</th>
<th>$Re_{\infty}$</th>
<th>$Pr$, $Pr_t$</th>
<th>no. of mesh-points</th>
<th>$\Delta h_{\text{min}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.53</td>
<td>1 400 000</td>
<td>0.72, 0.9</td>
<td>120 x 100</td>
<td>0.00003</td>
</tr>
<tr>
<td>turbulence model</td>
<td>$Y_{\text{min}}^+$</td>
<td>drop of $\epsilon_{\infty}$</td>
<td>pitch</td>
<td></td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>2.2</td>
<td>$5 \times 10^6$</td>
<td>0.95 * chord</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.10: Flow parameters, standard configuration 5, case 12
Table 4.11: Flow parameters, international standard configuration 5, case 12

between each other. Neither of the methods can predict the very low pressure distribution close to the leading edge measured in the experiments.

Figure 4.14: Steady state surface pressure distribution of standard configuration 5, case 12, \( i = 7.5^\circ \)

The present method INSTHPT shows a completely separated flow. VOLSOL is almost completely separated and gives only an attached flow between eighty and ninety percent of chord. Reasons for the different behavior of VOLSOL and INSTHPT could be the different implementation of the Baldwin-Lomax turbulence model. The experiments showed fully separated flow for an angle of attack of ten degrees or higher, but partly separated flow for an angle of attack of eight degrees [Fransson and Verdon, 1993].

Figure 4.13 shows the length of the separation bubble against the angle of incidence. Comparison is made again with VOLSOL [Groth et al., 1996] and VOLFAP [Siden, 1991] and good agreement is found. VOLSOL seems to predict smaller separation bubbles than the other two codes especially for small angles of incidence. The computations for all three
<table>
<thead>
<tr>
<th>$Ma_{-\infty}$</th>
<th>$Re_{-\infty}$</th>
<th>Pr, $Pr_t$</th>
<th>no. of mesh-points</th>
<th>$\Delta h_{\min}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.57</td>
<td>1 400 000</td>
<td>0.72, 0.9</td>
<td>120 x 100</td>
<td>0.00003</td>
</tr>
<tr>
<td>turbulence model</td>
<td>$Y_{m+}^*$</td>
<td>drop of $e_{R_{-\infty}}$</td>
<td>pitch</td>
<td></td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>2.8</td>
<td>$10^5$</td>
<td>0.95 * chord</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.12: Flow parameters, standard configuration 5, case 18

<table>
<thead>
<tr>
<th></th>
<th>Exp.</th>
<th>INSTHPT</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_i s_1$</td>
<td>0.5</td>
<td>0.57</td>
</tr>
<tr>
<td>$\beta_1$</td>
<td>69.3</td>
<td>69.3</td>
</tr>
<tr>
<td>$M_i s_2$</td>
<td>0.38</td>
<td></td>
</tr>
<tr>
<td>$\beta_2$</td>
<td>61.3</td>
<td></td>
</tr>
<tr>
<td>$\alpha$</td>
<td>10.0</td>
<td>10.0</td>
</tr>
<tr>
<td>Turbulence</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Model</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Governing</td>
<td></td>
<td></td>
</tr>
<tr>
<td>equations</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Baldwin-</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Lomax (B-L)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Navier-</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Stokes (2D)</td>
</tr>
</tbody>
</table>

Table 4.13: Flow parameters, international standard configuration 5, case 18

codes are performed with the turbulence model by Baldwin and Lomax [1978]. Values for higher angles of attack than six degrees are not shown in the diagram because for eight degrees incidence angle the flow is completely separated in the predictions of INSTHPT and more or less fully separated in the results obtained with VOLSOL.

10 degrees incidence The fourth test case shows the surface pressure distribution of the fifth standard configuration for an angle of incidence of ten degrees. The same mesh as for the previous cases, shown in figure 4.10, is used.

The necessary information is summarized in the tables 4.12 and 4.13.

This case is characterized by a complete separation of the flow on the suction side of the blade in the computations as well as in the experiments. The numerical results presented in figure 4.15 show higher pressure values than measured in the experiments, though the back pressure from the experiments is taken for the numerical calculations. Lower values could be achieved by decreasing the angle of incidence as in the previous case. So far no other numerical data is published on this case. Further investigations should show if a better agreement with the experiments can be achieved when the calculations are performed with an inlet Mach number according to the experiments. However, the present study uses the values given by Fransson and Verdon [1993].

Baldwin Lomax turbulence model for separated flow Boussinesq [1877] introduced an analogy for the turbulent stresses similar to the viscous, laminar stresses. He described the turbulent shear stress $\overline{\nu}$ as a product of the turbulent viscosity $\nu_t$ and the
velocity gradient perpendicular to the mean flow:

$$\vec{w} = -\nu \frac{\partial U}{\partial y}$$  \hspace{1cm} (4.1)

One can show that this assumption does not account for rotational effects. This is the major drawback of the models based on the Boussinesq assumption. The Baldwin Lomax turbulence model is based on this assumption and can therefore not really describe rotational turbulent flow, i.e. the turbulent viscosity $\nu_t$ in turbulent separated flow.

The Baldwin and Lomax turbulence model is a two-layer, zero equation (algebraic) model. It is patterned after the Cebeci-Smith model [Cebeci and Smith, 1974] and introduces a modification that eliminates the need to search for the edge of the boundary layer to determine the length scale. Its strength and weakness are well known in the CFD community; it predicts accurately the steady flows with little or no separation and performs poorly if there is large separation, either shock-induced or otherwise.

Despite its disadvantages the model is used, because of its simple implementation and the fact that it is computationally less expensive than one equation or two equation models, like the k-\$\varepsilon\$ turbulence model [Hölm, 1997 a].

Some of the constants of the theory are determined by correlating with experimental data. The parameter $c_{uk}$ is often changed from its original value 0.25 to 1.0 [Haase et al., 1993]. The higher value gives a stronger interaction of the inner and outer formulation of the Baldwin-Lomax turbulence model and a better convergence, but leads to smaller separated areas [Haase et al., 1993]. This has been investigated for standard configuration 5. For an angle of incidence of 7.5 degrees the higher value of $c_{uk}$ gives lower separation, i.e. a separation bubble of 23 % of chord against 90 % of separation for $c_{uk}$ of 0.25. Here, the results for $c_{uk} = 0.25$ are given. For an angle of incidence of 10 degrees only a value of 1.0 for $c_{uk}$ gives a non-oscillating, converged, fully separated solution and is shown here.
4.1. **STEADY STATE RESULTS**

<table>
<thead>
<tr>
<th>$Ma_{-\infty}$</th>
<th>$Re_{-\infty}$</th>
<th>Pr, $Pr_t$</th>
<th>no. of mesh-points</th>
<th>$\Delta h_{\min}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.70</td>
<td>1 000 000</td>
<td>0.72, 0.9</td>
<td>140 x 100(40)</td>
<td>0.00003</td>
</tr>
<tr>
<td>turbulence model</td>
<td>$Y_{\min}^+$</td>
<td>drop of $\epsilon_{L_{-\infty}}$</td>
<td>pitch</td>
<td></td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>1.8</td>
<td>$10^9$</td>
<td>1 * chord</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.14: **Flow parameters, standard configuration 10, $M_1 = 0.7$**

<table>
<thead>
<tr>
<th></th>
<th>Exp.</th>
<th>INSTHPT viscous</th>
<th>INSTHPT inviscid</th>
<th>Hall</th>
<th>Verdon</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\dot{M}_{s1}$</td>
<td>-</td>
<td>0.70</td>
<td>0.70</td>
<td>0.70</td>
<td>0.70</td>
</tr>
<tr>
<td>$\beta_1$</td>
<td>-</td>
<td>55.0</td>
<td>55.0</td>
<td>55.0</td>
<td>55.0</td>
</tr>
<tr>
<td>$\dot{M}_{s2}$</td>
<td>-</td>
<td>0.45</td>
<td>0.47</td>
<td>0.47</td>
<td></td>
</tr>
<tr>
<td>$\beta_2$</td>
<td>-</td>
<td>40.0</td>
<td>40.3</td>
<td>41.1</td>
<td></td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>Baldwin-Lomax (B-L)</td>
<td>$-$</td>
<td>$-$</td>
<td>(B-L)</td>
<td></td>
</tr>
<tr>
<td>Governing equations</td>
<td>Navier-Stokes (2D)</td>
<td>Euler (2D)</td>
<td>Euler (2D) [Hall and Clark, 1991]</td>
<td>Navier Stokes (2D) [Ayer and Verdon, 1996]</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.15: **Flow parameters, international standard configuration 10, $M_1 = 0.7$**

**St cf 10**

In order to study the shock capturing capabilities of the Advection Upstream Splitting Method the tenth standard configuration [Verdon, 1987], a compressor cascade, is chosen. The tenth standard configuration, included by a proposal of Dr. J.M. Verdon [1987], is a two-dimensional compressor cascade of modified NACA 0006 profiles that operate at subsonic inlet and outlet conditions.

In the beginning the subsonic case is studied. The calculations are conducted on a H-type mesh (see figure 4.16). The flow parameters are given in the tables 4.14 and 4.15 for the calculations performed with INSTHPT and other used codes.

Figure 4.17 shows the steady state inviscid and viscous results for an inlet Mach number of 0.7 and an inlet flow angle of 55°. The overall agreement of the codes is reasonably good. Bigger differences are found close to leading and trailing edge, which is due to the coarser mesh of the inviscid grid.

Consequently, the skin friction is investigated. Figure 4.18 shows the skin friction of INSTHPT in comparison with results of Ayer and Verdon [1996]. Reasonable agreement is found. A small separation is shown at the last 10 % of the suction side.

The second test case which is proposed by Verdon is a transonic flow situation around the same profile. For this calculation a different mesh is used, which is shown in figure 4.19.
Figure 4.16: H-type mesh used for standard configuration 10, 140 x 100 meshpoints

Figure 4.17: Steady state surface pressure distribution for STCF 10, $M_1 = 0.7$

It has got 140 x 100 meshpoints for the viscous and 140x40 for the inviscid calculations, respectively. The mesh lines are normal to the local blade surface. This is chosen in order to capture the shock reasonably well. This is due to the numerical formulation of the scheme. The limiter described in chapter 3 is only applied to the $\xi$-direction in the computational domain. It results from the original purpose of the code, namely the calculation of nozzle flow. Therefore, the mesh is created in a way that the shock is parallel to the mesh on the suction side. A very intensive investigation of this was performed by Börjesson [1997]. The main conclusions of that study are that a fine mesh resolution at the leading and trailing edge is necessary to achieve agreement with the solutions of other authors, e.g. Verdon, in that region. Moreover, normal mesh lines aligned with the shock give a good and sharp shock prediction, which is mainly due to the implemented shock capturing procedure in the code.

Figure 4.20 shows the steady state isentropic Mach number distribution for an inlet Mach number of 0.8 and an inlet flow angle of 58°. The viscous and inviscid results for
### 4.1. STEADY STATE RESULTS

<table>
<thead>
<tr>
<th>$Ma_{-\infty}$</th>
<th>$Re_{-\infty}$</th>
<th>$Pr, Pr_l$</th>
<th>no. of mesh-points</th>
<th>$\Delta h_{min}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.80</td>
<td>1 000 000</td>
<td>0.72, 0.9</td>
<td>140 x 100 (40)</td>
<td>0.00003</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>$Y^{+*}_{min}$</th>
<th>drop of $\epsilon_L$</th>
<th>pitch</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baldwin-Lomax</td>
<td>1.8</td>
<td></td>
<td>1 * chord</td>
</tr>
</tbody>
</table>

Table 4.16: Flow parameters, standard configuration 10, $M_1 = 0.8$

<table>
<thead>
<tr>
<th>$\text{Exp.}$</th>
<th>$\text{INSTHPT viscous}$</th>
<th>$\text{INSTHPT inviscid}$</th>
<th>$\text{Huff inviscid}$</th>
<th>$\text{Verdon viscous}$</th>
<th>$\text{Grüber viscous}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{Mis}_1$</td>
<td>-</td>
<td>0.80</td>
<td>0.80</td>
<td>0.80</td>
<td>0.80</td>
</tr>
<tr>
<td>$\beta_1$</td>
<td>-</td>
<td>58.0</td>
<td>58.0</td>
<td>58.0</td>
<td>58.0</td>
</tr>
<tr>
<td>$\text{Mis}_2$</td>
<td>-</td>
<td>0.49</td>
<td>0.49</td>
<td>0.477</td>
<td></td>
</tr>
<tr>
<td>$\beta_2$</td>
<td>-</td>
<td>42.0</td>
<td>41.0</td>
<td>42.0</td>
<td></td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>Baldwin-Lomax (B-L)</td>
<td>-</td>
<td>-</td>
<td>(B-L)</td>
<td>(B-L)</td>
</tr>
<tr>
<td>Governing equations</td>
<td>Navier-Stokes (2D)</td>
<td>Euler (2D)</td>
<td>Euler (2D) [Huff et al., 1991]</td>
<td>Navier-Stokes (2D) [Ayer and Verdon, 1996]</td>
<td>Navier-Stokes (2D) [Grüber and Carstens, 1996]</td>
</tr>
</tbody>
</table>

Table 4.17: Flow parameters, international standard configuration 10, $M_1 = 0.8$
the present method INSTHPT are compared with a method solving the Navier-Stokes equations developed by Ayer and Verdon [1996] and a 2D cascade solver based on the Euler equations by Huff et al. [1991]. The strength of the shock and the position are reasonably well predicted comparing the inviscid computations. For the viscous computations the computed Mach number distribution smears out a bit which indicates shock boundary-layer interaction, shown by both viscous calculations. The shock causes the flow to separate which can be seen in the skin friction plot.

Figure 4.21 shows the computed skin friction against results obtained by Ayer and Verdon [1996]. The shock induced separation agrees reasonably well.

Figure 4.20 compares furthermore the results for INSTHPT with results from Grüber [Grüber and Carstens, 1996]. The results from Grüber show an overshoot in front of the shock, which can not be seen is the predictions of INSTHPT as well as in the results by Ayer and Verdon [1996]. Reason for the overshoot could be the fact that Grüber
4.1. STEADY STATE RESULTS

Figure 4.20: Steady state Mach number distribution of standard configuration 10, $M_1 = 0.8$ calculates with 2nd order accuracy over the shock whereas in INSTHPT in the vicinity of the shock the spatial discretisation is switch to first order. Beside the shock region both methods give almost identical results. Grüber does not use mesh lines parallel to the shock front, as shown in figure 4.19, for a better shock capturing, but try to improve the shock prediction by using the Fromm limiter [Venkatakrishnan, 1993] with less success than with the present method INSTHPT. The steady state results for this case are given here since a comparison of the unsteady prediction is shown in the next chapter.

Figure 4.21: Skin friction of standard configuration 10, $M_1 = 0.8$

Stcf 11

The 11th standard configuration [Fransson et al., 1998] represents a turbine blade geometry with transonic design flow conditions. Figure 4.23 shows the comparison of predicted
steady results for the subsonic case [Khalifa, 1999], see tables 4.18 and 4.19, in terms of the isentropic Mach number distribution for a mesh given in figure 4.22. All calculations show the same behavior, i.e. the slight overprediction of the Mach number in the middle of the blade on the suction side in comparison with the experiments. This can be due to real flow effects, e.g. 3D flow field, which cannot be described by the used prediction models. Good agreement between the numerical predictions and the experiments is found on the pressure side of the blade. In general the numerical results show no difference between the inviscid (INSTHPT) and viscous calculations (INSTHPT and VOLFAP [Sidén, 1991]), which can be expected for a subsonic flow case where the flow is not separated.

Table 4.18: Flow parameters, standard configuration 11, subsonic case

<table>
<thead>
<tr>
<th>$M_{\infty}$</th>
<th>$R_{\infty}$</th>
<th>$Pr, Pr_t$</th>
<th>no. of mesh-points</th>
<th>$\Delta h_{\text{min}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$565,818$</td>
<td>$0.72, 0.9$</td>
<td>$88 \times 60$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>turbulence model</td>
<td>$Y^+_{\min}$</td>
<td>drop of $\epsilon_{\infty}$</td>
<td>pitch</td>
<td></td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>$1.0 \times 10^6$</td>
<td>$0.72 \times \text{chord}$</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 4.22: Mesh used for standard configuration 11, subsonic case

The off-design case calculations [Khalifa, 1999] show good agreement with the experimental data, see tables 4.20 and 4.21, with respect to the isentropic Mach number, see figure 4.25. In the leading edge region, see figure 4.24, the numerical results are strongly dependent on the grid resolution. The shock on the suction side of the blade smears out due to the fact that the mesh lines are not parallel to the shock front and probably because of shock boundary layer interaction. None of the codes can predict the measured pre-shock Mach number. This has to be seen in the context of shock sensibility of the experimental data, i.e. fairly small inlet flow changes gave significantly different pre shock conditions in the experiments. The viscous code is able to predict the separation bubble indicated by the deceleration, which occurs on approximately 15\% -30\% relative chord on the suction side. Difference between inviscid and viscous calculations are found on the
### 4.1. STEADY STATE RESULTS

<table>
<thead>
<tr>
<th></th>
<th>Exp.</th>
<th>INSTHPT viscous</th>
<th>INSTHPT inviscid</th>
<th>VOLFAP</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Ma_{\infty}$</td>
<td>0.31</td>
<td>0.29</td>
<td>0.29</td>
<td>0.29</td>
</tr>
<tr>
<td>$\beta_1$</td>
<td>-15.2</td>
<td>-15.2</td>
<td>-15.2</td>
<td>-15.2</td>
</tr>
<tr>
<td>$Ma_{\infty}$</td>
<td>0.69</td>
<td>0.69</td>
<td>0.69</td>
<td>0.69</td>
</tr>
<tr>
<td>$\beta_2$</td>
<td>66.7</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>Baldwin-Lomax (B-L)</td>
<td>Baldwin-Lomax (B-L)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Governing equations</td>
<td>Navier-Stokes (2D)</td>
<td>Euler (2D)</td>
<td>Navier-Stokes (2D)</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.19: Flow parameters, international standard configuration 11, subsonic case

![Mach number distribution](image)

Figure 4.23: Steady state Mach number distribution of standard configuration 11, subsonic case

Suction side of the blade close to the leading edge where the separation takes place and in the shock region, due to viscous shock boundary layer interaction predicted by the viscous model. On the pressure side of the blade the agreement for the numerical predictions and the experiments is good.

<table>
<thead>
<tr>
<th>$Ma_{\infty}$</th>
<th>$Re_{\infty}$</th>
<th>$Pr$, $Pr_1$</th>
<th>no. of mesh-points</th>
<th>$\Delta h_{\text{min}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.37</td>
<td>1 241 399</td>
<td>0.72, 0.9</td>
<td>93 x 59</td>
<td>0.00003</td>
</tr>
<tr>
<td>turbulence model</td>
<td>$Y_{\text{min}}^+$</td>
<td>drop of $\epsilon_{\infty}$</td>
<td>pitch</td>
<td></td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>1.8</td>
<td></td>
<td>0.72 * chord</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.20: Flow parameters, standard configuration 11, transonic case
Figure 4.24: *Mesh used for standard configuration 11, transonic case*

<table>
<thead>
<tr>
<th></th>
<th>Exp.</th>
<th>INSTHPT viscous</th>
<th>INSTHPT inviscid</th>
<th>VOLFAP</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M/s_1$</td>
<td>0.40</td>
<td>0.38</td>
<td>0.38</td>
<td>0.37</td>
</tr>
<tr>
<td>$\beta_1$</td>
<td>-34.0</td>
<td>-33.0</td>
<td>-33.0</td>
<td>-33.0</td>
</tr>
<tr>
<td>$M/s_2$</td>
<td>0.99</td>
<td>0.99</td>
<td>0.99</td>
<td>0.99</td>
</tr>
<tr>
<td>$\beta_2$</td>
<td>58.7</td>
<td>59.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>Baldwin-Lomax (B-L)</td>
<td>-</td>
<td>(B-L)</td>
<td></td>
</tr>
<tr>
<td>Governing equations</td>
<td>Navier-Stokes (2D)</td>
<td>Euler (2D)</td>
<td>Navier-Stokes (2D)</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.21: *Flow parameters, international standard configuration 11, transonic case*

4.1.3 Conclusions

The results show that the developed scheme gives an accurate prediction of the typical boundary layer properties, i.e. velocity distribution and skin friction on a flat plate for laminar and turbulent flow.

The studied flow cases around cascaded blades demonstrate the ability of the scheme to accurately resolve the flow field for separated flow as well as for viscous shock boundary layer interaction.
4.2 Unsteady results

4.2.1 Flat plate

Interblade phase angle $\sigma = 90.0$

Calculations on a flat plate are performed in order to compare the results obtained with the van Leer- and the AUSM-splitting. The numerical analysis is performed on a staggered row of flat plates with an inflow angle of 45 degrees and zero steady load. The inlet Mach number is 0.7. The results are given as amplitude and phase of the unsteady pressure, based on a Fourier decomposition for the fundamental frequency of the unsteady blade pressure, due to a torsional vibration around mid-chord with a reduced frequency of 1.0 based on full chord. The results, given in the figures 4.26 and 4.27, show good agreement between the van Leer scheme and the AUSM scheme and prove that the one-sided differences of the AUSM-Splitting do not introduce a phase error in the unsteady blade pressure. Hambraeus [1995] validated the van-Leer scheme for the flat plate against other results and obtained good agreement, e.g. LINSUB [Whitehead, 1987]. Therefore, it is assumed that the unsteady scheme developed by Hambraeus [1995] can serve as code to compare with and consequently the AUSM give the same results as the van-Leer scheme.

The resonance or cut-off conditions [Smith, 1971] for the two-dimensional flat plate configuration are $\sigma_{-\infty:+\infty} = -29.40^\circ$ and $\sigma_{+\infty:-\infty} = 107.26^\circ$.

The superresonant blade motion occurs at interblade phase angles between these cut-off values and sends a propagating wave in the upstream and/or downstream region of the flow. The blade motions at $-90^\circ \leq \sigma \leq -29.40^\circ$ and $107.26^\circ \leq \sigma \leq 270^\circ$ are subresonant.

Figure 4.25: Steady state Mach number distribution of standard configuration 11, off-design case
4.2.2 Stecf 5

2 degrees incidence  Steady and unsteady results for the two degrees case are given in the appendix in the paper by Höhn and Fransson [1996] and by Höhn [1996].

The cascade is forced into a pure torsional vibration mode of a frequency of 550 Hz and a amplitude of 0.1° with an interblade phase angle of 180°. The used steady state initial solution is characterized by an incidence angle of 2° and an inlet Mach number of 0.5. The results for the unsteady amplitude and phase of the surface pressure distribution are in good agreement with calculations based on FINSUP (see Whitehead [1990]). For this case there is no difference expected between the inviscid and the viscous solution, since the angle of attack is small and no separation is found in the steady state solution.

4 degrees incidence  The shown unsteady viscous flow situation is case 7 of standard configuration 5 according to Fransson and Verdon [1993]. The cascade is forced into a pure torsional vibration mode of a frequency of 550 Hz and an amplitude of 0.1° with
an interblade phase angle of 180° and a reduced frequency of 2.04 based on full chord. The steady state initial solution used is characterized by an incidence angle of 4° and an inlet Mach number of 0.5, see chapter 4.1.2. As the figures 4.28 and 4.29 show the results for the unsteady amplitude and phase of the surface pressure distribution are in reasonable agreement with experiments presented by Fransson and Verdon [1993]. The results are based on the fundamental Fourier coefficient of the unsteady pressure obtained by a Fourier analysis in time. The viscous computation is performed with the explicit parallelized unsteady version of the code on a IBM SP2 [Höhn, 1996]. The inviscid code, using the same capacitive boundary conditions and running for 12 cycles as well (which is found to be sufficient for an unsteady periodic solution), shows on the pressure side (LS) as expected no big difference to the viscous code for the unsteady amplitude. The difference in the leading edge region is due to a coarser mesh for the inviscid solution. The used mesh had 80 x 40 mesh points for the inviscid computations. On the other hand, the suction side shows a big difference, which is expected due to the fact that separation occurs in the viscous solution. Since the results cannot really be compared with the experiments where only one blade is excited \(^1\), a comparison with the Navier-Stokes code VOLFAP [Sidén, 1991] is made under the same conditions as for the results obtained with the present method. The results obtained with VOLFAP [Sidén, 1991] by Hedberg [1994], see figure 4.28, agree well with the present viscous method on the pressure side but generally have higher amplitudes for the unsteady pressure amplitude on the suction side, especially close to the leading edge where separation occurs. The results agree fairly well in phase with the present method for the suction side as well as for the pressure side, as can be seen in figure 4.29.

![Graph](image_url)

**Figure 4.28:** Amplitude of the unsteady pressure over chord for standard configuration 5 case 7, \(i = 4°\), reflecting boundary conditions applied in INSTHPT

Figure 4.30 and figure 4.31 show the unsteady pressure with respect to amplitude and

\(^1\)Consequently, the experiments show only the eigeninfluence of the blade on itself, which is shown to be the biggest contribution [Bölcs and Fransson, 1986]. The movement of neighboring blades give only a minor contribution on the amplitude of the unsteady pressure distribution and almost no influence on the unsteady phase.
Figure 4.29: Phase of the unsteady pressure over chord for standard configuration 5 case 7, $i = 4^\circ$, reflecting boundary conditions applied in INSTHPT

Figure 4.30: Amplitude of the unsteady pressure over chord for standard configuration 5 case 7, $i = 4^\circ$, nonreflecting boundary conditions applied in INSTHPT

phase for case 7 for INSTHPT and VOLFAP. The viscous and inviscid results for INSTHPT are obtained using the implicit time marching method with non-reflecting boundary conditions. In general, good agreement with VOLFAP is found with respect to the prediction of the unsteady amplitude and phase.

Comparing these results with the results obtained by the parallel, viscous, explicit version of the code using capacitive boundary conditions, see figures 4.28 and 4.29, one can see that the unsteady pressure amplitude is slightly higher for the calculations using non-reflecting boundary conditions. The predicted phase is almost the same for the computations performed with the explicit viscous unsteady version of INSTHPT and the implicit viscous unsteady version of INSTHPT, which leads to the conclusion that the influence of the boundary conditions, i.e. the use of reflecting or non-reflecting boundary conditions, is small for this particular test case.
4.2. UNSTEADY RESULTS

Figure 4.31: Phase of the unsteady pressure over chord for standard configuration 5 case 7, $i = 4^\circ$, nonreflecting boundary conditions applied in INSTHPT

6 degrees incidence The shown unsteady viscous flow situation is case 11 of standard configuration 5 according to Fransson and Verdon [1993]. The cascade is forced into a pure torsional vibration mode of a frequency of 550 Hz and an amplitude of 0.1° with an interblade phase angle of 180° and a reduced frequency of 2.04 based on full chord. The steady state initial solution used is characterized by an incidence angle of 5° and an inlet Mach number of 0.5 and given in the previous steady state section.

The unsteady results are presented as amplitude and phase lead towards the blade motion of the unsteady surface pressure distribution. This information is obtained by a Fourier analysis in time, which is performed on the surface pressure during one period of the blade motion based on the fundamental frequency, i.e. the frequency of the blade motion.

Figure 4.32 shows the amplitude of the unsteady surface pressure obtained on the suction and the pressure side of the blade for inviscid and viscous flow in comparison with experiments [Fransson and Verdon, 1993] and other numerical results obtained with the unsteady Navier-Stokes solver VOLFAP [Siden, 1991]. On the suction side the separated flow causes a peak in the unsteady pressure amplitude which is shown in the experiments and is predicted by the numerical methods. The inviscid result shows no peak at all on the suction side, which can be explained by the fact that no separation occurs in the inviscid steady solution. In general the amplitude for the unsteady pressure is lower for the present method and closer to the experiments than the results obtained with VOLFAP. On the pressure side the numerical results are one upon the other and are closer to the measured values.

The general tendency that the predictions of the three numerical results present are higher with regard to the amplitude of the surface pressure distribution is probably due to the fact that in the experiments only one blade is excited and consequently only the eigenfluence of the unsteady pressure on the blade itself is measured, for a detailed discussion on that issue see Bölcs and Fransson [1986].

Figure 4.33 shows the phase of the unsteady surface pressure. The numerical results
show good agreement to each other. The experimental results are slightly off on the suction side of the blade. However, the shift in phase on the suction side located at the end of the separation bubble is predicted by the numerical methods at the same position as measured in the experiments. The present inviscid results are in good agreement

Figure 4.32: *Amplitude of unsteady surface pressure distribution of standard configuration 5, case 11, i = 6°*

Figure 4.33: *Phase of unsteady surface pressure distribution of standard configuration 5, case 11, i = 6°*

with the other viscous predictions on the pressure side of the blade. On the suction side the steady separated flow in the viscous predictions lead to other unsteady results for the phase than the attached inviscid steady flow.
8 degrees incidence  The present unsteady viscous flow situation is case 12 of standard configuration 5 according to Fransson and Verdon [1993]. The steady state initial solution used is characterized by an incidence angle of 7.5° and an inlet Mach number of 0.5 and given in the previous steady state section. The cascade is forced into a pure torsional vibration mode of a frequency of 200 Hz and an amplitude of 0.1° with an interblade phase angle of 180° and a reduced frequency of 0.74 based on full chord.

Figure 4.34 shows the amplitude of the unsteady surface pressure obtained on the suction and the pressure side of the blade in comparison with experiments [Fransson and Verdon, 1993] based on a Fourier decomposition in time for the first harmonic. Moreover, results presented by Soize [1992] are shown, who couples a boundary layer code and an inviscid code on the suction side of the blade where separation occurs. On the pressure side an inviscid method alone is used. On the suction side the separated flow causes a peak in the unsteady pressure amplitude which is shown in the experiments and by Soize and not in the numerical predictions of the present method. The numerical predictions of INSTHPT shows an inviscid behavior with respect to the amplitude. The reason for this could be the completely separated flow of the steady state initial solution. The steady state results by Soize predict a length of the separation bubble of 39 per cent of chord, which is much lower than for the codes used in the previous steady state flow section.

Figure 4.35 shows the phase of the unsteady surface pressure. The experimental results are slightly off on the suction side of the blade, whereas the pressure side is in good agreement with the experiments. The boundary layer code by Soize is slightly off on the pressure side of the blade. The reason for that could be the fact that the flow is almost completely separated on the suction side of the blade which then affects also the pressure side of the neighboring blade and therefore the boundary layer assumption is not valid anymore.

![Graph showing unsteady surface pressure distribution](image-url)

Figure 4.34: Amplitude of unsteady surface pressure distribution of standard configuration 5, case 12, \( i = 8° \)
Figure 4.35: Phase of unsteady surface pressure distribution of standard configuration 5, case 12, $i = 8^\circ$.

10 degrees incidence This present unsteady viscous flow situation is case 18 of standard configuration 5 according to Fransson and Verdon [1993]. Based on the steady state initial solution which is characterized by an incidence angle of $10^\circ$ and an inlet Mach number of 0.5 and given in the previous steady state section, the cascade is forced into a pure torsional vibration mode of a frequency of 550 Hz and an amplitude of 0.1$\mu$ with an interblade phase angle of $180^\circ$ and a reduced frequency of 2.04 based on full chord.

Figure 4.36 shows the amplitude of the unsteady surface pressure obtained on the suction and the pressure side of the blade in comparison with experiments [Fransson and Verdon, 1993] based on a Fourier decomposition in time for the first harmonic. On the suction side the separated flow causes a peak in the unsteady pressure amplitude which is shown in the experiments and weakly shown in the numerical predictions. In general the amplitudes are higher than for the experiments, which is probably due to the fact that only one blade is oscillating in the experiments [ Bölcs and Fransson, 1986].

Figure 4.37 shows the phase of the unsteady surface pressure. The experimental results are slightly off on the suction side of the blade, whereas the pressure side is in good agreement with the experiments. Unfortunately, no published numerical data at all is available for this case.

Acoustic resonances The blade motions at $-90^\circ \leq \sigma \leq \min(\sigma^-_{\infty})$ and $\max(\sigma^+_{\infty}) \leq \sigma \leq 270^\circ$ are subresonant. The superresonant blade motion is occurring at interblade phase angles between these cut-off values and sends a propagating wave in the upstream and / or downstream region of the flow, see table 4.22.

Damping Figure 4.38 shows the computed aerodynamic damping as a function of the incidence angle for standard configuration 5 for the implicit inviscid and viscous codes of INSTHPT compared with the experimental data [Fransson and Verdon, 1993] and the inviscid code FINSUP [Hedberg, 1994] and the viscous code VOLFAP [Hedberg, 1994]. The tendency for the damping measured in the experiments is predicted by the viscous
Figure 4.36: Amplitude of unsteady surface pressure distribution of standard configuration 5, case 18, $i = 10^\circ$

Figure 4.37: Phase of unsteady surface pressure distribution of standard configuration 5, case 18, $i = 10^\circ$

code INSTHPT. However, for small incidence angles (less than $6^\circ$) lower damping values are predicted and for high angle of incidence higher positive damping rates are computed. The inviscid method of INSTHPT gives lower values for the damping than the viscous code of INSTHPT. The same tendency is shown by FINSUP and VOLFAP. No published numerical data is available for the angle of incidence of 7.5 and 10 degrees. In general the

<table>
<thead>
<tr>
<th>incid. angle</th>
<th>$\sigma^\pm_{-\infty}$</th>
<th>$\sigma^\pm_{+\infty}$</th>
<th>$\sigma^\pm_{-\infty}$</th>
<th>$\sigma^\pm_{+\infty}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$4^\circ$</td>
<td>$-39.60^\circ$</td>
<td>$109.37^\circ$</td>
<td>$-39.60^\circ$</td>
<td>$97.46^\circ$</td>
</tr>
<tr>
<td>$6^\circ$</td>
<td>$-40.56^\circ$</td>
<td>$118.73^\circ$</td>
<td>$-42.51^\circ$</td>
<td>$99.66^\circ$</td>
</tr>
<tr>
<td>$8^\circ$</td>
<td>$-14.60^\circ$</td>
<td>$43.50^\circ$</td>
<td>$-14.60^\circ$</td>
<td>$34.91^\circ$</td>
</tr>
<tr>
<td>$10^\circ$</td>
<td>$-41.84^\circ$</td>
<td>$141.82^\circ$</td>
<td>$-47.91^\circ$</td>
<td>$97.05^\circ$</td>
</tr>
</tbody>
</table>

Table 4.22: Acoustic resonance conditions standard configuration 5
viscous predictions are more stable in an aeroelastic sense than the inviscid predictions for the present method. This is confirmed by the calculations performed by Hedberg [1994] with FINSUP and VOLFAP.

![Graph showing damping as a function of incidence angle for standard configuration 5]

**Figure 4.38: Damping as a function of incidence angle standard configuration 5**

### 4.2.3 Stcf 10

The tenth standard configuration, included by a proposal of Dr. J.M. Verdon [1987], is a two-dimensional compressor cascade of modified NACA 0006 profiles that operate at subsonic inlet and outlet conditions. As aeroelastic test cases single degree of freedom blade bending, normal to chord, and pitching motion at a red frequency of 2 based on full chord and interblade phase angles of $0^\circ$ and $90^\circ$ are investigated.

The resonance or cut-off conditions for the two-dimensional configuration for transonic flow are $\sigma_{-\infty} = -28.93^\circ$ and $\sigma_{+\infty} = 201.70^\circ$ in the far upstream region and $\sigma_{-\infty} = -34.22^\circ$ and $\sigma_{+\infty} = 67.35^\circ$ in the far downstream region.

The superresonant blade motion occur at interblade phase angles between these cut-off values and send a propagating wave in the upstream and/or downstream region of the flow. The blade motions at $-90^\circ \leq \sigma \leq -34.22^\circ$ and $201.70^\circ \leq \sigma \leq 270^\circ$ are subresonant.

### Case 23

Based on the steady state results given in figure 4.20 unsteady computations are performed for a pitching vibration around midchord with an amplitude of two degrees and a reduced frequency of 2 based on full chord [Fransson and Verdon, 1993]. The results for the 1st harmonic of amplitude and phase of the unsteady blade pressure are shown in the figures 4.39 and 4.40 and compared with results from Grüber [Grüber and Carstens, 1996]. The
Figure 4.39: *Amplitude of unsteady surface pressure distribution of standard configuration 10, case 23*

Figure 4.40: *Phase of unsteady surface pressure distribution of standard configuration 10, case 23*

overall agreement of the results is good. Differences can be found in the vicinity of the shock on the suction side of the blade especially for the phase, where bigger deviations are found between the present methods. Reasons for that can be the bigger difference of the steady state results in that region as well as the fact that the present method INSTHPT uses first order differences for the spatial discretisation where a shock occurs, whereas Grüber still calculate with 2nd order accuracy in the shock region.

The aerodynamic damping is computed as 0.853 for INSTHPT against 0.984 obtained by Grüber. Ayer and Verdon [1996] calculated 0.948 with the viscous code NPHASE solving the unsteady thin layer Navier-Stokes equations, see table 4.23.

**Case 24**

The cascade is forced to pure pitching vibration of a reduced frequency of 2 based on full chord and an amplitude of 2.0 degrees with an interblade phase angle of 90 degrees. The
steady state solution used is characterized by an inlet Mach number of 0.8 and an inlet flow angle of 58 degrees, see previous section.

Figures 4.41 and 4.42 show the amplitude and phase of the unsteady surface pressure distribution based on the first Fourier component. The results are given for the numerical method LINFLO developed by Verdon [Usab and Verdon, 1990; Fransson and Verdon, 1993], which solves the unsteady linearized equations for potential flow, the inviscid results by the present implicit method and the viscous results by the explicit viscous method and implicit viscous code. The unsteady explicit viscous code applies boundary conditions based on the theory of characteristics [Ott, 1991], whereas the implicit viscous code uses locally non-reflecting boundary conditions [Chakravarthy, 1983]. According to Grüber [1999] the boundary conditions are approximately quasi 2D non-reflecting boundary conditions. This case shows that the boundary conditions are of particular importance for the tenth standard configuration. For the amplitude the inviscid methods are in good agreement on the pressure side of the blade. On the suction side LINFLO predicts a much higher unsteady pressure peak induced by the shock. Both inviscid methods use non-reflecting boundary conditions. The viscous results are different with respect to the unsteady amplitude, probably due to the fact that reflecting boundary conditions are applied in the viscous explicit unsteady calculations, whereas non-reflecting boundary conditions are used for the implicit calculations. The prediction of the peak of the un-
steady pressure amplitude caused by the shock is quite different. The explicit viscous code predicts the shock further upstream than the implicit viscous version of the code. It is believed that this is due to the boundary conditions. The prediction of the unsteady phase is in much better agreement for the different present methods. The shift in phase due to the shock is predicted at a similar position for all codes and corresponds to the position of the peak in the unsteady surface pressure amplitude. The linearized method by Verdon predict a lower phase shift caused by the shock than the other non-linear methods. Rather good agreement is found on the pressure side of the blade for all methods besides the viscous explicit prediction.

The aerodynamic damping is computed as 0.63 for INSTHPT against 0.884 obtained by Verdon, due to higher unsteady surface pressure amplitudes of Verdon. The viscous predictions of INSTHPT are quite close to each other, see table 4.24.

**Table 4.24: Damping, international standard configuration 10, case 24**

<table>
<thead>
<tr>
<th>$\sigma$</th>
<th>LINFLO</th>
<th>INSTHPT (inviscid)</th>
<th>INSTHPT (viscous, impl; expl)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$90^\circ$</td>
<td>0.884</td>
<td>0.63</td>
<td>0.684; 0.64</td>
</tr>
</tbody>
</table>

**Case 31**

The cascade is forced to pure bending vibration of a reduced frequency of 2 based on full chord and an amplitude of 0.01 times the chord length with an interblade phase angle of 0 degrees [Fransson and Verdon, 1993]. The steady state solution used is characterized by
an inlet Mach number of 0.8 and an inlet flow angle of 58 degrees.

Figures 4.43 and 4.44 show the amplitude and phase of the unsteady surface pressure distribution based on the first Fourier component, which has the frequency of the blade motion.

In the picture for the amplitude the results for the explicit inviscid and implicit viscous method INSTHPT are compared with results from Huff’s nonlinear Euler method and Verdon’s linearized potential flow solver LINFLO [Usab and Verdon, 1990; Fransson and Verdon, 1993]. The inviscid methods show a higher peak for the amplitude of the unsteady pressure than the viscous predictions on the suction side of the blade. This is due to the sharper prediction of the steady shock position. The viscous steady results smear out slightly, since the steady results show a shock boundary layer interaction. The unsteady peak of the shock shows a similar behavior, i.e. the amplitude is smeared out and smaller than for the inviscid predictions. On the other hand, the pressure side shows no differences between the four different results, which can be expected from the steady results where the computed Mach numbers are in good agreement to each other and the flow behaves essentially inviscid.

The phase shows in principle the same behavior. The peak in the phase on the suction side induced by the oscillating shock is moved upstream slightly and spread out as observed in the steady shock position and strength for the viscous results. Both indicates a shock boundary layer interaction. Further downstream close to the trailing edge the phase differs significantly again from the two inviscid solutions, which is probably due to the shock induced flow separation.

The results of the phase presented by Verdon with LINFLO could be due to the fact that present method applies a time-linearization in time and that this causes the differences of the phase prediction in the vicinity of the shock. However, results presented by Whitehead [Whitehead, 1990; Fransson and Verdon 1993] using a time-linearized solver for the potential flow equations show better agreement with the non-linear methods.

On the pressure side the phase shows for all computations the same behavior which results from the steady state flow where the flow predictions obtained with the present inviscid and viscous method and results of inviscid methods of other researchers, i.e. Huff et al. [1991] and Verdon [Usab and Verdon, 1990; Fransson and Verdon, 1993], are close to each other.

The aerodynamic damping is computed as 3.71 for INSTHPT for the viscous implicit code against 4.93 obtained for the explicit code of INSTHPT. Verdon [Usab and Verdon, 1990; Fransson and Verdon, 1993] calculated 5.0 with the inviscid code LINFLO, see table 4.25.

\[\text{Note that an anharmonic pressure response due to shock motion occurs because shocks are fitted in LINFLO [Verdon, 2000], which also has an influence on the results.}\]
4.2. UNSTEADY RESULTS

Figure 4.43: Amplitude of unsteady surface pressure distribution of standard configuration 10, case 31

Figure 4.44: Phase of unsteady surface pressure distribution of standard configuration 10, case 31

4.2.4 Stcf 11

Cases 656 and 134; Interblade phase angle $\sigma = 180.0$

The cases 656 (subsonic) and 134 (transonic) according to Fransson et al. [1998] \(^3\) are considered here. For the unsteady calculations rigid body blade vibrations in traveling

\(^3\)In the referenced paper the cases are denoted as case 100 and 200 instead of case 656 (subsonic) and 134 (transonic).

<table>
<thead>
<tr>
<th>$\sigma$</th>
<th>LINFLO</th>
<th>INSTHPT (viscous)</th>
<th>INSTHPT (inviscid)</th>
</tr>
</thead>
<tbody>
<tr>
<td>180°</td>
<td>5.0</td>
<td>3.71</td>
<td>4.93</td>
</tr>
</tbody>
</table>

Table 4.25: Damping, international standard configuration 10, case 31
wave mode are assumed with an interblade phase angle of 180 degrees. Pure bending perpendicular to the chord line is considered with an amplitude of 0.0054 and 0.0035 with respect to chord and a reduced frequency of 0.9163 and 0.7104 based on full chord and inlet velocity for the subsonic and the transonic case. Figure 4.45 and 4.46 show the unsteady pressure response for the subsonic and figure 4.47 and 4.48 for the transonic case with respect to the first harmonics obtained by a Fourier transformation in the time domain. For both cases all codes predict stable pressure response on the pressure side of the blade.

Figure 4.45: Amplitude of unsteady surface pressure distribution of standard configuration 11, subsonic case 656, $\sigma = 180.0$

Figure 4.46: Phase of unsteady surface pressure distribution of standard configuration 11, subsonic case 656, $\sigma = 180.0$

The amplitudes on the suction side and the pressure side of the blade for the subsonic
4.2. UNSTEADY RESULTS

<table>
<thead>
<tr>
<th>( \sigma )</th>
<th>FINSUP</th>
<th>INSTHPT (viscous)</th>
<th>INSTHPT (inviscid)</th>
<th>VOLFAP</th>
</tr>
</thead>
<tbody>
<tr>
<td>180°</td>
<td>10.8</td>
<td>9.2</td>
<td>10.18</td>
<td>8.4</td>
</tr>
</tbody>
</table>

Table 4.26: Damping, international standard configuration 11, subsonic case

<table>
<thead>
<tr>
<th>( \sigma )</th>
<th>FINSUP</th>
<th>INSTHPT (viscous)</th>
<th>INSTHPT (inviscid)</th>
<th>VOLFAP</th>
</tr>
</thead>
<tbody>
<tr>
<td>180°</td>
<td>8.4</td>
<td></td>
<td>11.5</td>
<td>8.9</td>
</tr>
</tbody>
</table>

Table 4.27: Damping, international standard configuration 11, transonic case

case are predicted in a right way with the codes. The viscous results for the present method INSTHPT are higher especially on the suction side compared with the results obtained with VOLFAP [Fransson et al., 1998]. The inviscid results obtained with the present method are much closer to the VOLFAP predictions. In the phase prediction on the pressure side differences are shown downstream of midchord for the present method and VOLFAP. Again the inviscid results computed with the present method are in better agreement with the VOLFAP results. Most of the experimental data show instability whereas the numerical results are more represented in the stable region.

In the transonic flow all codes predict the perturbation pressure due the impinging shock on the suction side, but at different positions and with different strength. However, fairly good agreement for the present method and VOLFAP is obtained with respect to amplitude and phase of the unsteady surface pressure. The unsteady peak of the pressure amplitude in the vicinity of the steady state shock position is higher than the predicted values of VOLFAP and at a different position. This is probably due to the different prediction of the shock position in the steady state results. The phase predictions of the present computations are in good agreement, however the measured phase distribution on the suction side in the shock region cannot be predicted.

The tables 4.26 and 4.27 summarize the predicted damping for the methods compared. The subsonic results are in good agreement, especially the viscous methods predict a lower damping than the inviscid codes. The transonic results do not agree as good as the subsonic results. The damping for the inviscid results of the present method are higher than the other predictions due to higher predicted amplitudes.

The blade motions at \(-90^\circ \leq \sigma \leq \min(\sigma_{\pm\infty})\) and \(\max(\sigma_{\pm\infty}) \leq \sigma \leq 270^\circ\) are subresonant else superresonant for the subsonic and transonic case, respectively, see table 4.28.

<table>
<thead>
<tr>
<th></th>
<th>subsonic</th>
<th>transonic</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \sigma_{-\infty} )</td>
<td>(-10.58^\circ)</td>
<td>(-9.60^\circ)</td>
</tr>
<tr>
<td>( \sigma_{+\infty} )</td>
<td>(12.40^\circ)</td>
<td>(15.11^\circ)</td>
</tr>
<tr>
<td>( \sigma_{-\infty} )</td>
<td>(-37.00^\circ)</td>
<td>(-3.70^\circ)</td>
</tr>
<tr>
<td>( \sigma_{+\infty} )</td>
<td>(7.61^\circ)</td>
<td>(7.38^\circ)</td>
</tr>
</tbody>
</table>

Table 4.28: Acoustic resonance conditions standard configuration 11
4.2.5 Conclusions

The unsteady results show the applicability of the developed method for massively separated stalled subsonic and moderately separated transonic flow.

The present results for standard configuration 5, i.e. a compressor test case, show reasonable agreement with other authors and the given experimental data for angle of incidences between 2 and 10 degrees.

The 10th standard configuration demonstrates the prediction capabilities of the method in cases where unsteady shock boundary layer interaction occurs.

Finally, the turbine case standard configuration 11 prove the scheme for unsteady subsonic and transonic turbine flow cases.
4.3 Comparison of INSTHPT with other codes

The numerical method UNSFLO is mainly developed by Giles [1991] and can be used for flutter and forced response predictions. The code contains state of the art techniques with respect to the numerical solver and the boundary conditions. Due to the use of the thin layer Navier-Stokes equations in an O-grid around the blade the use of the code should be limited to moderately separated flow. INSTHPT applies the full Navier-Stokes equations in the whole flow field and not only around the blade profile.

INSTHPT uses the AUSMD scheme [Wada and Liou, 1994] which is a further development of the principle AUSM method and reduces the splitting errors for contact surfaces [Wada and Liou, 1994] in comparison against the AUSM Mach number splitting [Kroll and Radespiel, 1993], which is used by Grüber and Carstens [1996]. The principle AUSMD scheme is more complicated than the AUSM Mach number splitting, which results in larger expressions for example for the inviscid Jacobian matrices.

The prediction tool SAFESI developed by Weber [1997] solves the unsteady Navier-Stokes equations for S1-stream surfaces around vibrating blades for an O-type grid at the blade embedded in an H-type mesh, which is used in the blade passage. Turbulence is modeled by the Baldwin-Lomax turbulence model. The governing equations are solved using a finite difference upwind scheme explicitly or implicitly in time. The phasen shift between adjacent blade rows is modeled by the direct store method [Erdos et al., 1997]. The present method INSTHPT solves the blade flutter problem on multiple blade passages, which allows solutions as vortex shedding which are not periodic in time. For this phenomenon the direct store method fails.

NHASE [Swafford et al., 1994] is a multi-block, finite-volume analysis that solves the two-dimensional, non-linear unsteady, viscous flow equations through vibrating cascades. In particular the so-called thin layer Navier-Stokes equations are solved, which restricts the code to weakly separated flow, using the Baldwin-Lomax turbulence model. The code is accelerated by an implicit time integration scheme. In the far-field reflecting boundary conditions are used. To avoid reflections the mesh is axially stretched at the inlet and outlet of the computational domain.

4.4 Summary of the papers in the appendix

The work in the thesis is based on four papers which are summarized in this chapter. In the first paper the basic method solving the unsteady Navier-Stokes equations is developed and applied towards steady and unsteady flow cases around cascaded vibrating blades commonly denoted as flutter. In paper II the method is extended and computations for unsteady separated flow cases are conducted. Paper III describes the extension of the code towards an fully implicit scheme and gives results for first test cases. Finally, paper IV considers computations performed with the scheme on massively separated subsonic flow and moderately separated transonic flow for two compressor test cases.
4.4.1 Ecumas 96

In this paper an unsteady, compressible, two dimensional Navier Stokes solver is used to predict the flow around cascaded vibrating blades. The present method applies the Advection Upstream Splitting Method to discretise the convective terms and central differences for the diffusive terms on a structured H-type mesh. Turbulence is modeled by using the Baldwin-Lomax turbulence model. The time accurate integration of the governing equations is performed by applying an explicit 4 stage Runge-Kutta scheme.

The code is validated by flat plate cases for laminar and turbulent flow at different Reynolds numbers, laminar flow around a 2-D cylinder, a turbine and a compressor cascade, both for highly subsonic, turbulent flow conditions. Unsteady inviscid and viscous calculations on these configurations show the applicability of the present method to the blade flutter phenomenon.

The overall comparison between the predictions and the test data is reasonably good.

4.4.2 ISUAAT 97

This paper considers the time accurate simulation of flutter using the unsteady Navier-Stokes equations and is a further development of paper one, which explained the developed method and showed first unsteady results. Here the same method is applied to moderately separated subsonic and transonic compressor flow cases. The time accurate integration of the governing equations is performed by applying an explicit four-stage Runge-Kutta scheme and some of the calculations are conducted on a IBM/SP2 parallel computer using the message passing system MPI. At the farfield boundary, viscous effects are assumed to be negligible and for steady state and the unsteady flow capacitive boundary conditions based on the theory of characteristics for the locally one-dimensional problem are used. Unsteady inviscid calculations are performed with the same code using non-reflecting boundary conditions.

Emphasis is laid on non-linear effects within the blade flutter phenomenon, i.e. cases where separation and shock boundary-layer interaction occurs in compressor cascades. In detail, the pressure response of the flow on imposed blade motions is presented for these flow situations. The steady state results agree well for viscous flow for the two shown test cases with other viscous and inviscid flowsolvers. Unsteady viscous flow calculations on these flow configurations show the applicability of the present method to the blade flutter phenomenon. For separated subsonic unsteady viscous flow good agreement with other flow solvers is found, whereas the test case for transonic viscous unsteady flow showed a big influence of the boundary conditions on the results.

4.4.3 AIAA 98

In the paper the extension of the code to a fully implicit method is described and first results for separated subsonic and transonic flow in a compressor cascade are shown. The principle method used is the same as in the two previous papers, i.e. the Advection Upstream Splitting Method for the convective terms and central differences for the dif-
fuse terms. The time integration is performed with the parallel version of the code for the steady state flow and solving the equations with the implicit, approximately factored Beam-Warming algorithm for the unsteady Navier Stokes equations. At the farfield boundary, viscous effects are assumed to be negligible and for steady state capacitive boundary conditions based on the theory of characteristics for the locally one-dimensional problem are used. Unsteady calculations are performed with the same code using locally non-reflecting boundary conditions.

Non-linear effects within the blade flutter phenomenon, i.e. cases where strong separation and shock boundary-layer interaction occurs in compressor cascades, are studied. In detail the pressure response of the flow on imposed blade motions is presented for these flow situations. The steady state results agree well for viscous flow for the two shown test cases with other viscous and inviscid flowsolvers. Unsteady viscous flow calculations on these flow configurations showed the applicability of the present method to the blade flutter phenomenon for separated flow cases.

4.4.4 AIAA 99

The fourth paper intends to use the numerical method developed in the papers I - III to investigate non-linear effects within the blade flutter phenomenon, i.e. stall and transonic flow regimes, and their influence on the unsteady blade pressure distribution for vibrating blades. Stall flutter, i.e. self excited oscillations at high angle of attack, is still not fully understood. Therefore, two compressor test cases, characterized by strong separation and shock boundary layer interaction, are studied.

The obtained results are investigated with respect to the pressure response of the flow on imposed blade motions in areas where viscous effects, i.e. stalled flow and shock boundary layer interaction, occur.

The computations of the steady state results for the unsteady cases under investigation agree well for viscous flow for the two shown test cases with other viscous and inviscid flowsolvers and available experiments. Unsteady viscous flow calculations on these flow configurations show the applicability of the present method on stalled flow and in cases where shock boundary layer interaction occur.
Chapter 5

Conclusions and Future Work

5.1 Summary of the work

The present work can be summarized in the following points:

- Establishing of the FFANET [Tysel and Hedmann, 1988] mesh generator at the chair [Höhn, 1995 a; Börjesson, 1997]

- Introduction of the viscous terms in INSTHPT and validation for laminar viscous flow in combination with the van Leer scheme towards the 2D, unsteady, viscous, compressible Navier-Stokes equations [Höhn, 1995 a]


- Introduction of the AUSM scheme [Wada and Liou, 1994] in a given Euler code [Hambraeus, 1995] and validation for different test cases [Höhn, 1995 b] for steady and unsteady inviscid and steady state viscous flow

- Study of the shock capturing capabilities for the AUSM scheme for steady and unsteady viscous and inviscid flow [Börjesson, 1997]

- Study of the mesh movement using different approaches for the mesh movement with emphasis on high amplitudes of the blade motion [Enbuske, 1996]

- Parallelising the code INSTHPT on an IBM SP2 using the message passing system MPI [Höhn, 1996]
  - non-reflecting boundary conditions for the inviscid version of INSTHPT
  - reflecting boundary conditions for the viscous version of INSTHPT

- Validating the code for unsteady inviscid and viscous flow around cascaded blades on the IBM SP2 [Höhn and Fransson, 1996 and Höhn, 1996]

- Unsteady boundary conditions considering primary and secondary gust effects [Höhn, 1997 b; Höhn, 1999]
• Implicit unsteady version of INSTHPT [Höhn and Fransson, 1998 b] following Beam and Warming [1978]
  – The method is extended to multiple blade passages.
  – Unsteady implicit boundary conditions [Chakravarthy, 1983]

• Introduction and validation [Khalifa, 1999] of local time stepping for steady inviscid and viscous flow

• Calculation of unsteady viscous separated flow with the unsteady viscous implicit version of INSTHPT [Höhn and Fransson, 1998 b]

• Coupling of INSTHPT with the ICEM grid generator package [Khalifa, 1999]

• Calculation of stalled flow with the present method [Höhn and Fransson, 1999]

Summarizing the various items one can conclude that INSTHPT is available for steady state computations using an explicit time integration method. The code is speeded up by using local time stepping for the inviscid and the parallel viscous version of the code. The unsteady code uses explicit/parallel time integration for the inviscid and viscous code. Furthermore one module provides the possibility of conducting implicit computations for inviscid and viscous flow applying non-reflection boundary conditions. All features of the codes are validated by Khalifa [1999]. The inviscid explicit version of the code is extended to a gust code [Höhn, 1997 b].

5.2 Conclusions

The present method shows generally good agreement for the investigated test cases in comparison with experiments and results obtained by other flow solvers. The applicability is shown for flat plate cases for laminar and turbulent flow, laminar flow around a 2-D cylinder and four Aeroelastic Standard Configurations for steady state flow conditions.

The laminar viscous results for flat plate cases show good agreement with the Blasius solution for different Reynolds numbers. The separational behavior of the numerical approach is checked by the calculation of a laminar, steady state flow around a 2-D cylinder.

Moreover, the turbulent test calculations on flat plates show that the implemented Baldwin-Lomax turbulence model works for the proposed flow conditions.

The results show that the developed scheme gives an accurate prediction of the typical boundary layer properties, i.e. velocity distribution and skin friction on a flat plate for laminar and turbulent flow.

The present method is easier to implement, computationally less expensive than the flux difference splitting methods, but has in the boundary layer still the same accuracy [Bergamini and Cinella, 1994]. Moreover, there is no need to specify a numerical dissipation in the code due to the upwind formulation.

For the viscous steady state, fully turbulent, calculations for different standard configurations are shown. The predicted surface pressure coefficients generally agree well with
the experiments and results obtained by other flow solvers. Especially, flow cases where separation (standard configuration 5) and shock boundary layer interaction (standard configuration 10 and standard configuration 11) occur are investigated.

The studied flow cases around cascaded blades demonstrate the ability of the scheme to accurately resolve the flow field for separated flow as well as for viscous shock boundary layer interaction.

Unsteady inviscid calculations on cascaded blades, i.e. standard configuration 4, with the AUSM scheme are conducted [Hölm, 1996] and agree with the former used van Leer flux-vector splitting scheme [Hambraeus, 1995].

Thereafter, there are computations performed on STCF 5 at the following angles of incidences, i.e. 2°, 4°, 6°, 8° and 10° and the compressor configuration STCF 10 for subsonic and transonic flow. Finally, the new 11th standard configuration is studied. The calculations on the international standard configuration 5 show the applicability of the method for the blade flutter problem. In the separated regions the results obtained with the viscous solution differ from the inviscid ones, which indicates that the viscous formulation of the code accounts for unsteady viscous effects. Unfortunately, the experiments are performed for a single blade vibration, so it is not possible to really compare with them. However, good agreement is found with other numerical methods [Siden, 1991; Soize, 1992].

The calculation on the standard configuration 10 is performed for an inlet Mach number of 0.7 and 0.8. Good agreement is found with other researchers [Griüber and Carstens, 1996]. This standard configuration demonstrates the prediction capabilities of the method in cases where unsteady shock boundary layer interaction occurs. The predictions for the 11th standard configuration are in good agreement with VOLFAP [Siden, 1991] for for unsteady subsonic and transonic turbine flow cases.

The unsteady results show the applicability of the developed method for massively separated stalled subsonic and moderately separated transonic flow.

5.3 Future work

The following things could be done in order to improve the prediction capabilities of the present method:

- The Baldwin-Lomax turbulence model can be used for attached flows and weakly separated flows, but is not designed for large separation. A next step can be the replacement of the model by a two-equation model, for instance the k-ε model. Though it is also based on Boussinesq hypothesis [Boussinesq, 1877], i.e. the neglect of the rotational part of the strain rate tensor [Wilcox, 1993], it gives better results in separated areas solving transport equations for the turbulent length scale and the turbulent velocity scale.

- The code should be extended to Q3D or 3D for even more realistic flow calculations, i.e. the stream tube contraction in modern axial turbines.
The program should be speeded up further more. The following things could be implemented in order to be able to study other flow cases, i.e. the cases with small reduced frequency:

- Combining the available parallel code version with the implicit module
- Residual smoothing [Hirsch, 1990]
- Multi grid method [He, 1993]

Presently, the pressure is extrapolated with 1st order to the dummy cells on the blade in the inviscid and viscous code. Due to the fact that the calculation of this value is most important for the blade flutter problem it should be investigated if a more advanced extrapolation of this value, for instance the use of the normal momentum equation for unsteady Navier-Stokes equations according to Steger [1978] would improve the calculation of the pressure on the blade.

The transonic case for standard configuration 10 is investigated for a mesh with normal mesh lines on the suction side of the blade. It should be investigated if a flux limiter in two directions and consequently a shock detection in both direction of the computational domain would make it possible to use the same mesh as for the subsonic case.

The influence of neighboring blade rows simulated by gust boundary conditions on the flutter stability should be investigated, applying the developed boundary conditions described in the thesis in appendix E.

Based on the work of Hambraeus [1995] the non-reflecting boundary conditions used here should be tested if they are really highly non-reflecting.

In a future work it should be investigated how neighboring blade rows simulated as gust boundary conditions effect the stability of oscillating blade rows with respect to flutter.
Bibliography


[14] Blasius, H.; 1908


[16] Boussinesq, T. V.; 1877


"Aeroelasticity in Turbomachines, Comparison of Theoretical and Experimental Cascade Results.", Communication de Laboratoire de Thermique Appliqué et de Turbomachines, No. 1, École Polytechnique Fédéral de Lausanne, Switzerland, 1986.


[22] Carta, F. O.; 1982

[23] Carstens, V.; 1994


[26] Cebeci, T.; Bradshaw, P.; 1977

[27] Cebeci, T.; Smith, A. M. O.; 1974


[29] Chen, K.K.; Thysson, N. A.; 1971


[31] Coakley, T.J.; 1983


[34] Dorney, D. J.; Verdon, J. M.; 1994


[37] Erdos, J.L.; Alzner, E.; McNally, W.; 1977

[38] Eguchi, T.; Wiedermann, A.; 1995


[40] Fleeter, S.; Capece, V.; 1984


[42] Fransson, T. H.; 1986


"Viscous and Inviscid Linear/Nonlinear Calculations Versus Quasi 3D Experimental


[50] **Försching, H.; 1974**

[51] **Fristedt, E.; Tengwall, C. G.; 1995**

[52] **Gerolymos, G. A.; 1988**

[53] **Giles, M.; 1986**

[54] **Giles, M.; 1988**

[55] **Giles, M.; 1989**


[57] **Glauert, H.; 1928**

[58] **Groth, J. P.; Martensson, H.; Eriksson, L.-E.; 1996**

[59] **Grüber, B.; 1999**

[60] **Grüber, B.; Carstens, V.; 1996**

[61] **Grossmann, B.; 1986**
"Using MPI - Portable Parallel Programming with the Message Passing Interface.”,

"EUROVAL-An European Initiative on Validation of CFD Codes.”,

[64] Hall, K.C.; 1987
"A Linearized Euler Analysis of Unsteady Flows in Turbomachinery.”,

[65] Hall, K.C.; Clark, W.S.; 1991
"Prediction of Unsteady Aerodynamic Loads in Cascades Using the Linearized Euler
Equations on Deforming Grids.”,
AIAA Paper 91-3378.

"Calculation of Three-Dimensional Unsteady Flows in Turbomachinery Using the
Linearized Harmonic Euler Equations.”,

"Non-linear Euler Calculations with Improved Farfield Boundary Conditions for Un-
steady Flow Through Vibrating Cascades.” Licentiate Thesis, Royal Institute of Tech-

"On the Accuracy of Upwind Schemes for the Solution of the Navier-Stokes Equa-
tions.”

"An Implicit Flux-Vector Splitting Scheme for the Computation of the Viscous Hyp-
ersonic Flow.”

[70] Hedberg, K.; 1994
"Validation of Flutter Prediction Models Against Experimental Data for Standard
Configuration 5.”,
Diploma Thesis, Chair of Heat and Power Technology, Royal In-

[71] He, L.; 1993
"New Two-Grid Acceleration Method for Unsteady Navier-Stokes Calculations.”,

[72] He, L.; Denton, J.D.; 1993
"Inviscid-Viscous Coupled Solution for Unsteady Flows Through Vibrating Blades:
Part I & II.”,

[73] He, L; 1989
"An Euler Solution for Unsteady Flows Around Oscillating Blades.”,
[74] **Hirsch, C.; 1990**  

[75] **Holmes, D.G.; Mitchell, B.E.; Lorence, C.B.; 1998**  

[76] **Höhn, W.; 1995 a**  
"Extension of a Two Dimensional Flow Cascade Solver from Inviscid to Viscous: Continued Calculations with the Program INSTHPT.” Internal Report 95/02, Royal Institute of Technology, Chair of Heat and Power Technology, Stockholm, 1995.

[77] **Höhn, W.; 1995 b**  

[78] **Höhn, W.; Fransson, T.H.; 1996**  

[79] **Höhn, W.; 1996**  

[80] **Höhn, W.; 1997 a**  

[81] **Höhn, W.; 1997 b**  

[82] **Höhn, W.; 1997 c**  

[83] **Höhn, W.; 1998**  

[84] **Höhn, W.; Fransson, T.H.; 1998 a**  
"Flutter Analysis of Two Dimensional Viscous Subsonic and Transonic Flow in Turbomachines Using the Advection Upstream Splitting Method.”, 8th Symposium on

[85] Höhn, W.; Fransson, T.H.; 1998 b
"Flutter Analysis in Turbomachines Using a Modified Flux Vector Splitting Scheme."

"Flutter Analysis of Separated Flow in Axial Turbomachines."

[87] Höhn, W.; 1999
"Unsteady Reflecting Boundary Conditions Considering Primary and Secondary Gust Effects."

"Numerical Validation of Flow Solver VOLSOIL at HPT/KTH."

"Numerical Analysis of Supersonic Flow Through Oscillating Cascade Cascade Sections by Using a Deforming Grid."

"Euler Flow Predictions for an Oscillating Cascade Using a High Resolution Wave-Split Scheme."

"Description of a Parallel Unstructured 3D Navier-Stokes Code for Complex Geometry Turbomachinery CFD."
EPFL-LTT, Lausanne (CH) and ABB, Baden (CH), 1995.

[92] Isomura, K.; 1996
"A Numerical Investigation of a Flutter in a Transonic Fan."

"Transonic Aerofoil Calculations Using the Euler Equations."

"F100 Fan Stall Flutter Flutter Problem Review and Solution."

[95] Joubert, H.; 1984
"Supersonic Flutter in Axial Flow Compressor."
"Numerical Study of the Flutter Behavior of a Turbine Cascade at Off-Design Conditions.", to be submitted.


[99] Kau, H. P.; 1990

[100] Khalifa, A.; 1999


[103] Küssner, H. G.; 1929

[104] Leyland, P.; Ott, P.; Richter R.; 1994

[105] Lubomski, J. F.; 1980


"A Review of Turbomachine Troubles and Their Causes.", North East Institution of Engineers and Ship Builders, Newcastle-on-Tyne, February 16th, 1923.

[109] Namba, M.; 1977

[110] Ni, R.H.; Sisto, F.; 1976

[111] Ni, R. H.; 1979


[113] Ott, P.; 1991


[115] Polilhausen, E.; 1921

[116] Pearson, H.; 1953

[117] Platzer, M.F.; Carta, F.O.; 1987

[118] Platzer, M.F.; Carta, F.O.; 1988


[121] Rizzi, A.; 1994


[123] Rolls-Royce plc.; 1986

[124] Salaün, P.; 1987

[125] Shannon, J. F.; 1945

[126] Schlichting, H.; 1987


[128] Smati, L., Aubert, S., Ferrand, P.; Massao; 1998

[129] Smith, N. S.; 1971

[130] Soize, C.; 1992

[131] Srinivasan, A. V.; 1997
[132] Steger, J. L.; 1978


[134] Theodorsen, T. H.; 1935

[135] Törnig, W.; Spallucci, P.; 1988


[139] Van Leer, B; 1982

[140] Venkatakrishnan, V; 1993


[142] Verdon, J. M.; 1986

[143] Verdon, J. M.; 1987

[144] Verdon, J. M.; 1993

[146] Viviand, H.; 1974

[147] Wada, Y.; Liou, M. S.; 1994
"A Flux Splitting Scheme with High-Resolution and Robustness for Discontinuities.”


