# Table of Contents

1 Introduction 4

2 Literature Review 5
   2.1 State-of-the-Art Reviews ................................................................. 5
   2.2 Throughflow ......................................................................................... 5
   2.3 Quasi-Steady Rotor-Stator Interface (Mixing Plane) ........................... 9

3 Numerical Method 16
   3.1 The EURANUS/Turbo Code ................................................................. 16
      3.1.1 Overview ....................................................................................... 16
      3.1.2 Basic Numerical Method ............................................................... 16
   3.2 Turbomachinery-Specific Implementation Aspects ............................... 18
   3.3 Software Engineering and Post-Processing ......................................... 18
   3.4 Non-matching Connecting Boundaries .............................................. 19

4 Mesh Dependence Analysis 21
   4.1 Introduction. ......................................................................................... 21
   4.2 Motivation. .......................................................................................... 21
   4.3 Preliminary Studies on Mesh Influence .............................................. 22
      4.3.1 Prerotation ..................................................................................... 22
      4.3.2 Two-dimensional blade-to-blade analysis ...................................... 22
   4.4 3D Computations ............................................................................... 24
      4.4.1 Grid generation ............................................................................... 24
      4.4.2 Computed cases ............................................................................ 25
      4.4.3 Numerical Aspects ........................................................................ 25
      4.4.4 Convergence .................................................................................. 26
      4.4.5 Comparison with experimental data .............................................. 26
      4.4.6 Global quantities .......................................................................... 27
      4.4.7 Pressure distribution ..................................................................... 27
      4.4.8 Generation and distribution of losses .......................................... 27
      4.4.9 Backflow and prerotation ............................................................... 28
   4.5 Conclusions. ....................................................................................... 28
   4.6 Effect of Spatial Discretization in Viscous Flow ................................. 37
      4.6.1 Reflections on sources of error ....................................................... 37
      4.6.2 Presentation of the test case ............................................................ 37
      4.6.3 Description of the meshes .............................................................. 38
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.6.4 Tested spatial discretization schemes</td>
<td>39</td>
</tr>
<tr>
<td>4.6.5 Other numerical parameters</td>
<td>42</td>
</tr>
<tr>
<td>4.6.6 Convergence</td>
<td>43</td>
</tr>
<tr>
<td>4.6.7 Cost</td>
<td>43</td>
</tr>
<tr>
<td>4.6.8 Results</td>
<td>44</td>
</tr>
<tr>
<td>4.6.9 Conclusions</td>
<td>46</td>
</tr>
<tr>
<td>4.6.10 Related studies in the literature</td>
<td>46</td>
</tr>
<tr>
<td>4.7 A Matrix Dissipation for Incompressible Flow</td>
<td>54</td>
</tr>
<tr>
<td>4.7.1 Mathematical formulation</td>
<td>54</td>
</tr>
<tr>
<td>4.7.2 Evaluation with the laminar flat plate</td>
<td>60</td>
</tr>
<tr>
<td>5 An Euler Throughflow Model</td>
<td>62</td>
</tr>
<tr>
<td>5.1 Introduction</td>
<td>62</td>
</tr>
<tr>
<td>5.2 The Euler Throughflow Model</td>
<td>63</td>
</tr>
<tr>
<td>5.2.1 Governing Equations</td>
<td>63</td>
</tr>
<tr>
<td>5.2.2 The Blade Force</td>
<td>64</td>
</tr>
<tr>
<td>5.2.3 The Friction Force</td>
<td>65</td>
</tr>
<tr>
<td>5.2.4 Treatment of the Blockage Factor</td>
<td>66</td>
</tr>
<tr>
<td>5.3 A Robust Formulation of the Analysis Mode</td>
<td>67</td>
</tr>
<tr>
<td>5.3.1 Classical Model</td>
<td>67</td>
</tr>
<tr>
<td>5.3.2 Robust Model</td>
<td>68</td>
</tr>
<tr>
<td>5.4 Other Innovations</td>
<td>70</td>
</tr>
<tr>
<td>5.4.1 An Analysis Mode with Throat Control</td>
<td>70</td>
</tr>
<tr>
<td>5.4.2 Flow angle distribution for high-turning turbines</td>
<td>72</td>
</tr>
<tr>
<td>5.5 Throughflow Shock Capturing Properties</td>
<td>72</td>
</tr>
<tr>
<td>5.5.1 Preliminary Remarks</td>
<td>72</td>
</tr>
<tr>
<td>5.5.2 2D Euler Equations</td>
<td>72</td>
</tr>
<tr>
<td>5.5.3 Associated Throughflow Equations</td>
<td>73</td>
</tr>
<tr>
<td>5.5.4 The Hybrid Mode</td>
<td>76</td>
</tr>
<tr>
<td>5.6 Shock Structure in Supersonic Blade Passages</td>
<td>76</td>
</tr>
<tr>
<td>5.6.1 General Discussion</td>
<td>76</td>
</tr>
<tr>
<td>5.6.2 Control Volume Analysis</td>
<td>78</td>
</tr>
<tr>
<td>5.6.3 Recent Numerical Evidence</td>
<td>79</td>
</tr>
<tr>
<td>5.6.4 Conclusions</td>
<td>80</td>
</tr>
<tr>
<td>5.7 The Relevance of Captured Throughflow Shocks</td>
<td>81</td>
</tr>
<tr>
<td>5.7.1 Possible Interpretations and Modelling Approaches</td>
<td>81</td>
</tr>
<tr>
<td>5.7.2 Pitch-Averaged Representation of a Blade-to-Blade Normal Shock</td>
<td>81</td>
</tr>
<tr>
<td>5.7.3 Shock Comparison for Two Representative Operating Points</td>
<td>84</td>
</tr>
<tr>
<td>5.7.4 Conclusions</td>
<td>90</td>
</tr>
<tr>
<td>6 Transonic Axial Compressor Rotor</td>
<td>97</td>
</tr>
<tr>
<td>6.1 General Presentation of the Test Case</td>
<td>97</td>
</tr>
<tr>
<td>6.2 Preliminary Analysis at Peak Efficiency Operation</td>
<td>99</td>
</tr>
<tr>
<td>6.3 Detailed Analysis of the Design Speed Operating Line</td>
<td>105</td>
</tr>
<tr>
<td>6.3.1 Analysis of the Pitch-Averaged 3D Flow</td>
<td>105</td>
</tr>
</tbody>
</table>
# Table of Contents

6.3.2 Throughflow Calculation Strategy .................................................. 111
6.3.3 Comparison with Pitch-Averaged 3D Flow Fields ............................... 113
6.4 Conclusions ......................................................................................... 114

7 Other Throughflow Test Cases ................................................................ 124
  7.1 Four-stage low speed turbine ............................................................... 124
  7.2 Transonic Turbine Vane ...................................................................... 126
  7.3 Supersonic Impulse Turbine ................................................................. 130
  7.4 Bypass Turbofan Engine ..................................................................... 136
  7.5 Conclusions ......................................................................................... 145

8 Condensable Fluids .................................................................................. 146
  8.1 Thermodynamic Interpolation Tables .................................................. 146
    8.1.1 Methodology .................................................................................. 146
    8.1.2 Implementation .............................................................................. 146
    8.1.3 Interpolation and Table Types ....................................................... 149
    8.1.4 Combinations of Variables ........................................................... 150
  8.2 Application to Steam ........................................................................... 152
    8.2.1 Introduction .................................................................................. 152
    8.2.2 Steam Tables and Steam Properties Formulation .......................... 152
    8.2.3 Switched Condensation Model ...................................................... 153
    8.2.4 Ansaldo 4-stage LP Turbine ......................................................... 155
    8.2.5 Nucleating Nozzle Flow ............................................................... 158
    8.2.6 Conclusions .................................................................................. 161
  8.3 Application to LH2 .............................................................................. 162
    8.3.1 Motivation .................................................................................... 162
    8.3.2 Boundary Conditions for LH2 ...................................................... 162
    8.3.3 Validation of the Condensable Fluid Model with LH2 .................. 164
    8.3.4 LH2 Turbo Pump .......................................................................... 171

9 Conclusions ............................................................................................. 197

Bibliography ................................................................................................. 201

A The Mathematical Formulation of the System of Euler Throughflow Equations
  A.1 2D Non-linear Convection .................................................................. 213
  A.2 The 2D Euler Equations .................................................................... 215

B Equations Used for Shock Comparisons .................................................. 224

C Technical Details of Table Interpolation ............................................... 227
Chapter 1  Introduction

The work presented in this thesis touches upon a range of questions which are of fundamental importance to the accurate numerical prediction of fluid flow. An economically most important application are the complex internal flows in turbomachines. A detailed understanding of these flows becomes increasingly significant where efficiency must be maximized, as in power generation, and where systems must operate near their technological limit in a narrow window of opportunity. A prime example for the latter case is liquid fuel rocket propulsion for space flight, which is enabled by high-performance turbopumps. Apart from and complementary to (usually expensive) experiments, the only way to gain an understanding and therefore control of the numerous flow phenomena is through numerical calculation (frequently called simulation), there being no known analytical solutions to the governing Navier-Stokes equations for real-life flows.
Chapter 2 Literature Review

The economic importance and ubiquitous use of turbomachines meant that the calculation of the complex flows through them has, over this past century, attracted an enormous and still increasing research and development effort, resulting in an abundant body of literature and a wide range of numerical methods for flow analysis and design. For a global overview, we shall refer to a number of survey papers and focus instead on small number of special topics relevant for the present work. These are throughflow models, mesh dependence, the quasi-steady coupling of multiple blade rows, and the treatment of the thermodynamic properties of condensable fluids.

2.1 State-of-the-Art Reviews

Since numerical solution methods for flow problems have found widespread use, the state-of-the-art with a perspective to turbomachinery has been reviewed at regular intervals, for example by Japikse (1976), McNally and Sockol (1985), Lakshminarayana (1991), Hirsch (1994), and by Denton and Dawes (1999). Further discussions of the different mathematical models applicable to the calculation of the flow in turbomachines, spanning the range of dynamic levels of approximation from the full time-dependent Navier-Stokes equations to linearized potential flow models and singularity methods, can be found in Cumpsty (1989). A more comprehensive discussion of flow models, from a more general perspective, is given by Hirsch (1988). Denton (1985a), in an introduction to the application of the Euler equations to turbomachinery flows, gives a concise but very clear summary of the classical flow models and solution methods (streamline curvature, stream function, and potential flow), discussing the properties, advantages, drawbacks, and limitations of each (as applied to 2D flow).

2.2 Throughflow

Throughflow methods may be classified by the dynamic level of approximation, that is, which equation or set of equations is solved. Equations which are not solved are replaced by assumptions on the flow, for example, the constancy of total enthalpy and entropy along streamlines and an imposed flow angle or swirl distribution.

The concept of the meridional throughflow appears to have been introduced in the systematic and farsighted work of Wu (1951, 1952). With his quasi-3D stream surface description, Wu was the first to formally derive a consistent, theoretically grounded model for both the blade-to-blade flow and the meridional throughflow. He obtained an equation for the stream function which he proposed to solve by a finite difference method.

However, it was the streamline curvature method, which takes its name from the prominent role the radius of curvature of meridional streamlines plays in the one solved equation and iterative solution algorithm, which first found widespread use in practice and has seen enormous development efforts consecrated to it. Probable reasons may be the intuitive character of this approach and its immediate link with blade design parameters. Consisting of two nested integrations, an inner loop to satisfy the mass flow condition at each axial computing station, and an outer loop to find the shape of a finite number of meridional streamlines, it can handle limited regions of supersonic flow, which the matrix and streamfunction methods cannot (practically) do.

In both the streamline curvature method and the streamfunction method, the orginal set of five partial differential equations is, through a number of simplifying assumptions, reduced to a single partial differential equation (the so-called radial equilibrium equation for the SLC method (for one (usually the circumferential) velocity component) and the streamfunction equation for the streamfunction method), making them conceptually unable to predict choking or to capture shocks.

Although techniques have been devised to partially overcome the SLC method’s inability to predict choking, e.g. by Denton (1978) for turbines (making the assumption that the change in streamtube area between the throat and the trailing edge can be neglected and thus placing the throat at the trailing edge) and by Saryari and Bölcs (1995) for compressors, some limitations still remain, regarding for instance numerical robustness and resolution limits. The classical SLC and streamfunction methods are, however, still applied for example by (recent references).
Most recently, a combination of factors has naturally led to the emergence of throughflow models based on the solution of the Euler equations. These factors are: the limitations inherent in the two classical throughflow methods, the rapidly increasing choice and maturity of (more or less) efficient and robust algorithms to solve the Euler equations, and ever-increasing computing resources. The present author is aware of the following methods:

**Spurr (1980)**

A first throughflow method based on the Euler equations has already been presented in 1980 by Spurr. Spurr’s interest was in the application to steam turbines (calculated with the perfect gas), where extremely high Mach numbers (up to about 2) indeed pose severe challenges for traditional throughflow methods. The finite volume spatial discretization is solved through the original form of the so-called opposed difference technique introduced by Denton (1975, 1983), a simplified first order upwind scheme combined with a higher-order correction in which density, enthalpy, and the velocity vector are upwind-interpolated and pressure is downwind-interpolated. As a side note, this same philosophy would later be developed into the AUSM schemes by Liou and Steffen (1993). A stable time integration procedure is arrived at by solving the equations in a special order, immediately using updated variables as they become available. The most remarkable aspect of this early Euler throughflow method, however, is the fact that Spurr adheres to the time-marching philosophy consequently and applies it also to the blade force. Instead of directly obtaining the tangential component of the blade force from the tangential momentum equation, which would then not be solved inside blade passages, it is updated at each iteration from the additional equation (\(\Delta T\) in denominator?)

\[
\Delta f_{B, \theta} = \frac{\Delta V_{\theta}}{\Delta t}
\]  

where \(\Delta V_{\theta}\) is the difference between the value obtained from the tangential momentum equation and the that given by \(V_r, V_z\), and the imposed flow angle. The present work clearly confirms the superiority of this approach. As implemented by Spurr, with the conventional cylindrical source terms absorbed into the blade force and placing a single large control volume between the blades, the model requires the blade geometry (at least three sections for twisted blades) as input and therefore is only applicable in analysis mode; it has no design mode capability. Neither losses nor deviation have been considered, but the throughflow model has been successfully combined with a blade-to-blade module into a Q3D system, which was regarded as a preliminary step towards full 3D (judged too expensive). Especially the absence of stream surface twist in the blade-to-blade module is seen as a distinct limitation.

**Nigmatullin and Ivanov (1994)**

Within the larger framework of a design and analysis system for aeroengines, Nigmatullin and Ivanov (1994) also present a throughflow model based on the Euler equations. Although it is again limited to the analysis mode, it is interesting in two respects. First, it uses a monotone second order Godunov scheme and therefore represents one of the very few examples for the use of exact Riemann solvers in an industrial CFD code. The Godunov scheme enables a locally exact treatment of the discontinuities which may arise along the blade edges separating (and joining) zones in analysis mode (the blade passage) from zones in design mode (the connecting ducts) by solving a modified Riemann problem on the connecting cell face (which also must accomplish a jump in tangential momentum at the blade leading edges to accommodate varying incidence angles because a fixed flow angle is imposed throughout the full length of the passage). Second, the authors recognize the possibility of a discontinuous blade force where shocks are captured in analysis mode and point out an elegant solution exploiting the axisymmetry inherent in the throughflow model by giving up spatial continuity of the finite control volume. Without this modification, conservation errors would risk to be introduced because the blade force is algebraically determined from the tangential momentum equation, the velocity vector being constrained to the frozen S2 surface at all times. Time discretization is implicit and the resulting system solved by Gauss-Seidel line relaxation. The backgound of the authors being in aeroengines, the model is applied to transonic axial compressors and turbines.
**Stewart (1995)**

The ability of throughflow models based on the Euler equations to predict choking, capture shocks which respond to variations in back pressure, and to cope with complex, non-smooth geometries makes them a promising candidate for complete engine simulations, yielding a much improved physical model compared with the standard 1D analytical models, which also provides a consistent environment for higher-level component simulations. A first sketch of such a system is presented by Stewart (1995), who applies it to the simulation of a complete turbofan engine. Inside blade passages, a fixed flow direction taken from SLC or 3D flow simulations is imposed, which obviously contributes to the stability of the method. The blade force is time-dependent and updated from the following equation ($\Delta T$ in denominator?),

$$
\Delta f_B = \frac{\rho}{\Delta t} [(\mathbf{W} \cdot \mathbf{I}_{\text{Target}}) \mathbf{I}_{\text{Target}} - \mathbf{W}]
$$

representing a correction proportional to the deviation from the imposed (Target) flow direction. (Unusually, both the tangential and the meridional flow angle are imposed.) The set of five Euler equations is solved by a standard JST (1981) scheme (explicit Runge-Kutta time integration in combination with a second order central scheme) in parallel with a torque equation. The numerical robustness of the complete engine simulation system is determined by the underlying throughflow model. As implemented by Stewart, it requires an elaborate start-up procedure, and problems with numerical stability can be expected to worsen as more realism is introduced in the model, for example in the form of incidence and deviation corrections, relaxing the condition of a frozen flow direction.

Given its demonstrated potential as a corner-stone of advanced engine design systems (tying together all stages from predesign to final operating analysis), the Euler throughflow model therefore merits an in-depth analysis of its properties, to which the present work hopes to contribute.

**Boure and Gillant (1995)**

Boure and Gillant (1995) proposed an Euler throughflow model based on the finite volume Lax-Wendroff scheme using the cell vertex implementation of Ni (1982). Theirs is the only published Euler throughflow model to incorporates a dynamic unique incidence model, based on throat location and captured shock location, thus effectively limiting the imposed flow angle, which otherwise is allowed to adapt freely to the direction of the incoming flow over a small fraction of chord. The asserted need for unique incidence is in consideration with an analysis by Freeman and Cumpsty (1992), at least for the single presented transonic compressor rotor for which the tip relative Mach number remains below about 1.3 where passage choking should occur, the additional external input of a unique incidence angle, depending on how it is derived, does contain some additional information on the blade passage geometry that would not otherwise be visible to the throughflow model.

Boure and Gillant’s model represents a classical approach in so far as the blade force is algebraically determined from the tangential momentum equation,

$$
\rho b f_{B, \theta} = \rho b f_{F, \theta} - \nabla \cdot (\rho b \nabla V_\theta)
$$

the tangential velocity being hard-linked to the known meridional velocity and flow angle at all times through the relation $W_\theta = V_m \tan \beta$. As a perhaps important detail of the implementation, even though the tangential momentum equation is not solved inside blade passages, a corresponding residual is still required in the dissipation term of the Ni scheme through the product of the flux jacobian matrix and pseudo-vector of residuals. Rather than neglecting this contribution, Boure and Gillant assume the $\theta$-momentum residual to be proportional to the r- and z-momentum residuals according to the imposed flow direction. At high flow angles, which moreover tend to occur in the sensitive transonic and supersonic regions at compressor blade leading edges and turbine blade trailing edges, the contribution from $V_\theta$ in fact becomes dominant and omitting it might lead to insufficient dissipation and hence degraded numerical stability.

The application envisaged by the authors is off-design performance prediction for axial transonic compressors, exploiting the mass flow prediction and choking ability of the Euler throughflow
model. Qualitatively correct speed lines for total pressure ratio and efficiency are presented for one single test case. The authors observe the extreme sensitivity of predicted performance with respect to the blade modelling parameters, that is, the imposed spanwise profiles of exit flow angle and losses, but also and most importantly their streamwise distributions (as confirmed in the present work), without however investigating the precise mechanisms or identifying patterns that might guide a user in similar applications. That such guidelines are urgently needed if the Euler throughflow model is to establish itself as the basis of engine design systems is clear, though, and the present work aims to fill this gap.

**Damle et al. (1997)**

In an implementation recently put forward by Damle et al. (1997), all three momentum equations are solved also in blade passages and the blade force is relied upon to ensure that at convergence the actual velocity field will match the imposed target for swirl (in design mode) or relative flow angle (in analysis mode). The direction of this blade force is fixed in analysis mode and periodically updated in design mode by integrating a blade geometry from to the current solution, using however the imposed target swirl in place of the calculated tangential velocity. The magnitude of the blade force is given algebraically by the target swirl, which in design mode is direct input and in analysis mode is calculated from the flow tangency condition, through the steady state tangential momentum equation,

$$f_{B, \theta} = \nabla \cdot \left( \nabla (rV^*_\theta) - rF_{F, \theta} \right)$$

(2.4)

Losses would seem to be imposed dynamically from an efficiency target and an imposed shape of the TE entropy profile which are combined with the current solution into an entropy field. The numerical method is the JST scheme (Jameson, 1981), without any convergence acceleration techniques.

The authors successfully apply their method to a supersonic throughflow fan stage. The published results, comparing a design and an equivalent analysis solution, would suggest the absence of captured shocks in the analysis solution. In light of the findings of the present work, this is most surprising and we would explain it by the fact that the flow is axially supersonic throughout, with axial Mach numbers comprised between about 1.5 and 3.3, which avoids two factors associated with shock formation (in Euler throughflow analysis calculations) in transonic compressors, namely choking phenomena and a discontinuity in shock capturing properties at the blade edges. Although, by contrast with the bulk of recent throughflow literature which exclusively deals with analysis applications, Damle et al. make extensive use of the design mode, the declared goal being to devise a design tool free from the Mach number limitations of the classical throughflow methods and endowed with shock capturing capability, the question of shock capturing properties is not addressed, and it is not clear if the authors are aware of possible differences between the design and analysis modes in this regard. It will be shown that for the transonic designs prevalent in today’s industrial practice, the behavior of the two classical modes is indeed fundamentally different, and that this difference must be taken into account if realistic results are to be obtained.

**Liu et al. (2000)**

That the Euler throughflow model is compatible with a wide variety of numerical schemes is demonstrated by Liu et al. (2000) who use Van Leer’s (1981) flux vector splitting in combination with an explicit predictor-corrector scheme proposed by Warming and Beam (1976) and incorporating higher order terms in the corrector step. The model is of the algebraic type and applied to the analysis of known blade geometries only, the \(\theta\)-component of the blade force being calculated from the steady-state tangential momentum equation and the swirl distribution obtained directly from the current solution and the flow tangency condition. It presents a number of original (in the sense of uncommon) implementation details. To improve stability in the presence of shocks, a central-type second order artificial dissipation is added to the upwind scheme and this is formally written as a friction force, retaining only the component parallel to the flow direction. The imposed flow angle adapts to match the incoming flow, but only partly so, taking into account a fraction 0.5 to 0.8 of incidence. Although Liu et al. do not mention this, the ensuing swirl discontinuity explains why they found it necessary to reduce the tangential momentum equation to first order at blade edges. An application is presented
to the analysis of steam turbine last stage (with the perfect gas model) at extreme off-design operation resulting in extensive recirculation zones both inside and outside of blade passages which neither of the classical throughflow methods could represent.

Baralon et al. (1997)

In another recent implementation of an Euler throughflow model, Baralon et al. (1997) employed a time-dependent force. In their approach, the following equation for the tangential component of the blade force is solved in parallel with the main flow equations,

$$\partial_t f_{B,\theta} = \kappa \frac{(v_m + c)^2}{\Delta x^2} \hat{n} \cdot \hat{w}$$

The paper stands out through more complete than usual attempt to model potentially relevant physical phenomena such as spanwise mixing and end-wall skin friction. It is also noteworthy for some resourceful experimentation with modelling techniques in an attempt to cure what are perceived as deficiencies in the context of the intended application of axial transonic compressor flows. For the possible mismatch between the flow angle given as input at the leading edge (in the case of these authors straight from the blade geometry) and that of the incoming flow, a gradual correction to the imposed flow angle, a captured discontinuity, and an analytically resolved discontinuity (as employed by Nigmatullin and Ivanov, 1994, to whom no reference is made, however) are compared and weighed against each other, although without sufficient numerical evidence to allow any conclusion with regard to the unconditional superiority of one of them.

Faced with poorly predicted choking mass flows, the usual tangential blade blockage is replaced by a so-called normal blockage, introduced by Saryari and Bölcs (1995) in the framework of an SLC code for transonic compressor applications. The property of the analysis mode to capture blade-to-blade normal shocks is recognized, and also that this may lead to additional deviation if such a captured shock is located at the blade trailing edge, which is remedied by extending the region of controlled flow angle some way downstream. Neither of these ad-hoc fixes is conceptually appealing, since they imply both non-zero blade blockage and non-zero blade force outside of the physical blade passage. The method heavily relies on the blade geometry, which

It will be shown in the present work that correct choking behavior can be obtained by controlling, in a way perfectly consistent with the classical view of the axisymmetric throughflow as representing the tangential average, in a conjugated way the flow angle and blockage distributions so that they agree with the averaged flow, as from a theoretical viewpoint they should, and selected features of the blade geometry such as throat width.

2.3 Quasi-Steady Rotor-Stator Interface (Mixing Plane)

Wyss et al. (1993) compare three averaging techniques applied to the mid-span section of a transonic fan stage; only 2D computations are performed. The distance between rotor TE and stator LE is about one axial stator chord. The authors mention two possibilities for the quasi-steady calculation of multi-stage turbomachines: Adamczyk’s (1986) average passage model and the averaging-plane approach, used in their computations but qualified as ad-hoc. Three averaging schemes are examined: momentum, energy and area averaging. The steady averaging-plane solutions are compared with a time-averaged unsteady solution using blade pressure distribution, wake profiles, stage adiabatic efficiency and stator loss coefficient. One steady rotor computation is performed. The exit flow averaged according to each of the three schemes is then used as the inlet boundary condition for three steady stator computations. The stator massflow is made to match the rotor massflow through iterative adaptation of the imposed stator exit pressure. To be applicable to the simultaneous, quasi-steady computation of a rotor and a stator, the proposed averaging schemes would need to be extended by feedback from the stator block to the rotor block that would replace the constant exit pressure imposed in the single rotor computation. No indication is therefore provided concerning the numerical properties of the proposed averaging schemes if they were embedded in the global iterative procedure of a simultaneous computation.
The authors’ momentum averaging scheme (MAS) conserves the inviscid fluxes at the interface. Primitive variables, if required, must be decoded from these fluxes by solving a quadratic equation. For 2D adiabatic flow the MAS is equivalent to the mixed out flow. It is insensitive to the axial location where the averaging is performed and accurately represents the available thrust but overestimates losses.

In the energy averaging scheme (EAS) the flux of axial momentum is replaced by that of ideal total enthalpy, \( h_{\text{ideal}} = \frac{h_{\text{ref}}}{\gamma - 1} \). Primitive variables, if required, must be decoded from these fluxes by solving a transcendental equation. Wyss et al. use a Newton method. The EAS is intended to give good agreement with circumferentially averaged experimental data. It gives a good measure of total pressure and total enthalpy and hence adiabatic efficiency and loss.

The entropy averaging scheme (SAS) is mentioned, but results are not presented. It differs from the EAS in that the flux (or, equivalently, mass average) of entropy replaces that of ideal total enthalpy.

The area averaging scheme (AAS) uses area weighted integration of quantities that might be measured directly by a slow-response probe behind a rotor, namely static pressure, total pressure and total temperature. The flow direction is found using the MAS. The AAS does not conserve mass, momentum or energy.

Conclusions? Recommendations?

Sleiman et al. (1996) use a mixing-plane approach in a 3D Navier-Stokes finite element code, citing as sources Dawes (1992) and Giles (1988). They contrast this steady interface model with more expensive approaches, namely unsteady, time-accurate computations, which do not require any modeling, and Adamczyk’s (1990) passage-average model, which is steady but requires complete overlap between the grids of the upstream and downstream blade rows to account for body forces and deterministic stresses.

Circumferentially averaged radial profiles of absolute total temperature, density and radial, axial and absolute tangential velocity obtained from the solution of the upstream blade row are specified at the inlet plane of the downstream blade row. Circumferentially averaged static pressure obtained from the solution of the downstream blade row is specified at the exit plane of the upstream blade row. The averaged values are decoded from the inviscid fluxes (where only the subsonic solution is considered), but the description does not make it clear if any of the fluxes is indeed conserved. Convergence criteria are a residual drop of 2 to 3 orders and a mass flow error at the interface planes of less than 0.1%. The averaged flow states at the mixing-planes are considered part of the solution and updated together with the flow field. The loss-generating property of the mixing-plane approach is acknowledged but not assessed quantitatively.

Results are presented for a low-speed single compressor stage and a high-speed two-stage turboprop compressor.

Schulz et al. (1996) performed steady and unsteady coupled computations of the pump and turbine rotors of a hydrodynamic torque converter. The incompressible Reynolds-averaged Navier-Stokes equations in conjunction with a k-\( \varepsilon \) turbulence model are solved on meshes from 149000 to 375000 points by an implicit, non-time marching finite volume method. At the rotor-stator interface the three velocity components and two turbulence variables (Presumably pitch-averaged in steady computations; the averaging procedure is not mentioned, nor is the treatment of pressure) are transferred between blocks. The authors mention the possibility of scaling the velocity data to conserve mass, but do not say if such a scaling was actually applied in the presented computations. The convergence rate is improved, if the rotor-stator boundary condition is updated at each iteration rather than at larger intervals. Updating at larger intervals was necessary because fine meshes fitted into main memory only one at a time.

Gallus et al. (1995) investigate the 3D flow in a subsonic axial turbine stage. The aim is to study how well a steady-stage calculation procedure predicts the time-averaged flow characteristics. The unsteady computation requires 100 times the computing resources of the steady one. The authors cite previous successful applications of the mixing-plane approach, e.g. by Arts (1985) and Saxter and
Giles (1994). They themselves use a method developed by Copenhaver (1993) that is based on Dring and Spear’s (1991) work on wake mixing. Mixing-planes are established at the trailing edge of the stator and the leading edge of the rotor. Mesh independence of the steady-stage computation was verified. The only significant shortcoming of the steady-stage solution is a slight misprediction of the secondary flow and radial flow angle profiles at stage exit near the hub. It is suggested that the numerical results from the unsteady flow analysis be used to improve the modeling of the steady-stage calculation procedure. Further research is encouraged to achieve this improvement.

Copenhaver et al. (1993) apply a steady multiple blade row approach to a transonic axial compressor stage. Grid independent, steady solutions of the 3D Reynolds-averaged Navier-Stokes equations in conjunction with a $k-\epsilon$ turbulence model at a four-order RMS residual drop are obtained on a mesh of 600000 nodes in five hours of Cray-YMP single processor time (in 1992). To obtain stable and correct solutions for the closely space blade rows (streamwise gap approximately 15% of rotor chord at the hub and 43% at the tip), two interface planes are used, one at the rotor trailing edge and one at the stator leading edge. The rotor and stator meshes overlap in the blade free region. Circumferentially averaged static pressure of the stator flow field is used as the exit boundary condition of the rotor; all other variables are extrapolated. At the stator inlet, circumferentially uniform values of velocity and density are obtained from the rotor flow field by a mixing analysis based on the work of Dring and Spear (1991). We assume that the same holds for $k$ and $\epsilon$ and that static pressure is extrapolated. This allows the massflow of the stage to be controlled in the standard way through total pressure and total temperature at the rotor inlet and static pressure at the stator exit. The mixing-planes are updated at each iteration. The authors do not mention the ambiguity that would arise in their averaging procedure if the flow became axially transonic, but hint at possible instability should a shock cross a mixing-plane.

Conservation depends on the exact treatment of the stator inlet mixing-plane. In addition to the description given in the preceding paragraph, the authors state that at the stator inlet “radial distributions of all the variables […] are obtained from the rotor solution”, without clarifying if all of them are actually used.

Imposing all variables seems to be in contradiction with axially subsonic flow; however, the rotor solution at the rotor trailing edge, where the mixing-plane for the stator inlet is located, does in some way contain the required information on the stator static pressure, since this has been imposed at the rotor exit, located at the stator leading edge. While such an approach would indeed conserve all fluxes (neglecting viscous and turbulence contributions to momentum and energy), it raises some questions regarding the wellposedness of the resulting problem.

If static pressure is retained locally, two cases must be distinguished: Either is it replacing pressure in the circumferentially integrated rotor momentum fluxes, and new primitive variables are decoded afterwards, in which case all fluxes would be conserved, or is the rotor solution averaged independently, and the resulting pressure discarded in favor of that of the stator solution. In the latter case, conservation would be violated for the fluxes of normal momentum and total enthalpy. In any case the approach incurs a mixing loss.

Dring and Spear (1991) compare different averaging techniques for traverse data measured behind the stators of a low-speed two-stage axial-flow compressor. Two operating points are considered: nominal and near-stall. The near-stall operating point is characterized by hub corner separation in both stators. Spanwise profiles for absolute flow angle, relative flow angle and static pressure averaged in different ways are presented. Three averaging techniques are considered: area averaging, mass averaging, and conservative averaging. The latter is equivalent to the MAS of Wyss et al. (1993), with the same idea of obtaining aerodynamically equivalent uniform flow conditions through a 2D mixing process that conserves mass, axial momentum, tangential momentum and energy. The resulting values of tangential velocity ($v$) and enthalpy ($h$) are the respective mass averages ($\bar{v}$ and $\bar{h}$). If the flow is incompressible, as the authors assume, then the resulting axial (in 2D, normal in 3D) velocity ($u$) is the area average ($\bar{u}$).

The three averaging techniques are applied to static pressure directly. The flow angles are either mass averaged or computed from (i) the conservative averaged solution ($\alpha = \tan(\bar{v}/\bar{u})$) or (ii)
the area averaged velocity components \( \alpha = \arctan(\bar{v}/\bar{u}) \). In the region between the hub and 10%-30% span, where the tangential non-uniformity is greatest, differences of up to 13 deg in flow angle, 10% in static pressure and 30% in dynamic pressure are observed.

2D potential flow computations of the second rotor are presented that use as inlet condition the measured stator exit flow averaged according to the different schemes and compared to the experiment through blade pressure distribution and rotor exit flow angle. The wake mixing model (i.e., the conservative averaging scheme) yields by far the most accurate solution.

Eulitz and Engel (1996, 1997) perform steady-state 2D and 3D calculations of a subsonic turbine (stator–rotor–stator) to provide initial solutions for subsequent unsteady calculations. The steady-state calculations use a non-reflecting mixing-plane approach developed by Saxer and Giles (1993, 1994). The meshes at the mixing-plane boundaries are uniform in the tangential direction. The solution is circumferentially decomposed into Fourier modes. The zero-mode is given by the physical boundary condition, while the higher harmonics are obtained from the flowfield. This ensures the non-reflective property by allowing non-uniform tangential distributions of all variables. Characteristic variables are changed until the averaged upstream (index 1) and downstream (index 2) solutions match,

\[
\begin{bmatrix}
\delta \bar{w}_1 \\
\delta \bar{w}_2 \\
\delta \bar{w}_3 \\
\delta \bar{w}_4
\end{bmatrix} =
\begin{bmatrix}
-\bar{c}^2 & 0 & 0 & 1 \\
0 & \bar{p}\bar{c} & 0 & 1 \\
0 & 0 & \bar{p}\bar{c} & 1 \\
0 & -\bar{p}\bar{c} & 0 & 1
\end{bmatrix}
\begin{bmatrix}
\bar{p}_1 - \bar{p}_2 \\
\bar{u}_1 - \bar{u}_2 \\
\bar{v}_1 - \bar{v}_2 \\
\bar{\rho}_1 - \bar{\rho}_2
\end{bmatrix}
\] (2.6)

The direction of propagation of each characteristic is taken into account by adapting the sign of each row of the matrix of eigenvectors according to the flow state. It is not said if this is done automatically. Equation (2.6) is written for subsonic flow (normal to the interface) from side 1 to side 2. The averaged variables are decoded from the fluxes so that the interface is conservative but produces a mixing loss.

Saxer and Giles (1993b, 1994) solve the 3D Euler equations by a multigrid Lax-Wendroff algorithm for a transonic axial turbine stage, applying the non-reflecting boundary conditions developed by Giles (1990) at inlet, exit and the rotor-stator interface. In this technique, the solution is decomposed into Fourier modes in the circumferential direction. The zero-mode is used to impose the desired physical boundary conditions. The primitive coefficients (those corresponding to the circumferential variations of the primitive variables) of the higher modes are transformed to the respective characteristic coefficients. For each mode independently, the four (in 2D) or 5 (in 3D) characteristic coefficients must satisfy certain linear relations for the boundary condition to be non-reflective. To ensure the wellposedness of the pseudo-time evolution process, the resulting values for the incoming characteristics in function of the outgoing ones are not set directly, but underrelaxed. An empirical value for the underrelaxation factor is provided.

In 2D, the method yields exact (spatially) non-reflecting boundary conditions within the linearized framework on which the theory is based. Non-linearity (e.g., a shock crossing the boundary) introduces slight errors. In 3D, it is applied independently at each radial position, thus neglecting the influence of radial gradients, and is therefore referred to as quasi-three-dimensional.

At a steady rotor-stator interface, the procedure applied to the higher Fourier modes is the same as at an inlet or exit. The level, corresponding to the zero-mode, is adapted such that the jumps across the interface, equation (2.6), vanish in the circumferentially averaged solution. The averaging scheme chosen by the authors to compute these jumps, called stream-thrust flux-averaging, conserves mass, momentum and energy. It is the same scheme as the MAS of Wyss (1993), also used by Dring (1991), Copenhaver (1993) and (1996). In the considered turbine applications, the flow is subsonic normal to all boundaries, including the rotor-stator interface; the ambiguity that would arise in the decoding for normally transonic flows is not mentioned.
Merz et al. (1995) apply the non-reflecting boundary condition technique of Giles to the steady, coupled calculation of a turbine stage. The Reynolds-averaged Navier-Stokes equations in conjunction with an algebraic turbulence model are solved by a Jameson-type finite volume time-marching method. Total mesh size is 357000 points. The rotor-stator interface is identical to that used by Saxer and Giles, except that grid spacing is allowed to be non-uniform in the tangential direction. The azimuthal distributions on a uniformly spaced mesh required for the Fourier transform are obtained through a cubic interpolation-spline, keeping the same number of points. “Improvement in stability” is reported with the non-reflecting interface, compared with an algorithm based on mere circumferential averaging.

Arts (1985), in one of the early steady rotor-stator computations (referred to by the author as time averaged blade row interaction), applies a time-marching finite volume method for the solution of the 3D Euler equations to a transonic axial turbine stage. Total mesh size is only 11858 points (computed in 1983). At the upstream (stator exit) and downstream side (rotor inlet) of the rotor-stator interface, the same boundary conditions are applied as at, respectively, the inlet and exit. The imposed variables (absolute total pressure and temperature and two flow angles at the rotor inlet and static pressure at the stator exit) are calculated from the pitch-averaged conservative variables of the connected block. Arts mentions that these values “are repeatedly updated”, but does not give the exact update frequency.

Denton (1992) presents a conservative rotor-stator interface and applies it to a variety of axial compressor and turbine test cases. Both thin shear layer Navier-Stokes and Euler computations are performed with an explicit, cell-vertex finite volume method allowing two or three levels of multigrid. In the Euler computations, losses are included through a friction force. The periodic H-meshes used for all computations are rather coarse, typical sizes being 18050 points per row for viscous and 6760 points for inviscid computations.

The fluxes at the interface, called the mixing plane, are first calculated in the standard way from the local solution. A unique circumferential average is then determined; it is not said how the upstream and downstream sides enter into this average. Lastly, the original local fluxes at each side of the interface are replaced by those of the respective first inner layer, scaled by the ratio of the average flux at the interface to the average flux of the first inner layer. This procedure is termed extrapolation of the flux variations. Loss generation is addressed by stating that with inviscid flows the magnitude of entropy creation was found to be negligible even when using an older method that enforced circumferentially uniform flow at the mixing plane.

Test cases include a generic two- and four-stage axial turbine, a two-stage low-speed experimental turbine, a three-stage high-pressure-ratio transonic compressor, a four-stage, nine-blade row (IGV + four stages) industrial compressor and a large two-stage, six-blade row axial flow compressor with high subsonic blade surface Mach numbers. Instabilities at the mixing plane are reported for the three-stage high-pressure-ratio transonic compressor.

Dawes (1992) solves the 3D Reynolds-averaged Navier-Stokes equations with the Baldwin-Lomax or a one-equation turbulence model by a time-marching, cell-centered finite volume method on periodic H-meshes. The code allows axisymmetric throughflow computations to be performed by using only one cell in the tangential direction. Multiple blade rows can be coupled in a single computation, each row being modelled either as a 3D or throughflow row. The emphasis of the article is on the benefit of cheaply providing a machine environment for one 3D row by including the upstream and downstream rows in axisymmetric mode, rather than imposing artificial boundary conditions. The examples presented, an axial transonic compressor stage (using a total of 70000 grid points if both rotor and stator are modelled in 3D) and a low hub-tip ratio nozzle guide vane with and without the downstream rotor (50000 grid points for both blade rows), demonstrate particular sensitivity to the downstream radial pressure distribution.

For the inter-row mixing planes, which the author states must as a minimum ensure conserva-
tion of the machine massflow, his code has three options:

(i) Area averaging of \((\rho, \rho \mathbf{W}, p)\).

(ii) Passing mass averaged total pressure and temperature and two flow angles downstream and area averaged static pressure upstream.

(iii) Passing area-averaged Riemann invariants upstream or downstream as required by the three wave speeds.

No further details are given; the dearth of information is especially acute for (i), which is used for both test cases. Without additional measures, (ii) and (iii) do not yield mass conservation. It would also be valuable to know, if non-uniform circumferential distributions are allowed for some variables on one or both sides of the interface.

Arnone (1994) uses a Jameson-type cell-centered finite volume method on periodic and non-periodic C- and H-meshes. The convergence of the explicit Runge-Kutta scheme is accelerated through the usual techniques (multigrid, implicit residual smoothing, local time stepping). In a steady multi-row analysis communication between blade rows is through mixing planes. The total temperature, total pressure and two flow angles necessary as inlet condition for the downstream blade row are computed by pitch-averaging those quantities on the upstream blade row at the respective radius. Likewise, the radial pressure distribution necessary as exit condition for the upstream blade row comes from averaging on the downstream blade row. The type of averaging is not indicated, nor is the use of under-relaxation or any other stabilization technique mentioned. The extrapolated variables are probably the same as for a true inlet or exit boundary; the outgoing Riemann invariant at an axially subsonic inlet, density, the velocity vector and the circumferential distribution of pressure at an axially subsonic exit.

The rotor-stator boundary conditions are updated at each Runge-Kutta stage to provide as much of a link as possible between blocks. The interface is not conservative and probably also generates a mixing loss. Concerning conservation in Arnone’s code, the artificial dissipative fluxes are set to zero on solid walls, but non-matching periodic boundaries use simple linear interpolation. In view of the results of Dring and Spear (1991) and also from physical reasoning, averaging of the flow angles seems questionable.

The steady multi-row technique is applied to a two-stage transonic turbine. H-blocks with about 100000 points per blade row are used. The axial distance between the blade rows was chosen to provide adequate mesh quality and limit flow distortion by the mixing planes. Compared with the true geometry, it is increased by a factor of about four. The residual drops 4.8 orders in 300 multigrid cycles. The solution is analyzed qualitatively in terms of secondary flow patterns, which appear to be correctly captured.

Nigmatullin and Ivanov (1994) solve the 3D Euler equations by an implicit higher-order extension of Godunov’s method. After constructing left and right initial states through a second order upwind scheme, the Riemann problem at each cell face is solved exactly to obtain the flow state from which the time averaged flux is then calculated. The authors repeatedly stress the advantage of an exact Riemann solver for two problems peculiar to turbomachinery applications, namely the boundary between ducts and blade passages in throughflow computations and the rotor-stator interfaces in simultaneous, steady 3D computations of multiple blade rows.

In throughflow computations in analysis mode the conservative form of the Euler equations is different in ducts and blade passages, with the consequence that waves propagate at different speeds. At the cell faces representing leading and trailing edge, a modified Riemann problem is therefore solved, respecting the sense of wave propagation on each side. It may yield different cell face flow states (and hence different fluxes) for the duct side and the blade passage side of the cell face, linked through the conservation laws. As it is presented, the algorithm appears complicated, involving an iterative solution with several case distinctions. Apart from providing a “numerically clean” link between bladed and non-bladed domains, however, it offers the additional advantage of allowing discontinuous blockage.

At steady rotor-stator interfaces the two blocks communicate through an equivalent axisymmetric flow (EAF) that conserves mass, momentum and energy. The two tangential velocity compo-
ponents are passive quantities for which, depending on the local flow direction, either local values are retained or those of the EAF taken. On each side of the interface Riemann problems are solved for each cell face. In these, the left and right initial flow states are the local flow state and the EAF, in accordance with the assumption that each blade row sees only the circumferential average of the connected row. The Newton method is used to decode the EAF from the fluxes; no mention is made of the difficulty of selecting the sub- or supersonic solution. It is not clear from the description how the iterative process of calculating the EAF and solving the Riemann problems is arranged or how it is nested with the global iterations. It is claimed that the boundary condition is non-reflecting for the linearized problem of slightly perturbed axisymmetric flow.

Blade-to-blade mesh surfaces are surfaces of revolution to keep integration in the circumferential direction simple, but the rotor-stator interfaces may be spanwise non-matching. A combined mesh is then formed on each side by including the spanwise boundary node positions of the respective other side, which yields a spanwise matching connection. Each original boundary cell face may thereby be split spanwise in two or more sub-cell faces. Accordingly, each original flux is the sum of the corresponding sub-cell face fluxes.

Example for the effect of blade row interaction
Dorney and Sharma (1997) compare four coupling techniques: unsteady fully coupled, steady mixing plane, steady single blade row, and loosely coupled. The latter consists of a sequence of steady single blade row solutions with different positions of the incoming wake (inlet) or potential pressure disturbance (outlet). The mixing plane transfers mixed-out conserved fluxes. Using a thin shear layer code with the Baldwin-Lomax turbulence model, they calculate a 2D section of a transonic compressor stage composed of IGV (two passages modelled) and rotor ($M_{rel} = 0.97$, one passage modelled) on a mesh of about 45,000 points. The unsteady model costs about six times as much as any of the three steady models (which are roughly comparable in cost). The prediction of time-averaged blade pressure distribution and unsteady pressure envelope by the loosely coupled model is good in the IGV (which sees no wakes) but poor in the rotor (which sees the IGV wakes). The upstream branch of the detached rotor bow shock is responsible for 71.5% of the losses in the IGV. The loosely coupled model captures 23.3% of these shock losses, the mixing plane and single blade row models none. Errors on work input and total temperature ratio are about 5% and 0.5%, respectively, for the mixing plane and single blade row models, and about 1.5% and 0.15% for the isolated blade rows (which take their boundary conditions from a presumably tuned 1D calculation). Efficiency is overpredicted by 1 percentage point by the former two, and by 2 points by the latter. Errors on exit absolute Mach number are −0.9%, −6.0%, and −6.6% for the mixing plane, single blade row, and loosely coupled models. On exit absolute flow angle (46.8 deg), errors are 2.8 deg, 0.6 deg and 1.1 deg.
Chapter 3  Numerical Method

3.1 The EURANUS/Turbo Code

3.1.1 Overview

The EURANUS/Turbo code (Hirsch et al., 1991a, 1991b) solves the integral form of the 3D Reynolds-averaged Navier-Stokes equations in cartesian coordinates for multiblock configurations. The equations are discretised in space on structured meshes following the finite volume method. The central approximation to the convective fluxes is complemented by any one of a number of artificial or upwind dissipation formulations. In the basic version, this is the classical JST blend of scalar second and fourth order artificial dissipation. Explicit time integration by a Runge-Kutta method is combined with implicit smoothing of the residuals and embedded in a FAS multigrid cycle. In the turbomachinery version, individual blade rows may be modelled either in 3D or axisymmetrically with an integrated Euler throughflow model (Sturmayr and Hirsch, 1999a, 1999b). The various numerical techniques are outlined in the following.

3.1.2 Basic Numerical Method

The EURANUS code (Hirsch et al., 1991a and b; Rizzi et al., 1993) solves the integral form of the three-dimensional Reynolds averaged Navier-Stokes equations in a cartesian coordinate system. The EURANUS/Turbo version extends this code to rotating systems, solving

\[
\int_{V} \frac{\partial}{\partial t} U dV + \oint_{S} \hat{\mathbf{F}} \cdot d\hat{\mathbf{S}} = \int_{V} \mathbf{Q} dV
\]

with

\[
\mathbf{U} = \begin{bmatrix} \rho \\ \rho \mathbf{W} \\ \rho \mathbf{E} \end{bmatrix}, \quad \hat{\mathbf{F}} = \hat{\mathbf{F}}_i - \hat{\mathbf{F}}_v = \begin{bmatrix} \rho \mathbf{W} \\ \rho \mathbf{W} \mathbf{W} + \rho \mathbf{I} \\ \rho \mathbf{W} H + \mathbf{I} \end{bmatrix} - \begin{bmatrix} 0 \\ \tau \tau \tau \\ \tau \tau \tau \end{bmatrix}
\]

where dV denotes a volume element, d\hat{\mathbf{S}} a surface element, \rho density, \mathbf{W} = (W_x, W_y, W_z)^T the relative velocity vector, E total internal energy per unit mass, p pressure, \mathbf{I} the identity matrix, \tau the shear stress tensor, H total enthalpy per unit mass, and k thermal diffusivity. The centrifugal and Coriolis accelerations due to system rotation are represented by source terms,

\[
\mathbf{Q} = \begin{bmatrix} 0 \\ -\rho [2 \hat{\Omega}_{sys} \times \mathbf{W} + \hat{\Omega}_{sys} \times (\hat{\Omega}_{sys} \times \mathbf{W})] \\ -\rho (\hat{\Omega}_{sys} \times \hat{\mathbf{U}}) \cdot \mathbf{W} \end{bmatrix}
\]

where \hat{\Omega}_{sys} denotes the angular velocity vector of the system. For computational efficiency, rotation is restricted to be about the z-axis. Equation (3.12) is discretized in space according to the finite volume approach, using structured meshes with hexahedral cells,

\[
\Delta V \frac{\partial}{\partial t} \mathbf{U} + \sum_{n=1}^{6} \hat{\mathbf{F}}_n \cdot \hat{\mathbf{S}}_n = \Delta V \mathbf{Q}
\]

with \Delta V and \hat{\mathbf{S}} denoting respectively the cell volume and the cell face normal vector. The viscous flux \hat{\mathbf{F}}_v is evaluated directly on the cell faces, using Gauss’s theorem to compute the gradients. For the inviscid flux \hat{\mathbf{F}}_i, both a central scheme with artificial dissipation (Jameson et al., 1981; Jameson,
Numerical Method

1982) and upwind schemes (Hirsch, 1990) are available. The present computations use the central scheme with added fourth order artificial dissipation, for which the numerical flux on $\xi$-cell faces is

$$
(F_{i+1/2} \cdot S_{i+1/2})^* = \frac{1}{2} (F_i + F_{i+1} + \frac{2}{\delta} S_{i+1/2} - d_{i+1/2})
$$

(3.5)

with the artificial dissipative flux

$$
d_{i+1/2} = \varepsilon_i^{(4)} \delta_0^2 U_i^{AD} - \varepsilon_i^{(4)} \delta_0^2 U_{i+1}^{AD}
$$

(3.6)

and with analogous expressions for the $\eta$- and $\zeta$-cell faces, with $j$ and $k$ instead of $i$. In Equation (3.6), $U_i^{AD}$ is the solution to Equation XX with $\rho H$ replaced by $\rho E$ and $\delta_0$ being a second difference operator. Robustness on high aspect ratio meshes is increased by multidimensional eigenvalue scaling of the dissipation factor (Martinelli, 1987), for instance in the $\xi$-direction,

$$
\varepsilon_i^{(4)} = s \lambda_\xi^{(4)}
$$

(3.7)

with $s = \max \left[ 1, \left( \lambda_\eta^{(4)} / \lambda_\xi^{(4)} \right)^{\sigma}, \left( \lambda_\zeta^{(4)} / \lambda_\xi^{(4)} \right)^{\sigma} \right]$ and $\sigma$ a constant in $[0, 1]$. $\lambda_\eta$, $\lambda_\xi$, $\lambda_\zeta$ are the spectral radii in the cell face normal directions, scaled by the cell face area, for instance in the $\xi$-direction,

$$
\lambda_\xi = \left| \frac{\partial p}{\partial \eta} \right| + \varepsilon_i \left| \frac{\partial p}{\partial \zeta} \right|
$$

(3.8)

where $c$ denotes the speed of sound. Time discretization is accomplished by an explicit k-stage Runge-Kutta scheme (RK) in conjunction with implicit residual smoothing and multigrid (MG; Jameson et al., 1981; Jameson and Baker, 1983). The Runge-Kutta scheme is embedded in a FAS-multigrid cycle (Brandt, 1977; Hackbusch, 1985), offering the choice between V, W, or sawtooth strategies. Residual smoothing uses variable coefficients according to Radespiel et al. (1990). Smoothing of the coarse grid corrections is performed before prolongation to the next finer grid level. A good initial solution is established through full multigrid, in which the inclusion of the next finer grid level depends upon the reduction of the residual by a given amount, typically 2 to 3 orders of magnitude.

The loops resulting from the above techniques and the multiblock capability are nested as follows:

- Loop over the grid levels
  - Loop over the smoothing sweeps
    - Loop over the Runge-Kutta stages
  - Loop over the domains

The inlet boundary condition imposes the absolute total pressure, the direction of the absolute velocity vector, and the absolute total temperature. The exit boundary condition imposes the static pressure at some reference radius and determines the radial pressure distribution form radial equilibrium, using the trapezoidal rule for integration. On rotating Navier-Stokes walls, too, the pressure gradient in the wall normal direction obeys radial equilibrium. Since the mesh is generally not orthogonal to the wall, two corrections are required compared with zero pressure gradient in the wall normal index direction, here taken to be $\eta$, $\xi$ and $\zeta$ being the two tangential directions,

$$
\frac{\partial p}{\partial \eta} = -\frac{1}{S_\eta^2} \left[ \frac{\partial p}{\partial \xi} \frac{S_\eta}{S_\xi} \frac{\partial p}{\partial \zeta} + \frac{\partial p}{\partial \zeta} \frac{S_\eta}{S_\zeta} \frac{\partial p}{\partial \xi} \right] - \rho \Delta V \left( \frac{\partial \eta}{\partial \xi} \right) + \frac{\partial p}{\partial \xi} \frac{\partial \eta}{\partial \xi}
$$

(3.9)

**Remark on the energy equation in a rotating system**

In differential form in a steadily rotating system
The viscous fluxes are system invariant (independent of system rotation), and independent of the velocities in which they are expressed, since the velocity field describing the difference between an inertial and a steadily rotating system consists in a solid body rotation. A rotating system is in fact just a special case of arbitrary motion of the control volume, as described by an ALE formulation. Introducing the rothalpy, the energy equation becomes

\[ \partial_t (\rho E) + \nabla \cdot (\rho \mathbf{W} \mathbf{H}) - \nabla \cdot (\tau \cdot \mathbf{W} + k \mathbf{V} T) = -\rho \Omega \times (\Omega \times \mathbf{W}) \]  

(3.10)

The steadily rotating system being imposed by the nature of the problem and the desire for the simplest possible description of the flow, two questions remain where there is a true choice. The first one concerns the velocity components for which to solve (including the kinetic part of the total energy, although both could in principle be treated separately).

### 3.2 Turbomachinery-Specific Implementation Aspects

Although turbomachines are naturally described in cylindrical coordinates and the early calculation methods, including the first Euler solvers, did indeed formulate and solve the governing equations in cylindrical coordinates, current practice favors cartesian coordinates. Several reasons may be advanced for this choice, the chief three being the inherent simplicity, uniformity within a code which may be, as in the present case, a huge software project with applications to a large variety of flows (software engineering and maintenance costs), and finally the fact that turbomachinery flow simulations increasingly include elements outside the main flow path (e.g., bleed air, disk leakage, or cooling flows) which may not be cylindrical in shape.

The choice of coordinates has an impact on the numerical dissipation, both inherent and artificial, and for certain types of flow working with cylindrical velocity components might enjoy an advantage, an example being axisymmetric swirling flow in ducts, which appears uniform in cylindrical coordinates and non-uniform in cartesian coordinates. Numerical tests confirm that this is a measurable effect, leading to increased losses if, for example, swirl-free flow in an inlet duct is calculated in the relative system.

### 3.3 Software Engineering and Post-Processing

The numerical solution of the governing equations for one particular case, defined by the geometry, the mesh used to represent it, and the boundary conditions, is only one part of the use of CFD in industry. Equally important in the processing of the raw solution data and its presentation to the user, to give new insights into physical mechanisms in the flow (which may be inaccessible with experimental methods), show the physically relevant variables, allow convenient comparison of alternative configurations and thus provide a basis for design decisions. For turbomachinery applications, the azimuthal average, represented on a meridional projection of the machine, has proven particularly valuable in this regard. EURANUS has therefore been equipped with an option to extract the pitch-averaged solution from an arbitrary multiblock mesh. The algorithm is outlined in the following.

A 2D meridional mesh is required to support the pitch-averaged solution. Ideally, the spatial resolution of this mesh should be comparable to the 3D mesh of the computation. On too coarse a mesh, details would be lost, while too fine a mesh would be overkill and suggest a false sense of accuracy not sustained by the underlying calculation. It is therefore composed of any number of hub-to-tip mesh surfaces, or parts thereof, extracted from the 3D mesh. These may be boundary surfaces as well as inner mesh surfaces. It is not uncommon, for example, to choose a zig-zagging path through a H-O-H mesh configuration, ensuring even streamwise grid density, with some refinement in the areas of leading and trailing edges. For convenience of handling in the interactive visualization software (CFView), the individual patches are fused into one surface, complemented by one additional surface representing the contours of the blade, the visualization of which is of vital importance in assessing the calculated flow solution.
It was found that for realistic meshes, a naive, straightforward search strategy leads to unacceptable computing times. This problem is not going to be relieved by the advent of faster computers, since meshes grow to immediately exploit new resources, as the past several years have so impressively demonstrated. An efficient search strategy has therefore been devised which proved to perform very satisfactorily in day-to-day use, although it cannot lay any claims to optimality.

3.4 Non-matching Connecting Boundaries

Figure 4.3 illustrates the idea of avoiding skew meshes by allowing the mesh lines crossing a periodic boundary to be discontinuous. It compares the same detail of the leading edge region for a matching grid (Figure 4.3a) and for a non-matching grid (Figure 4.3b). Although the matching grid can be made orthogonal upstream and downstream of the blade passage, the critical regions with strong flow gradients remain covered by extremely skew cells. The non-matching grid, on the other hand, has quasi-orthogonal cells throughout, as seen from the global view in Figure 4.3c.

To facilitate understanding of the difficulties associated with non-matching boundaries (NMB), especially in 3D, we shall first outline the concept of connecting boundaries within the cell centered framework of EURANUS (Eliasson, 1993):

Each block is surrounded by two layers of dummy cells. Any type of boundary condition is set by assigning suitable values to these dummy cells. In the case of a matching connecting boundary, both the geometry and the values assigned to these dummy cells are identical to those of the first and second inner layer of the connected block. The rotational and translational periodicity conditions required in turbomachinery applications are simply special cases of a connecting boundary.

While mere copying operations suffice to handle matching boundaries, NMB require some form of interpolation. The issues of conservation, accuracy, and computational effort therefore require careful consideration. No single optimum strategy exists. Of the two philosophies that can broadly be distinguished, namely (i) the Chimera or overlapped grid approach (Benek et al., 1985) and (ii) the patched grid approach, the latter was adopted for EURANUS because of its compatibility with the described framework. It assumes the boundary grid points of the connected blocks to lie on the same continuous curve in 2D or surface in 3D. Out of conservation considerations, the original development was centered around the decomposition of the connecting interface into patches (the shaded area in Figure 3.1a).

Figure 3.1b schematically illustrates the three steps performed at each iteration. First, the dummy cell variables are set by bilinear interpolation in the two tangential directions (step 1). Flux computation can then proceed independently for each block, not distinguishing between boundary cell faces and inner cell faces (step 2). This step is identical for matching and non-matching connections. In the latter case, however, conservation is not guaranteed, since in general

\[ \int_{\text{interface } A} \mathbf{F}_A \cdot d\mathbf{S} \neq \int_{\text{interface } B} \mathbf{F}_B \cdot d\mathbf{S} \quad (3.12) \]

Local and hence global conservation can be achieved by mixing the residual contributions of the cell faces sharing patch P and reassigning the result to both cell A and cell B, accounting for the fractions of the boundary cell face areas covered by patch P (step 3). Simple considerations in 2D show that the applicability of the conservative treatment is limited to planar interfaces.

In 3D, two additional difficulties arise. First, due to the quadrilateral shape of the patches, the described technique is limited to configurations with no crossover between grid lines belonging to the same family. For non-trivial configurations, this condition was found hard to satisfy during grid generation. Second, on curved Navier-Stokes walls a serious mismatch between the two discrete boundary representations occurs (Figure 3.1c). Obviously, interpolation in physical space would introduce severe errors in the boundary layer velocity profiles, if the flow is assumed normal to the paper plane. Both problems were solved by mapping the 3D interface into 2D parametric space. Simultaneous transformation of two discretized surfaces is by no means trivial, though. The technique employed is described in Section XX [Boundary Conditions: Non-Matching Connections]. While the resulting scheme is inherently non-conservative, the errors were found to be negligible in
the present applications.

Since the interpolation information (stencils and coefficients) is determined once during startup, there is practically no computational overhead associated with the use of NMB. Figure 3.2 shows the test case of a 3D bump channel with two different grids on both sides of a curved interface (Figure 3.2a). The good continuity of the pressure contours across the interface on the back wall and the symmetry plane can be appreciated from Figures 3.2b and 3.2c.

Figure 3.1 Non-matching connecting boundaries, (a) Patch shared by two boundary cell faces, (b) schematic representation of the non-matching interface treatment, (c) grid mismatch on a curved Navier-Stokes wall.

Figure 3.2 Inviscid flow at Mach 0.4 through a bump channel, (a) geometry with surface meshes on the inlet section, the exit section, the curved non-matching interface, and the lower wall, (b) pressure contours on the back wall, (c) pressure contours on the symmetry plane.
Chapter 4  Mesh Dependence Analysis

4.1 Introduction

Helical pump inducers found in liquid propellant rocket motors represent a particularly challenging problem for CFD simulations, due to the extremely high blade angle and the importance of viscous effects. A typical example is a non-cavitating LH2 pump inducer. Preliminary studies in two dimensions for the tip section of such an inducer have permitted the identification of some of the key factors for the efficient computation of reliable solutions. It is demonstrated that with adequate meshes representative solutions of the two-dimensional turbulent flow can be obtained with as few as 1500 to 5000 cells. The very low degree of grid skewness required is made possible by a non-matching patched grid technique applied to the periodic boundary. Extrapolation of the findings to the three-dimensional inducer configuration suggests that meshes of the order of 100,000 to 400,000 cells should offer an acceptable level of numerical dissipation and mesh independence to within engineering accuracy. At this level of resolution, mesh influence is investigated by comparing computations on five different grids. The effect of turbulence is included through the Baldwin-Lomax model. Because of the availability of experimental results, the modelled flow is that of air at low Mach number..

4.2 Motivation

The fuel supply systems of liquid propellant rocket engines rely on centrifugal pumps to accomplish the steep pressure rise from a few bar in the reservoirs to the order of one hundred bar in the combustion chamber, while sustaining the required high massflows. To minimize weight, these pumps operate at high suction specific speeds, of the order of 60,000 to 70,000. The problem of cavitation is controlled by preceding the centrifugal main stage with an axial inducer providing sufficient initial head rise and swirl to avoid further cavitation (Ridwan et al., 1994). From a CFD point of view, inducers are challenging test cases because of

• the extremely high stagger angle;
• the swept back leading edge;
• the features of the flow in the long, narrow blade passage, namely important viscous contributions to the work transfer as well as strong secondary flows and radial motion.

Few 3D Navier-Stokes computations have been reported to date. One well documented case is the analysis of the Ariane 5 LH2 pump inducer by Le Fur (1989) and by Moore et al. (1990) with an incompressible elliptic flow solver. Figure 4.4a shows a 3D view of the inducer considered in the present study. Four blades with an average stagger angle of 80 deg are wrapped around by an average of 240 deg. The shroud is at constant radius, while the hub-to-tip ratio gradually increases from inlet to exit. Tip clearance is ignored, although the shroud is stationary, while the entire hub is rotating with the blades. Experimental data for air flow are available at 10,000 rpm. Our computations model air as an ideal gas with gas constant $R = 287 \text{ J/kg/K}$, isentropic exponent $\gamma = 1.4$, Prandtl number $Pr = 0.72$, and constant molecular dynamic viscosity $\mu = 1.874 \times 10^{-5} \text{ Pa s}$. In this context, it is interesting to remark that, within the present range of relative Mach number (roughly 0.1 to 0.4) and for similar flow conditions, the ideal gas law provides a closer approximation to the actual behavior of LH2 than the assumption of incompressible flow (Gerolymos and Geai, 1994). The Reynolds number, based on the shroud diameter and blade tip velocity, is $1.13 \times 10^6$. The walls are assumed to be adiabatic and fully turbulent.

Applying structured H-meshes with matching periodic boundaries, the large stagger angle provokes strongly sheared cells on blade-to-blade mesh surfaces. The pronounced leading edge sweep—the leading edge joins the blade tip almost tangentially—has the same effect on meridional mesh surfaces. It is known from recent analyses (Turner et al., 1993; Hirsch, 1994a) that highly sheared cells have a most detrimental effect on accuracy. Earlier Euler and Navier-Stokes calculations on the described inducer geometry with 3D periodic H-meshes did indeed show considerable mesh dependence, leading to CFD results of very poor reliability. Small changes in the grid point distribution and
Modifications of the mesh size led to large changes in the calculated massflow and pressure ratio. Another consequence of the poor mesh quality were high losses because of excessive numerical dissipation.

In order to sort out the different factors influencing the calculated results and in an attempt to separate numerical effects from other modelling parameters, a systematic study was initiated. This study covered first 2D blade-to-blade computations on a representative section of the described inducer, comparing different meshes for compressible and incompressible Euler as well as compressible turbulent Navier-Stokes calculations. This analysis provided clear guidelines with regard to mesh properties for the obtaining of consistent, mesh independent results, which were then applied to the 3D computations. The ability to treat non-matching connecting boundaries was found to be an essential element.

Section 4.3 therefore introduces this concept and explains the techniques applied in the present calculations. Section 4.3 summarizes the main results of the preliminary studies in 2D. It also addresses numerical prerotation. The final 3D calculations are discussed in Section 4.4, including the generation of structured meshes for this type of geometry, which turns out to be a non-trivial problem.

### 4.3 Preliminary Studies on Mesh Influence

#### 4.3.1 Prerotation

In earlier rotor computations, a certain amount of numerical prerotation has consistently been observed, i.e., conservation of angular momentum was not satisfied upstream of the blade row. To eliminate any influence from badly conditioned meshes, the phenomenon has been analysed on a simplified geometry consisting of the annular passage between two cylinders with four radial fins acting as blades, with the aim of identifying the relevant parameters and assessing its harmfulness with regard to global accuracy.

Figure 4.13 illustrates the effect of progressive refinement around the leading edge. The sequence of four plots shows the axial evolution of the azimuthal average of $V_\theta$ at mid-height between the two cylinders. The leading edge is situated between the two points marked by horizontal dashes. The flow is inviscid with $M_{rel., tip} = 0.03$ and $b_{tip} \approx 45$ deg, $M$ and $\beta$ denoting respectively Mach number and relative flow angle. Inflow is purely axial.

The number on the meshes in the right margin indicates the number of uniform cells between inlet and leading edge, respectively 8, 16, and 32 for cases a, b, and c. In case d, the first cell upstream of the leading edge and the following half cell were refined by a factor of twenty. It is seen that when going to finer meshes, prerotation gradually disappears, while the exit value of 3.2 m/s and also the evolution inside the larger part of the blade passage remain unchanged. Hence, a very fine mesh in the axial direction around and upstream of the leading edge is required to suppress prerotation.

Navier-Stokes simulations indicate that, at high Reynolds numbers typical of the present pump inducer, viscosity does not appear to influence the numerical upstream prerotation. Slight compressibility ($M_{rel., tip} = 0.6$), on the other hand, reduces the extension of the upstream influence, at least for this test case.

#### 4.3.2 Two-dimensional blade-to-blade analysis

A systematic study in 2D, documented in a series of unpublished reports (Sturmayr and Hirsch, 1993; Sturmayr and Hirsch, 1994a, b, and c), has preceded the 3D computations. Compressible and incompressible Euler as well as compressible, turbulent Navier-Stokes calculations have been performed on the flattened tip section of the inducer on meshes from 356 to 25088 cells. The tip section was chosen because of the most severe conditions with respect to stagger angle induced mesh skewness. To compute incompressible flow, a preconditioned version of the governing equations is solved (Hirsch and Hakimi, 1995). The investigation’s objective was to answer the following questions:

- What criteria does the mesh have to satisfy for maximum accuracy and minimum mesh sensitivity?
Can the inducer flow be accurately predicted on reasonably sized meshes allowing extension to 3D while respecting current limits on computing power?

Inviscid calculations should provide a rough idea of numerical loss levels. In a first step, Euler calculations were performed on two families of meshes, skewed periodic H-meshes such as Figure 4.1a, and quasi-orthogonal non-matching meshes. Two different non-matching topologies were compared:

- A single block H-mesh with elliptical arc extensions requiring non-conservative NMB (shown in Figure 4.1c).
- A 5-block mesh with straight line extensions, optimal point distribution, minimum skewness, and planar connections enabling conservative NMB (Figure 4.2a).

Contour plots reveal that in both cases, the non-matching periodic interface is completely transparent to the flow. As an example, pressure contours on a 2816-cell mesh with elliptical arc extensions are shown in Figure 4.2b. Comparing with the skew, matching meshes of Figure 4.1a, the blade pressure distribution is more accurately captured on the coarsest quasi-orthogonal meshes of 356 and 704 cells than on the finest tested skew meshes of 1728 and 6912 cells.

A detailed investigation of the loss generation mechanisms in the Euler computations produced the following conclusions:

- The global loss level is significantly lower on quasi-orthogonal than on sheared meshes, and on fine than on coarse meshes.
- For practically all incidence angles, except those very close to zero, the major source of loss generation is the sharp leading edge.

Confidence in the CFD predictions for the nearly orthogonal non-periodic meshes is provided by the fact that for the same flow conditions, the blade pressure distributions obtained with compressible Euler, incompressible Euler, and compressible turbulent Navier-Stokes computations are in very good agreement, on coarse as well as on fine meshes. Next to accuracy, a second important requirement for the use of CFD in an industrial environment is computational efficiency. In this regard, too, the quasi-orthogonal meshes enabled by the NMB technique provide an advantage. All 2D inducer calculations (both Euler and Navier-Stokes, compressible and incompressible), including the finest tested meshes of 11264 cells (elliptic extensions, in a single block) and 25088 cells (straight extensions, in five blocks), converge to single precision machine accuracy in less than 400 MG cycles. In excess of 1000 cycles are required on the skew meshes. The results on both families of non-periodic meshes being equivalent in any respect, the single block topology was elected for the 3D calculations for generality, ease of handling, and facilitated post processing.

An important parameter in turbulent Navier-Stokes calculations is the refinement of the mesh towards the walls. Knowing that close to 90% of the friction losses are generated in the inner region of a turbulent boundary layer, the mesh must be fine enough to allow the inner turbulent viscosity to be appropriately captured (Hirsch, 1994). Since the algebraic Baldwin-Lomax model (Baldwin and Lomax, 1978) performs as well a task in reproducing the logarithmic velocity profile as do two equation transport models, and since no major separated regions are to be expected, the former was preferred here for its robustness and efficiency. A $y^+$ value of 1 to 10 for the first inner grid point usually ensures adequate resolution of the inner region, also if calculations are to be performed on coarser grid levels. We use geometric stretching ($\Delta y_{i+1} = q\Delta y_i$) in the wall normal direction, with constant-size cells in the core region. On coarse meshes, severe stretching with $q$-factors of 2 or more may result. This does not, however, pose any difficulties for the flow solver. The example of the 1568 cell mesh shows that physically meaningful solutions can still be obtained, while a more even distribution of the same number of cells risks to fall into essentially an Euler computation with no indication of loss levels whatsoever.

Two meshes of respectively 1568 and 25088 cells were compared. While the fine mesh captures details that are ignored on the coarse mesh, for instance separation at the blunt trailing edge, the latter represents the global traits of the solution with practically the same accuracy as the former (wall pressure distribution, massflow, flow angles, and pressure ratio). Global parameters are compared in
Table 4.1, reporting absolute flow angle at exit ($\alpha_e$), axial velocity at inlet ($V_z/V_{z,\text{ref}}$), massflow error ($\Delta \dot{m}/\dot{m}$), and static ($\Pi = p/p_{\text{in}}$) and total pressure ratio ($\Pi_t = p_{t,\text{abs}}/p_{t,\text{abs, in}}$). A difference of 20% in total pressure loss at exit, $C_{p*}$, suggests that one fourth roughly of the losses on the 1568 cell mesh are of numerical origin.

<table>
<thead>
<tr>
<th>Mesh Size</th>
<th>$\alpha_e$ [deg]</th>
<th>$V_z/V_{z,\text{ref}}$</th>
<th>$\Delta \dot{m}/\dot{m}$ [%]</th>
<th>$\Pi$</th>
<th>$\Pi_t$</th>
<th>$C_{p*}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1568</td>
<td>50.85</td>
<td>12.50</td>
<td>0.15*10^{-4}</td>
<td>1.0087</td>
<td>1.102</td>
<td>0.0075</td>
</tr>
<tr>
<td>25088</td>
<td>47.04</td>
<td>12.63</td>
<td>0.21*10^{-4}</td>
<td>1.0088</td>
<td>1.100</td>
<td>0.0062</td>
</tr>
</tbody>
</table>

Figure 4.3 compares the velocity profiles along a blade normal section that cuts the blade twice, once at 1/4 of chord (position 1) and a second time at 3/4 (2). Hence, one complete blade passage can be observed. 16 cells are distributed across the passage on the coarse mesh, 64 on the fine mesh. Figures 4.3c and 4.3d are enlarged views of the superposition of 4.3a and 4.3b at respectively positions 1 and 2. The comparison shows that turbulent boundary layers, and in particular the loss relevant inner region, can be well represented with a surprisingly low number of points, if these are adequately distributed. Extrapolating to 3D, it was concluded that meshes of 100000 to 400000 cells should allow a realistic representation of inducer flows, including loss prediction, provided excessive skewness is avoided.

### 4.4 3D Computations

#### 4.4.1 Grid generation

Generation of the 3D grid proceeded in essentially two steps:

1. Complete description of the computational domain by a number of curves and surfaces, generated by a simple modelling system designed for that specific purpose.

2. Interactive grid point distribution and subsequent generation of surface and volume grids with the IGG system. Hub and shroud were thereby first created in 2D $r,\theta$-planes and subsequently projected onto their respective surface of revolution.

Even point distribution and minimum skewness were achieved by generating the grid in 5 blocks which were then fused into one. The inlet and exit sections have respectively been placed 3 axial cords upstream of the leading edge root and 2 axial cords downstream of the trailing edge root (Figure 4.4a). The dimensions of the different meshes are given in Table 4.2.

<table>
<thead>
<tr>
<th>Code</th>
<th># Cells</th>
<th># Cells BB</th>
<th># Cells ME</th>
<th>Dimensions $r \times \theta \times z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>000</td>
<td>380928</td>
<td>11904</td>
<td>7936</td>
<td>$32 \times 48 \times 248$</td>
</tr>
<tr>
<td>911</td>
<td>190464</td>
<td>2976</td>
<td>7936</td>
<td>$64 \times 24 \times 124$</td>
</tr>
<tr>
<td>011</td>
<td>95232</td>
<td>2976</td>
<td>3968</td>
<td>$32 \times 24 \times 124$</td>
</tr>
<tr>
<td>111</td>
<td>47616</td>
<td>2976</td>
<td>1984</td>
<td>$16 \times 24 \times 124$</td>
</tr>
<tr>
<td>022</td>
<td>23808</td>
<td>744</td>
<td>1984</td>
<td>$32 \times 12 \times 62$</td>
</tr>
</tbody>
</table>
The order of the indices is I, J, K, corresponding respectively to the radial, azimuthal, and axial directions. The code in the first column of Table 4.2 indicates the resolution with respect to the basic grid of 380928 cells, 011 signifying that only every second point has been retained in the azimuthal and axial direction. Mesh 911 with 190464 cells is refined in the radial direction. The columns named BB and ME contain the number of cells on blade-to-blade and meridional sections.

Taking into account the fact that hub and blades are rotating while the shroud is stationary, the initial spacings on the walls, $\Delta y/r_{tip}$, imposed on level 000 are respectively $7.7 \times 10^{-5}$, $5.5 \times 10^{-5}$, and $3.3 \times 10^{-4}$.

With classical structured grids, skew cells on meridional mesh surfaces are unavoidable in the region where the swept back leading edge joins the blade tip (Figure 4.4b). An unexpected consequence of this is the appearance of overlapping cells on coarse grid levels, if the intended clustering is applied. The relaxed clustering that was found possible in this region does not allow the shroud boundary layer to be captured accurately, nor does it permit significant clustering in the axial direction (Figure 4.4c). Any possible effort was made to minimize the extension of this region of sub-optimal mesh quality, which it must be stressed affects only the radial direction; on blade-to-blade surfaces, the meshes are quasi-orthogonal throughout, including at the leading edge, Figure 4.4c. Moreover, the sweep of the leading edge is not mirrored on the neighboring blade suction surface, giving rise to the warped stream-normal inner mesh surfaces of Figure 4.4d.

### 4.4.2 Computed cases

Table 4.3 summarizes the computed cases and essential global quantities.

<table>
<thead>
<tr>
<th>Case</th>
<th>Mesh Size</th>
<th>$\frac{\dot{m}}{\dot{m}_{ref}}$</th>
<th>$\frac{\Delta \dot{m}}{\dot{m}}$ [%]</th>
<th>$\Pi$</th>
<th>$\Pi_l$</th>
<th>$C_p$ *</th>
</tr>
</thead>
<tbody>
<tr>
<td>000</td>
<td>380928</td>
<td>.9191</td>
<td>.063</td>
<td>1.0055</td>
<td>1.0089</td>
<td>.00910</td>
</tr>
<tr>
<td>911</td>
<td>190464</td>
<td>.9295</td>
<td>.138</td>
<td>1.0055</td>
<td>1.0091</td>
<td>.00956</td>
</tr>
<tr>
<td>011</td>
<td>95232</td>
<td>.9092</td>
<td>.089</td>
<td>1.0055</td>
<td>1.0091</td>
<td>.00999</td>
</tr>
<tr>
<td>111</td>
<td>47616</td>
<td>.8115</td>
<td>1.08</td>
<td>1.0055</td>
<td>1.0103</td>
<td>.01380</td>
</tr>
<tr>
<td>022</td>
<td>23808</td>
<td>.7102</td>
<td>1.21</td>
<td>1.0058</td>
<td>1.0130</td>
<td>.01980</td>
</tr>
</tbody>
</table>

Five computations, all with identical boundary conditions, were performed, one on each of five different meshes: 000 on the basic level of 380928 cells to establish a reference solution, cases 911, 011, and 111 all coarsened azimuthally and axially, but each with different radial resolution, and case 022 keeping the basic radial resolution, but coarsened twice in the azimuthal and axial direction. The rationale behind these choices is to investigate systematically the influence of mesh resolution in the three directions.

### 4.4.3 Numerical Aspects

The present computations use the central scheme with added fourth order artificial dissipation. Robustness on high aspect ratio meshes is increased by multidimensional eigenvalue scaling of the dissipation factor (Martinelli, 1987). Residual smoothing uses variable coefficients according to Radespiel et al. (1990). Smoothing of the coarse grid corrections is performed before prolongation to the next finer grid level. Restriction is linear, prolongation constant. The inlet boundary condition imposes the absolute total pressure, the direction of the absolute velocity vector, and the absolute total temperature. The exit boundary condition imposes the static pressure at some reference radius and determines the radial pressure distribution form radial equilibrium.
4.4.4 Convergence
The following convergence relevant parameters were used for all 3D computations: a CFL number of 4, a fourth order dissipation coefficient \( \kappa(4) \) of 0.05 and a Martinelli exponent \( \sigma \) of 0.5 (Equation XX), no artificial dissipation on solid boundaries, 4-stage RK time integration with coefficients \( \alpha_1 = 0.25, \alpha_2 = 0.333, \alpha_3 = 0.5, \) and \( \alpha_4 = 1 \), evaluation of the artificial dissipation on each stage and evaluation of the viscous fluxes only on the first and third stages, 3- or 4-level MG in a V-cycle, one RK sweep on each grid level, smoothing of both the MG corrections and the residuals with coefficients \( \text{CFL}^*/\text{CFL} \) of respectively 2.0 and 1.85.

Figure 4.5 shows the residual drop associated with each of the conservative variables for the fine mesh case 000 and the evolution of massflow in the inlet and exit sections. In conformance with theory, the required number of MG cycles is independent of the mesh size, e.g., case 000 with 4-level MG converges in the same number of cycles as case 111 with 3-level MG. Starting from a uniform initialization, a sufficiently converged solution is obtained in 2000 MG cycles. The increased effort compared with the 2D computations is explained by the poorly conditioned mesh at the leading edge tip. Adjustment of a new massflow by modification of the exit pressure requires 200 to 1000 MG cycles.

4.4.5 Comparison with experimental data
Figure 4.11 compares the distribution of the azimuthal averages of absolute flow angle \( \alpha = \tan(V_\theta/V_z) \), loss coefficient \( C_{p^*} \), pressure coefficient \( C_p \), and angular momentum \( rV_\theta \) with the respective experimental data along a radial section, 0.026 axial chords downstream of the trailing edge root. \( C_p \) is defined as

\[
C_p = \frac{p - p_{\text{abs, in}}}{\frac{1}{2} \rho_{\infty} \Omega^2 \rho_{\text{sys}}^2 \rho_{\text{tip}}^2}
\] (4.1)

Losses are assessed by means of the normalized loss of rotary stagnation pressure,

\[
C_{p^*} = 1 - \frac{p_{\text{t, rot}}}{p_{\text{t, abs, in}}}
\] (4.2)

where \( p_{\text{t, rot}} \) is the total pressure associated with rothalpy,

\[
p_{\text{t, rot}} = p \left( \frac{T_{\text{t, rot}}}{T} \right)^{\frac{\gamma}{\gamma - 1}}
\]

\[
T_{\text{t, rot}} = T + \frac{W^2 - U^2}{2c_p}
\] (4.3)

For each of the four quantities, good to excellent agreement with the measurements is observed. The calculation with 64 radial sections, case 911, gives an excellent prediction of the radial gradients of \( \alpha, C_{p^*}, \) and \( rV_\theta \), while the two calculations with 32 radial sections, cases 000 and 011, do not capture the fine details of the radial gradients. These two calculations at 95232 and 380928 cells show very similar results, indicating that for this geometry, the streamwise and pitchwise mesh point densities are less sensitive parameters. As for the coarse mesh computation of 47616 cells, case 111, this clearly has insufficient resolution.

The general tendency towards overestimating \( C_p \) and, to a lesser extent, \( \alpha \) and \( rV_\theta \), might be accounted for by the somewhat lower massflow compared with the experimental massflow \( \dot{m}_{\text{ref}} \).

It can be noticed also that the calculation with the highest radial clustering, case 911, shows an increased loss coefficient in the shroud region. This can probably be connected to the increased upstream vortex strength in the leading edge shroud region, as seen in Figure 4.12, which is further discussed in Section 4.4.7 (pressure distribution and global flow field).
4.4.6 Global quantities

Globally, almost the same solution can be obtained on the 95232-cell mesh as on the 380928-cell mesh (Table 4.3). The massflow of case 011 differs from case 000 by only 1.1%. For case 111, the difference is far bigger, of the order of 10%. This might be attributed to the poor mesh quality in the leading edge tip region on meridional mesh surfaces, although even larger deviations observed in case 022, which retains 32 cells in the radial direction but has only 12 cells across the narrow blade passage, suggest that 24 cells between opposing walls can be considered a practical lower limit. Global conservation of massflow is excellent for cases 000 and 011, the error being well below one tenth of a percent.

Similar observations hold for pressure ratios and losses. Considering $C_p*$ and taking again case 000 as a reference, the differences are below 10% for cases 011 and 911, but of the order of 50% for case 111.

The main observation is the importance of sufficient mesh resolution in the radial direction in order to obtain consistent results, as was also seen by comparing the radial distributions in Figure 4.11.

4.4.7 Pressure distribution

The $C_p$ contours on pressure and suction side are shown in Figure XX for the fine mesh computation 000. These figures are also available for an independent computation of a similar configuration by Moore et al. (1990). Despite the coarse mesh used by Moore et al., their $C_p$-contours are in good qualitative agreement with the present ones.

Figures XX and XX compare the blade pressure distribution along sections at 5%, 50%, and 95% of span for respectively cases 011 and 111 with the reference solution 000. The 95232-cell mesh captures practically the same distribution as the four times finer mesh, with slightly lower values in some places. The 911 $C_p$ distributions are nearly identical to case 011. On the 47616-cell mesh, the good agreement along the tip section—while severe underprediction occurs at midspan and hub—is accounted for by the smoother mesh resulting from the relaxed clustering in the shroud boundary layer. These comparisons confirm the importance of radial resolution in obtaining reliable results.

4.4.8 Generation and distribution of losses

If the axial evolution of $C_p*$ is compared (in the azimuthal average) along sections at 10%, 50%, and 90% of duct height, only the 47616-cell mesh (111) is found to differ markedly from an otherwise coherent picture, with the exception of the hub and shroud leading edge regions, where the sensitivity to spanwise mesh resolution is once more demonstrated by an ~20% difference between level 911 on the one hand and levels 000 and 011 on the other hand. By the time the flow attains the trailing edge, intense mixing due to secondary flows has made this deviation disappear.

One important source of this secondary fluid motion is the centrifugal effect in the blade boundary layers. With appropriate clustering at the walls, this effect is comparatively easy to capture and is therefore well represented on the coarse as well as on the fine meshes. Representative for all cases, it is visualized in Figure 4.9 on level 011 (95232 cells).

It may be expected that such strong centrifugal secondary flows, which in contrast to the pressure-driven transverse flows are oriented radially outwards on both the suction side and pressure side and which are comparable in magnitude to the axial velocity in the inlet duct, have some impact on the distribution of losses. That this is indeed the case can be seen from Figures 4.6 and 4.8, which show contours of the loss coefficient $C_p*$ in two mutually perpendicular cross sections, revealing the structure inside the blade passage, which characterized by a low-loss core close to the hub and a progressive increase towards the shroud wall: Figure 4.6 is a meridional cut, comparing cases 911 and 000, and Figure 4.8 an axis-normal cut at mid-chord, comparing cases 911, 000, and 011. All five views clearly show that the net effect of the discussed flow structure is an accumulation of high-loss fluid at the casing, originating from the blade boundary layers. The dramatic changes between the different meshes demonstrate once more the importance of adequate mesh resolution, the spanwise direction being the most sensitive. The finest tested meshes would appear to represent a lower limit
for a quantitatively accurate prediction of this type of flow with important viscous effects. As a final note on this subject, we observe that the super-linear steepening of the loss gradients with mesh resolution also goes some way towards confirming the second-order accuracy of the spatial discretization.

4.4.9 Backflow and prerotation

Related to the loss distribution discussed in the preceding section is one most peculiar feature of inducer flows, in particular in the presence of leading edge sweep, namely a casing recirculation zone located at the inlet to the blade passage. In the pitch-averaged solution, or in any meridional section, this backflow appears as a powerful vortex in an otherwise unspectacular flow field, Figure 4.12. This backflow phenomenon is well known experimentally. Depending on the pump design and operating point, it may extend several diameters upstream (Brennen, 1994) and may occupy a substantial fraction of the span. In the present case, the vortex is seen to block some 10% to 20% of the annulus height. The flow ejected back out of the inducer has significant (absolute) swirl, and it has experienced high losses. Both these effects can be observed on Figure 4.10, which is the analogue of Figure 4.11 for a spanwise section 0.072 axial chords upstream of the LE root, showing radial profiles of loss coefficient, $C_p^*$, absolute angular momentum, $rV_\theta$, absolute flow angle, $\alpha$, and relative velocity, $W/U$.

Given its profound impact on the overall flow field, presumably associated with degraded performance, the importance of an accurate quantitative CFD prediction is manifest. Although a systematic analysis is beyond the scope of the present work, those observations which were made shall therefore not go unreported. Regarding first the mechanism behind the generation of the backflow vortex, tip leakage flows intensifying with blade loading were thought to be the cause, referring again to the discussion in Brennen (1994). However, the fact that the vortex is observed practically unchanged in calculations with or without a modelled tip gap would seem to invalidate this hypothesis, even though the relative motion between the blade tips of the unshrouded impeller and the stationary shroud presumably play an essential role.

Concerning second the size of the vortex, and in particular its upstream extension, this is known to increase with blade loading and thus with reduced massflow. The series of solutions shown in Figure 4.12, all obtained on level 911 and representing massflows $\dot{m}/\dot{m}_\text{ref}$ of 0.936 (Figure 4.12a), 0.938 (b), and 0.941 (c), indeed fits the empirical trend. However, given the relatively small variations in massflow and the comparatively large differences in vortex length, a second, numerical factor might be the main reason. It has been found that the amount of upstream extension is sensitive to the initial solution in the following manner. Starting from a (reasonable) uniform initial flow field immediately on the finest grid tends to yield a short recirculation zone, while initialization of the finest grid level with full multigrid favors a long recirculation zone. Experimental data on the calculated geometry indicate a short recirculation zone.

Associated with the physical prerotation from the vortex is a certain amount of numerical prerotation which appears on all meshes, since none satisfies the severe criteria with regard to axial clustering established in Section 4.3.1.

4.5 Conclusions

A systematic investigation to obtain reliable CFD simulations of the complex three-dimensional flow in an LH2 rocket pump inducer has been performed. The analysis of a representative two-dimensional blade-to-blade section has allowed to establish a certain number of criteria for low mesh sensitivity and accurate loss prediction. Consistency of the results has been demonstrated by comparing the solutions obtained with different physical models on meshes of vastly different resolution. Appropriate clustering in the turbulent boundary layers, even on rather coarse meshes, and the avoidance of skew meshes have emerged as key parameters. The latter is achieved through the use of non-matching periodic boundaries.

These guidelines have been applied to the three-dimensional case, where different meshes between 23808 and 380928 cells were compared. A certain minimum resolution is required in the radial
direction, determined to be about 64 cells. Confirming the conclusions of the 2D analysis, between 100000 and 200000 well distributed mesh points offer a realistic representation of inducer flows. The dominant effect clearly appears to be the radial grid point distribution. Completely mesh independent results seem to require at least 64 blade-to-blade sections.

It should be stressed that the presented solutions were obtained by straightforward application of EURANUS, without any tuning of parameters influencing the physical model. The demonstrated consistency of results and good agreement with experimental data thus indicate a genuine predictive capability.

Figure 4.1 Inducer 2D tip section, comparison of (a) skew mesh with matching periodic boundaries, (b) quasi-orthogonal mesh with non-matching periodic boundaries, (c) global view of non-matching grid.
Figure 4.2  2D blade-to-blade analysis of inducer tip section, (a) quasi-orthogonal non-matching mesh in 5 blocks, (b) pressure contours obtained on single-block mesh (2816 cells).

Figure 4.3  Inducer 2D tip section, turbulent velocity profiles along a blade normal section, 1 = at 1/4 of chord, 2 = at 3/4 of chord, (a) 1568 cells, (b) 25088 cells, (c) comparison of 1568 and 25088 cells at position 1 (zoom), (d) at position 2.
Figure 4.4 Inducer, views of the mesh, (a) global view (level 010, 190464 cells), (b) axial projection of the suction side surface grid (level 111, 47616 cells), (c) detail of the shroud surface grid at the leading edge (level 000, 380928 cells), (d) axial projection of blad-to-blade mesh surfaces (level 000, 380928 cells).
Figure 4.5  Inducer convergence history, case 000 (380928 cells), (a) residuals, (b) massflow.

Figure 4.6  Inducer, contours of the loss coefficient $C_p*$ in a meridional cut, (a) case 911, (b) case 000.
Figure 4.7 Inducer, isolines of axial velocity, $V_Z$, in an axis-normal section at mid-chord, (a) case 011, (b) case 000, (c) case 911.

Figure 4.8 Inducer, isolines of the loss coefficient, $C_{p^*}$, in an axis-normal section at mid-chord, (a) case 011, (b) case 000, (c) case 911.

Figure 4.9 Inducer, velocity vectors on a mesh surface near the pressure side, case 011.
Figure 4.10  Inducer, radial profiles of the azimuthally averaged solution 0.072 axial chords upstream of the leading edge root, (a) loss coefficient, $C_{p}^{*}$, (b) absolute angular momentum, $rV_{\theta}$, (c) absolute flow angle, $\alpha = \text{atan}(V_{\theta}/V_{m})$, (d) relative velocity, $W/U$.
Figure 4.11  Inducer, radial profiles of the azimuthally averaged solution 0.026 axial chords downstream of the trailing edge root, (a) loss coefficient, $C_p^*$, (b) absolute angular momentum, $rV_\theta$, (c) absolute flow angle, $\alpha = \tan(V_\theta/V_m)$, (d) relative velocity, $W/U$. 
Figure 4.12 Streamlines for azimuthally averaged inducer solution, case 911 with tip gap, (a) 0.936 m_{ref} with full multigrid, (b) 0.938 m_{ref} without full multigrid, (c) 0.941 m_{ref} without full multigrid.

Figure 4.13 Numerical upstream influence of a rotor (prerotation, M_{rel,tip} = 0.03, b_{tip} = 45 deg), axial evolution of the azimuthal average of V_θ at mid-height between two cylinders with four rotating blades, (a) 8 cells upstream of leading edge, (b) 16 cells, (c) 32 cells, 7+20 cells.
4.6 Effect of Spatial Discretization in Viscous Flow

4.6.1 Reflections on sources of error

A discrete numerical solution will in general never match exactly the real physical flow it is meant to represent, the error being composed of two parts: a modelling error and a numerical error.

Examples for modelling errors would be those introduced by the treatment of turbulence (for which the category is obvious from the term turbulence model, although the turbulence models themselves rely on the evaluation of flow gradients, which in turn introduces numerical errors), by the fluid thermodynamic properties (the accuracy of the state equation, laws linking forces and deformation, and temperature gradients and heat transfer, chemical reactions and dissociation phenomena at high temperatures), by a simplified description of two-phase flows (wet steam considered as a single fluid in thermodynamic equilibrium), by a simplified representation of details of the geometry (tip gaps, secondary flow paths, disk leakage and cooling hole flows), by a simplified treatment of combustion and heat transfer (turbine blade cooling and combustion), by a lower dynamic level of approximation (inviscid or potential flow), by the throughflow and distributed loss models, by time dependent versus steady state analysis and various shades in between that have been proposed specifically for turbomachinery.

Numerical errors are those arising from the discrete numerical solution of the set of partial differential equations (which by definition exactly describe the adopted physical model) and depend on factors such as spatial and time resolution, the spatial and time discretization schemes, and the achieved or achievable convergence level.

In order to make a given physical flow problem mathematically tractable with given computing resources, a number of assumptions and simplifications have to be introduced. According to the aspect concerned, these may be sorted into the categories space (the represented level of geometrical detail), time (the represented level flow unsteadyness), fluid physics (the fidelity of the thermodynamic and transport properties), and boundary conditions (which involve elements of both space and time). It is here assumed that fluid can be considered as a continuum.

From an economical perspective, one might conclude that a reasonable balance must be struck between the two sources of error, and that, unless there are compelling reasons to the contrary, in an efficient model neither error should be orders of magnitude larger or smaller than the other, for in that case one could make a sacrifice where the error is small and improve where it is large, thereby reducing the total error at no additional cost. Although, with the possible exception of spatial and time resolution, the available options will usually be discrete choices, such as the type of turbulence model, may be separated by large cost gaps, thinking for example of steady-state vs. unsteady simulations, and although the situation may be further complicated by certain physical principles one might wish the model to obey, conservation and the second law coming to mind, in the interest of obtaining the best possible (in the sense of most accurate) result with a given amount of resources, the described trade-off between numerical and modelling errors should from time to time be recalled and verified.

For numerical schemes with separate space and time discretizations the numerical accuracy of a steady state solution is determined only by the spatial discretization. In the following, we propose to examine the numerical error resulting from different spatial discretizations of the inviscid terms, and to assess the sensitivity to mesh quality and refinement.

4.6.2 Presentation of the test case

The test case is the laminar boundary layer developing along the walls of a 2D channel, Figure 4.14. The channel half-height is 1/20 of the channel length which is dimensioned such that at exit the boundary layer (as delimited by the 99% velocity) occupies 1/10 of the channel half-height. The fluid is a perfect gas with the thermodynamic properties of air (ratio of specific heats $\gamma = 1.4$, specific gas constant $R = 287 \text{ J/K/kg}$) but with 100 times the dynamic viscosity of air at $\mu = 0.00147 \text{ Pa s at } T_{\text{ref}} = 300 \text{ K}$), variable with temperature according to Sutherland’s law. This choice hails from the industrial background which provided the incentive for the systematic study here reported. It offers the advantage of avoiding excessive cell aspect ratio and ensures rapid convergence. The Prandtl number
is 0.708. Inlet total pressure and temperature of respectively 150 000 Pa and 300 K with an exit static pressure of 110 000 Pa result in a mildly compressible flow at an average free stream Mach number around 0.67. Profiles will be examined at 77% length where the Reynolds number based on distance from the leading edge is 820 000. The channel wall is adiabatic.

4.6.3 Description of the meshes

Calculations have been performed on seven different meshes. Figure 3.3 shows the wall-normal grid point distribution for six of them. Also indicated is the approximate range of the expected velocity and thermal boundary layer thicknesses at the location of the test section, estimated from the Blasius solution with corrections for the effect of frictional heating on density and viscosity, which would suggest \( \delta = 0.023 \) m at \( \text{Re}_x = 820 000 \). (Although there also is a mild favorable pressure gradient, for the considered test case geometry and flow conditions its effect is small compared with that of heating.) Concerning the thermal boundary layer, the ratio \( \delta_T/\delta \) is proportional to \( 1/\sqrt{\text{Pr}} \). For gases, the Prandtl number is of the order of 1 and the thermal and velocity boundary layers will therefore be of similar thickness, with an estimated value of \( \delta_T = 0.028 \) m in the present case. For liquids, by contrast, typical values for the Prandtl number range between 10 and 1000, leading to thermal boundary layers which can be considerably thinner than the velocity boundary layer. (The turbulent Prandtl number, on the other hand, always is of the order of unity, with the possible exception of liquid metals, see Kays, 1994.) In laminar heat transfer applications involving liquids, it should therefore be carefully checked if the thermal boundary layer does not impose a more severe clustering requirement than the inertial one. By analogy with the law-of-the-wall coordinate,

\[
y^+ = y \frac{u_\tau}{\nu} \quad \text{where} \quad u_\tau = U_\infty \sqrt{\frac{C_f}{2}}
\]

one might define a thermal boundary layer coordinate as

\[
y_T^+ = y \frac{u_q}{a} = y \text{Pr} \frac{u_q}{\nu} \quad \text{where} \quad u_q = U_\infty \sqrt{\frac{C_h}{}}
\]

is a “heat flux velocity”, where \( a \) denotes thermal diffusivity, and where the heat transfer coefficient \( C_h \) is based on some maximum heat flux (not necessarily occurring at the wall). For the Blasius solution, the inertial and thermal law-of-the-wall coordinates can then be expressed in function of the Reynolds and Prandtl numbers as

\[
y^+ = \sqrt[3]{0.332} \frac{y}{x} \frac{\text{Re}_x^{3/4}}{3} \quad \text{and} \quad y_T^+ = \sqrt[3]{0.332} \frac{y}{x} \frac{\text{Re}_x^{3/4} \text{Pr}^{2/3}}{3}
\]

which would suggest that the ratio of thermal to inertial clustering scales as \( \text{Pr}^{-2/3} \).

In the streamwise direction, all meshes count 128 uniform cells. The tested meshes are representative of what might be encountered in industrial applications where with a single multiblock mesh of a given total size (in terms of grid points) one must accommodate different flow phenomena in sometimes highly complex geometries, resulting in a compromise which, at least for structured meshes, also involves grid generation time as a limited resource. An attempt is here made to identify (i) the meshing requirements for different schemes and (ii) schemes which are tolerant with respect to mesh quality. Table 4.4 gives an overview of the tested meshes, listing the number of cells in the wall normal direction, and the initial spacing at the wall, in terms of physical distance, as a multiple of boundary layer thickness, and in law-of-the-wall coordinates (based on the Blasius solution).

Mesh #2 is uniform with 32 cells across the channel half-height, which leaves only between three and four cells in the boundary layer.

Mesh #3 is strongly clustered at the wall, with \( y_T^+ < 1 \). Despite an increase in the number of cells, this leads to fairly large spacing around the boundary layer edge where the resolution is only about twice as good as for Mesh #2. Of the total of 48 cells, 20 of constant size are placed in the channel center. This type of distribution is inspired by turbomachinery applications, where sufficient spatial resolution of the core flow was found to be essential.
We mention in passing that the distribution algorithm is another outcome of the present work. It combines geometric series clustering at either end of a segment (an edge of a block) with cells of constant size in the middle, taking as input the total number of grid points, the left and right initial spacings (which may be different), and the number of constant cells. Together with the criteria that the stretch factor be the same in the left and right clustering zones and that the cell size be continuous between the clustered and uniform zones, this defines a unique grid point distribution with an automatic balance of the points available for clustering between the left and right zones. Implemented in a interactive grid generation system, the algorithm invites swift experimentation and since its inception has served in the generation of many hundred turbomachinery (and other) meshes.

Mesh #4 is the inverse of Mesh #3, in the sense that it contains 32 uniform cells near the wall and 16 stretched cells towards the center, offering a $y_1^+ < 10$, the idea being to provide (and examine the effect of) equal resolution in three zones of particular interest, namely the high-shear, zero-heat flux zone at the wall, the high-second derivative zone at the ‘knee’ of the velocity profile, and the (numerically caused) negative-entropy zone following it and which may extend out up to two boundary layer thicknesses.

Mesh #5 is structurally identical with Mesh #4, except that the uniform refined region has been reduced to the thickness of the boundary layers, leading to a mesh that is approximately twice as fine as Mesh #4.

Mesh #6 is a reference mesh fine-tuned for this particular boundary layer. Compared with Meshes #3, #4, and #5, the number of cells has been doubled and the grid points are distributed for maximum accuracy, starting with $y_1^+ = 1$ at the wall, using only mild stretching across the boundary layer and terminating with strong stretching in the core flow region.

Mesh #8, finally, is a normal industrial mesh of 48 cells using the above described standard distribution with reasonable clustering near the wall ($y_1^+ = 3$) and 10 constant cells in the center. In 3D applications, however, resolving a blade-to-blade passage with nearly 100 grid points is costly even today and may be unacceptable in design, time-dependent, or multi-blade row applications. The first coarse level of this mesh, which with $y_1^+ = 7$ and 24 well-distributed cells still looks reasonable, has therefore been included in this campaign. It shall be referred to as Mesh #81.

4.6.4 Tested spatial discretization schemes

A unified framework is used in Euranus to represent both central and upwind schemes. The inviscid flux function is thereby composed of a second order central physical flux, which is the same for all schemes, and a dissipative flux, which can represent upwind or central schemes. In the i-direction,

$$\left(\hat{F} \cdot \hat{S}\right)_{i+1/2} = \hat{F} \left(1/2 (V_i + V_{i+1})\right) \cdot \hat{S}_{i+1/2} - D_{i+1/2}$$  \hspace{1cm} (4.7)
Among the several possibilities for evaluating the central physical fluxes on the cell faces, averaging of the primitive variables \( V = (\rho, u, v, w, p) \) was found to work well in practice. Eq. (4.7) reflects this choice, used for all calculations presented in this thesis. Likewise, the interpolation is arithmetic rather than weighted. Although the formal accuracy of the scheme is lost on non-uniform meshes if the cell geometry is not taken into account, the loss of accuracy is found to be negligible in practice if the mesh spacing is varying smoothly. This condition is usually satisfied, even in an industrial context. Holmes and Tong (1985) discuss the advantages and drawbacks of four different possibilities in the framework of an Euler solver, considering averaging of the fluxes, of the conservative variables, and two intermediate averaging procedures defined by calculation of respectively the volume or mass fluxes from averaged velocities or velocity density products and separate averaging of the pressure and the variables convected by the chosen flux, pointing out that the original Jameson scheme is arranged around the cell face volume flux.

We observe that the second order central scheme has no inherent dissipation mechanism because the even derivatives in the Taylor series all cancel out.

Curbing the tendency of second and higher order schemes to generate oscillations around captured discontinuities and generally in regions of strong gradients requires some non-linear mechanism to locally introduce increased numerical dissipation. One such mechanism is based on the concept of flux limiters, another on blending in central second order dissipation through a gradient sensor. On the other hand, the TVD approach (the limiters) can be viewed as a rational way of introducing artificial dissipation into central schemes. In principle, either approach can be applied with both central and upwind discretizations, although historically the idea of limiters originates with upwind schemes and the pressure sensor blend with central schemes. Yee’s symmetric TVD schemes are an example of using limiters with central schemes. Addition of central second order dissipation to upwind schemes is less common, and we are in fact not aware of any in-depth analysis of this variant. Euranus contains implementations of the classical central scheme with both a scalar and a matrix valued dissipation, the symmetric TVD scheme, and a fully upwind second order extension of Roe’s (1981) flux difference splitting (FDS) TVD scheme.

For the scalar central JST scheme, the artificial dissipative flux on i-cell faces is schematically given by

\[
D_{i+1/2} = \varepsilon_{i+1/2}^{(2)} \delta U_{i+1/2}^{AD} + \varepsilon_{i+1/2}^{(4)} \delta^3 U_{i+1/2}^{AD}
\]

where

\[
\varepsilon_{i+1/2}^{(2)} = \frac{1}{2} \kappa^{(2)} \tilde{\lambda}_{i+1/2} \max(v_{i-1}, v_i, v_{i+1}, v_{i+2})
\]

\[
\varepsilon_{i+1/2}^{(4)} = -\max \left( 0, \frac{1}{2} \kappa^{(4)} \tilde{\lambda}_{i+1/2} - \varepsilon_{i+1/2}^{(2)} \right)
\]

We say schematically because here again various choices are possible as to where the scaled non-linear dissipation coefficients \( \varepsilon^{(2)} \) and \( \varepsilon^{(4)} \) are computed (in cell centers or on cell faces) and at what level they are multiplied into the differences. The parameters \( \kappa^{(2)} \) and \( \kappa^{(4)} \) are constant dissipation coefficients with values 1 and 0.1 found empirically and in agreement with the amount of dissipation introduced by upwind schemes of the respective order. To ensure a scale invariant amount of artificial dissipation, the spectral radius must be proportional to the cell face area. On meshes with high-aspect ratio cells, this scaling may, however, lead to insufficient dissipation in the stretched direction, with an attendant loss of robustness. This problem can be overcome by imposing a lower limit on the area-scaled spectral radius in function of the other two directions (Martinelli, 1987),

\[
\tilde{\lambda}_{i+1/2} = \lambda_\xi \max \left[ 1, \left( \frac{\lambda_{\eta}}{\lambda_\xi} \right)^\sigma, \left( \frac{\lambda_{\xi}}{\lambda_\xi} \right)^\sigma \right]
\]

where

\[
\lambda_\xi = \sqrt{\xi \cdot \tilde{S}^2} + \left| e \tilde{S}^\eta \right|
\]

and where a recommended value for the exponent \( \sigma \) is 1/2. The second order artificial dissipation is
activated by the maximum of four consecutive values of a switch function \( \nu_i \), Eq. (4.9), which in the standard implementation is a normalized second difference of the pressure,

\[
\nu_i = \frac{p_{i+1} - 2p_i + p_{i-1}}{p_{i+1} + 2p_i + p_{i-1}}
\]  

(4.11)

In the following, an attempt shall be made to present all schemes in a unified frame in order to make the structural similarities and conceptual differences stand out most clearly. We first note the relation between variations in the characteristic, conservative, and primitive variables,

\[
\delta W = R^{-1} \delta U = L^{-1} \delta V
\]  

(4.12)

where the matrices \( R \) and \( L \) respectively contain the right eigenvectors of the jacobian matrix of the flux vector with respect to conservative and primitive variables, e.g. Hirsch (1990). The dissipative flux corresponding to the various schemes can then be written as follows:

**Scalar:**

\[
d = \hat{\lambda}[\epsilon(2)\delta U + \epsilon(4)\delta^3 U]
\]  

(4.13)

**Matrix:**

\[
d = R|\Lambda|[\epsilon(2)\delta W + \epsilon(4)\delta^3 W]
\]  

(4.14)

**TVD:**

\[
d = R|\Lambda|(1 - Q)\delta W
\]  

(4.15)

The non-linear dissipation coefficients \( \epsilon(2) \) and \( \epsilon(4) \) are defined as in Eq. (4.9), but without the spectral radius, which now appears explicitly in (4.13) and is replaced by the absolute eigenvalues in Eq. (4.14). In the scalar dissipation formulation, there would appear to be only a limited choice in the variables of which differences are taken since the variations should parallel those of the variable for which the respective equation is solved, for instance the conservative variables. In a matrix formulation, on the other hand, the differences can in principle be taken on any complete set of dependent variables so long as those variations are then translated through the proper Jacobian matrix into variations of the dependent variables for which one solves and the gradients of which are represented by the spatial differential operator we wish to approximate. Choosing, for example, the primitive variables and using Eq. (4.12), instead of Eqs. (4.14) to (4.15) we could construct the schemes

**Matrix:**

\[
d = R|\Lambda|L^{-1}[\epsilon(2)\delta V + \epsilon(4)\delta^3 V]
\]  

(4.16)

**TVD:**

\[
d = R|\Lambda|L^{-1}(1 - Q)\delta V
\]  

(4.17)

Analogous forms can be written for differences of the conservative variables, where \( R^{-1} \) would replace \( L^{-1} \) and \( U \) would replace \( V \). For a central-type matrix dissipation, the natural choice would seem to be the primitive variable formulation, and for upwind TVD schemes the characteristic variable formulation. These are the formulations implemented in Euranus and compared below.

In the above expressions, the diagonal matrix \( |\Lambda| \) contains the absolute eigenvalues of the flux jacobian

\[
\Lambda = \frac{\partial (\hat{\Phi} \cdot \hat{S})}{\partial U} = R|\Lambda|R^{-1}
\]  

(4.18)

In the unified implementation of Euranus, the limiter function \( Q \) acts on an effective ratio, \( r \), of the differenced variables. Defining, for instance for characteristic variables, the left and right ratios

\[
r^+ = \frac{\delta W_{i+3/2}}{\delta W_{i+1/2}} \quad \text{and} \quad r^- = \frac{\delta W_{i-1/2}}{\delta W_{i+1/2}}
\]  

(4.19)

the fully upwind FDS TVD scheme is given by
while the symmetric TVD scheme is obtained by applying the limiter to a blend of the left and right ratios. Of the various possibilities that have been examined by Lacor et al. (1993), ranging from \( \min(r^-, r^+) \) (most dissipative and most robust) to \( \max(r^-, r^+) \) (most compressive and least robust), the neutral van Leer average has been selected for this test,

\[
    r = \begin{cases} 
        r^- & \text{if } \lambda > 0 \\
        r^+ & \text{if } \lambda < 0 
    \end{cases} \quad (4.20)
\]

As for the limiter itself, although all classical TVD limiters are implemented in Euranus, only the van Leer limiter,

\[
    Q(r) = \frac{r + |r|}{1 + r} \quad (4.22)
\]

is used in the present examples because it is a smooth function of its argument (except where the slope changes sign, where it is zero) and offers a well-balanced compromise between compressive and dissipative properties.

The affinity between the matrix central and symmetric TVD schemes can be appreciated from the fact that if the respective non-linear mechanisms are taken out (the damping mechanism represented by the second order dissipation in the matrix central scheme, by choosing \( \varepsilon^{(2)} = 0 \), and the monotonicity preserving mechanism represented by slope limiting in the symmetric TVD scheme, by choosing as effective ratio the algebraic mean and setting \( Q(r) = r \)) and if differences of the same variables are used, Eqs. (4.14) and (4.15) or (4.16) and (4.17), then for a fourth order dissipation coefficient \( \kappa^{(4)} = 0.25 \) both become identical.

4.6.5 Other numerical parameters

All computations use a four stage Runge-Kutta scheme with optimized coefficients in function of the selected spatial discretization. Convergence to the steady state is accelerated through multigrid with the maximum allowable number of grid levels in a V-cycle (performing 1, 2, 4, 8, 16, 32 time steps on respectively grid levels 1, 2, 3, 4, 5, 6), through local time stepping, and through implicit residual smoothing with a time step-dependent variable coefficient. Restriction and Prolongation in the multigrid are both linear, no additional smoothing is applied to the multigrid corrections. For the scalar central scheme, Martinelli scaling of the spectral radius with an exponent of 0.5 was activated. To illustrate the resulting convergence rates, we refer to Section 4.6.6 and cite as an example the symmetric TVD scheme on mesh 8, which requires 700 MG cycles to reach machine accuracy after a 5-order residual drop.

Averaging to obtain the cell face values in the inviscid fluxes is arithmetic (as opposed to weighted) and performed on the variables (as opposed to the fluxes). The gradients in the viscous fluxes are calculated directly on the cell faces by applying the divergence theorem to an auxiliary control volume formed by the left and right half-cells (as opposed to being first calculated in the cell vertices and then interpolated onto the cell faces, which would be computationally less expensive but also less accurate).

Neither with the central nor with the TVD schemes is any artificial or upwind dissipation applied on physical boundaries (inlet, outlet, and solid walls): the flux term \( D \) or \( d \) is set to zero. Despite the fact that it leads to an imbalanced difference for the first inner cell, the residual of which essentially receives a third or first difference instead of a fourth or second difference, this treatment is invariably found to yield the highest accuracy, in particular as regards skin friction and heat flux at solid walls, without causing any perceptible negative repercussions on robustness, at least for the Mach numbers encountered in turbomachines. It offers the additional advantage that global conservation of physical fluxes is preserved.
4.6.6 Convergence

Convergence histories for the five compared schemes are shown in Figure 4.16 for the computations on mesh #8. These curves confirm the empirical fact that (within limits that could be loosely stated as the criterion that the augmented numerical dissipation should drive the scheme towards a first order upwind discretization, but not beyond) increased dissipation has a favorable effect on robustness and convergence rate, with two patterns clearly emerging: while the first order upwind and scalar central schemes exhibit a rapid and monotone drop of the residual from the uniform initial solution down to machine accuracy, the less diffusive schemes with matrix-valued dissipation, irrespective of any limiter-enforced TVD-like conditions, all pass through an initial phase of irregularly varying and even increasing residual. Once this has been overcome, however, the same convergence rate as with the less accurate schemes is recovered. Initialization of the fine-grid solution with full multigrid presumably would have a beneficial effect but has not been tested here.

The observed penalty due to higher-order matrix-type dissipation in terms of increased total iteration count depends on the mesh and generally remains small, as witnessed by Figures 4.17 and 4.18, which for the first order upwind, scalar central, and symmetric TVD scheme show the convergence histories on all tested meshes, grouped by scheme. Another, possibly more serious problem appears from these figures, namely the sudden slowdown which occurs after about 2 orders of residual reduction for the symmetric TVD and first order upwind schemes on the two meshes with the strongest clustering, #3 (red) and #6 (cyan). An illuminating analysis of this breakdown of multigrid convergence on high-aspect ratio meshes has been given by Pierce (1997), who has identified semicoarsening in conjunction with a matrix-valued time step as an effective remedy, which has, however, the inconvenience of being incompatible with implicit residual smoothing. The fact that the slowdown does not yet appear with the central scheme is attributed to the Martinelli scaling of the spectral radius. At present, no equivalent technique is applied in any of the matrix and upwind schemes, and the possibility of doing so would clearly merit investigation if these more advanced discretizations were to be routinely used for turbomachinery applications.

As a consequence of the discrete action of both the directional switching and the limiter, convergence was particularly difficult to obtain with the FDS TVD scheme. On some meshes, it was found necessary to provide a fixed direction of the wind as external input, which obviously would not be possible for more realistic test cases. On one mesh (#4), even this measure does not allow convergence with the van Leer limiter, and the shown results for this mesh exceptionally were obtained with the minmod limiter. Interestingly, on this same mesh the unlimited fully upwind scheme does not converge, either. The potentially deleterious effect of the limiter in the FDS TVD scheme is once more illustrated on mesh #8, Figure 4.16, where the convergence rates of the unlimited scheme (black) and with the van Leer limiter (cyan) are compared. The limited calculation ultimately also attains machine accuracy, but only after about 5000 iterations, ten times as many as are required for the unlimited scheme. No such convergence problems were encountered with the symmetric TVD scheme. In summary, it would therefore seem to be the high-order upwind character which is most problematic through its tendency to limit cycles, and one may wonder if upwinding is not best reserved as an effective means of locally introducing large amounts of dissipation, to be associated with an essentially central discretization of the higher-order background dissipation.

While convergence problems usually are associated with fine meshes, caused for example by resolved intrinsically unsteady flow phenomena such as vortex shedding, the present results show that for certain types of spatial discretizations, strong transverse flow gradients arising in poorly resolved viscous shear layers can be equally problematic.

4.6.7 Cost

As implemented in Euranus and for this 2D test case, the cost of all tested schemes is comparable (measured as CPU time per iteration). Taking as a basis the scalar central scheme, there is no cost penalty when going to a TVD scheme (±1%), while the matrix central scheme costs 19% more and the first order upwind scheme 8% less. In this test, the artificial dissipation terms have been evaluated on each Runge-Kutta stage, the physical viscous terms only on the first and second stages. Although
a multigrid Fourier analysis by Zhu (1996) on a simplified 1D model problem indicates that there should not be any further improvement if the artificial dissipation is evaluated also on the third and fourth stages, for complex 3D test cases this slightly more expensive strategy is sometimes found to be more robust. The described choice also lets appear any cost difference in the artificial dissipation formulation most prominently.

### 4.6.8 Results

The motivation for this study was provided by the puzzling observation that the spatial discretization scheme most used in practice, the scalar central scheme with central fourth (and possibly second) order artificial dissipation, although dissipative by design to ensure numerical stability, leads to the generation of negative entropy in smooth viscous flows, mostly in the vicinity of solid walls.

In Euler calculations, the generation of an entropy layer along curved solid walls is a familiar and reasonably well understood phenomenon, in the cell centered finite volume implementation of Euranus related to the fact that the wall pressure has to be obtained through some form of extrapolation. As a consequence of a slight mismatch between this extrapolated pressure and the pressure that would result naturally if the wall were an inner streamline in a larger solution domain, the velocity vector in the first inner cell centers is forced off the wall-parallel direction, thereby acquiring a component in the wall-normal direction, where a curvature-related pressure gradient acts. Losses then arise from the work of this spurious wall-normal velocity component against the centrifugal pressure gradient, much in the same way the friction force functions in the distributed loss model. To facilitate mental visualization, the reader is invited to imagine the test case used to verify the above reasoning, namely a 2D free vortex flow bounded by Euler walls with a hub-ratio of 1/2.

One might therefore think that a similar type of mistuning, introduced through the wall boundary condition, would be responsible for the entropy anomaly observed in viscous solutions. It is here shown to be associated with overshoots in the velocity profile (for which the scalar central scheme is well known) and thus detached from the wall, caused exclusively by the spatial discretization scheme used for the convective terms. The connection with velocity overshoots is in fact immediately obvious from the incompressible laminar flat plate, if the total pressure is taken as a measure for entropy. Strong gradients perpendicular to the flow direction cause locally high levels of artificial or upwind dissipation, which, contrary to the intuitive notion “dissipation = losses = decrease in total pressure” that is usually applied when considering the evolution of entropy in numerical flow solutions, are responsible for the observed total pressure rise when following a streamline.

Results are shown in Figures 4.22, 4.23, and 4.24, as well as 4.21. The figures have been selected to allow a rapid visual assessment of the tested schemes and their sensitivity to mesh spatial resolution. The points shown in the profiles correspond to cell center values; no averaging to cell corners has been performed (which would have a smoothing effect).

As a first overview, Figure 4.22 presents a direct comparison of the five tested schemes on mesh #81 through the classical streamwise (a) and normal velocity profiles (b) as well as profiles of static (c) and total temperature (d) and total pressure (e). Accuracy is judged by how closely the calculated profile follows the reference solution (the symmetric TVD scheme on mesh #6), an by the predicted wall temperature. In this first respect, the matrix central and symmetric TVD schemes turn out to be the most accurate by far, while (on this relatively coarse mesh) the scalar central scheme falls somewhere between the first order upwind and matrix second order schemes. Note that for this flow with a (quasi-)uniform pressure field, the pressure switch-activated second order artificial dissipation would have strictly no effect in the steady-state solution.

As with convergence, the FDS TVD scheme again occupies a place apart. On the one hand, its theoretical intrinsic second order-accuracy and targeted numerical dissipation are proven by the fact that, no matter which variable is examined, the influence of the boundary layer is not diffused far into the free stream, as is the case for the first order upwind and scalar central schemes. On the other hand, however, the solution is marred by a peculiar overshoot around the ‘knee’ of the velocity profile, which we attribute to the combined action of upwind extrapolation and the limiter in this region of
strongly varying slopes, and, perhaps partially as a consequence, by a poor prediction of wall temperature (43\% error based on $T_{\text{wall}} - T_{\infty}$ compared with −17.5\% for the symmetric TVD scheme, −9.5\% for the matrix central scheme and +8.5\% for the scalar central and first order upwind schemes).

The overshoot in velocity and total pressure giving rise to extended regions of negative entropy with the scalar central scheme is clearly visible in Figures 4.22a and e, where it is seen that the matrix central scheme also presents a mild overshoot. Regarding the two tested central/symmetric schemes with matrix dissipation, the tradeoff is between slightly better accuracy with the matrix central scheme and monotone behavior with the symmetric TVD scheme.

Contrary to the unphysical overshoot in total pressure, a similar local maximum in total temperature, Figure 4.22d, is physically correct, as confirmed by Figure 4.19 where we have plotted the Blasius solution for the temperature profile on a laminar adiabatic flat plate for five Prandtl numbers between 0.0001 and 2. For a Prandtl number of unity, the total temperature is seen to be constant and (for this particular flow) the shear work and heat conduction terms in the energy equation exactly cancel each other, meaning that all dissipated heat is immediately conducted away. For Prandtl numbers less than unity, conduction prevails leading to an adiabatic wall temperature below the free-stream total temperature, while for Prandtl numbers larger than unity viscous dissipation dominates, resulting in adiabatic wall temperatures above the free-stream total temperature. Only for Prandtl numbers far away from unity is there no over-/undershoot in the total temperature profile to the respective other side.

The total temperature profile, because of its peculiar shape and narrow range (11\% of the static temperature in the present case) is a sensitive indicator of solution accuracy, as is total pressure. In Figures 4.23 and 4.24 both are plotted on all meshes, grouped by scheme in an attempt to provide a visual impression of the mesh sensitivity of the different schemes.

This representation essentially confirms the preceding analysis on the coarse mesh (#81). Excellent coherence is obtained with central/symmetric-type second order matrix schemes which of the tested schemes are least sensitive to grid density. The scalar central scheme produces total pressure overshoots on all meshes except the very fine reference mesh (#6). Although more accurate globally, the spread of the scalar central solutions on coarse meshes is almost as wide as with the first order upwind scheme.

With the FDS TVD scheme, either the limiter or the upwind extrapolation have an erratic effect on all meshes except the reference mesh #6. Note that there is no total pressure overshoot mesh #4, on which the more dissipative minmod limiter had to be used, Figure 4.24d. There is, however, no similar beneficial effect on the accuracy of the temperature field, where gradients are over-accentuated and as a consequence wall temperatures severely underestimated. Not even on mesh #6 with 54 cells in the boundary layer is a mesh independent solution obtained with the FDS TVD scheme, cf. the dashed line in Figure 4.21. With the same limiter and a comparable effective ratio, the symmetric TVD scheme does not produce unphysical overshoots on any meshes.

The scalar central scheme, while clearly more diffusive than the matrix schemes, has the remarkable property of giving a usable prediction of wall temperature on all meshes except the very coarsest one. With the matrix central and symmetric TVD schemes, a minimum of 16 optimally or 25 sub-optimally distributed grid points in the boundary layer are required. In terms of sharpness of profiles, on the other hand, the scalar central scheme requires about twice the number of points (at least 26) of the corresponding matrix schemes, for which 10 to 16 cells ensure reasonable mesh independence.

It has been argued, for example by Dervieux (1996, at a presentation given at the VUB), that a first order scheme can be made to appear second order accurate globally, if it is used in conjunction with adaptive refinement. The present numerical experiments would indicate that the success of such an approach is limited to situations where captured shocks are the only areas of appreciable flow gradients. Viscous, and more generally, any non-uniform flows require at least a second order accurate discretization of the convective terms.

In an attempt to explain the poor accuracy of the FDS TVD scheme, the leading terms in the truncation error have been compared for the (unlimited) $\kappa$-family of upwind schemes and for the central scheme with added fourth order artificial dissipation with a coefficient $\kappa^{(4)} = 0.1$, Figure 4.20. For the central scheme, we have also considered the possibility of obtaining a third order scheme by
adding a suitable central third difference to cancel the leading error term. For the fully upwind scheme (κ = –1), the coefficients of the error terms are found to be tremendously larger than for the standard central scheme, by factors of approximately 2, 5, 13, and again 5 for respectively the \( \Delta x^2 \partial_x^3 \), \( \Delta x^3 \partial_x^4 \), \( \Delta x^4 \partial_x^5 \), and \( \Delta x^5 \partial_x^6 \) terms. Depending on the flow gradients, the sign of the leading error term, which is opposite for the fully upwind and central scheme, may also play a role. Hence, from this analysis one would indeed expect a somewhat reduced accuracy with the second order fully upwind discretization. Note the increase by a factor 4 of the \( \Delta x^4 \partial_x^6 \) coefficient for the third order-corrected central scheme.

### 4.6.9 Conclusions

From the above discussion, we retain the following conclusions.

For the examined laminar shear flow at high Reynolds number, Matrix dissipation gives significantly higher accuracy than scalar dissipation, which suffers from excessive eigenvalue scaling of the linear field in the wall-normal direction.

The symmetric TVD scheme would appear as an excellent choice: of the tested schemes, it offers the best monotonicity properties, at the cost of minimally reduced accuracy compared with the matrix central scheme.

Schemes which involve abrupt switching of interpolation stencils should be avoided. Concerning the tested second order fully upwind scheme, we observe that for mesh-aligned flow, switching of the interpolation stencil in the cross-flow (wall-normal) direction triggered by tiny fluctuations of the transverse velocity component is indeed undesirable. Considerations of this kind should also flow into the design of multi-dimensional upwind schemes, as considered for example by Van Ransbeek (1998), which are unlikely to be successful in an industrial solver unless they incorporate a smooth, continuous link between the interpolation stencil and the selected waves.

First order (upwind) solutions are unacceptable, even on highly refined meshes.

It is recognized that the preceding analysis is only one small contribution towards identifying a suitable spatial discretization scheme for turbomachinery applications. No two-parameter scheme (e.g., CUSP, AUSM) has been included, and many interesting questions raised have not been answered. Given its already superior performance, it might, for example, be interesting to investigate for the symmetric TVD scheme the effect of different limiters and effective ratios, in order to identify one combination suitable as a default value for industrial users. Such a study had already been undertaken by Zhu and Lacor (1993), focussing however on shock smearing and shock induced oscillations, the target application being hypersonic re-entry flows. In turbomachinery applications, the here examined viscous accuracy with limited mesh resolution is a more pressing concern. Also, given the fundamental similarity of the symmetric TVD and matrix central schemes, it would be most interesting to explore the effect of additional switches, based for instance on temperature or the velocity vector, and not necessarily all associated with the same value of \( \kappa^{(2)} \). In fact, in the standard implementation and in 3D, there are five independent switches in the limited TVD schemes (one for each characteristic variable), but only one in the matrix central scheme.

### 4.6.10 Related studies in the literature

Similar conclusions have been reached by Allmaras (1993) and by Swanson et al. (1998), without, however, paying attention to the temperature field.

With explicitly added scalar central background artificial dissipation, endeavours are sometimes made to reduce this by selecting small values of the dissipation coefficient, for instance by Kallinderis and McMorris (1995) who dynamically couple it to the solution, arguing that once a certain level of convergence has been achieved, less dissipation is required to stabilize the scheme, and a gain in accuracy is indeed demonstrated when it is reduced. So far, however, published techniques of this kind have been limited to simple 2D test cases such as the flat plate and isolated airfoils and it may be feared that their success in an industrial environment would be modest and switching to a matrix-type dissipation should be the preferred first measure.
Jorgenson and Turkel (1993) show how through a modified pressure switch the classical JST central scheme can be made TVD. The TVD theory is, however, based on the 1D scalar case, and, as has been pointed out, the pressure switch is wholly ineffective in suppressing unphysical overshoots caused by transverse viscous gradients. In the context of matrix dissipation, these authors, considering only the Euler equations and hence concerned with contact discontinuities, therefore propose a temperature based switch for the linear field. A lower limit is enforced on the absolute eigenvalues (an entropy fix, in upwind terminology) to prevent vanishing dissipation around sonic and stagnation points.

Figure 4.14 Developing laminar boundary layer in a channel: test case definition.

Figure 4.15 Local view of tested meshes for laminar boundary layer. Mesh ID number, total number of cells in the wall normal direction, and approximate boundary layer thicknesses.
Figure 4.16 Developing laminar boundary layer along channel wall: convergence histories of the density residual on mesh #8 (blue = first order upwind, red = central scalar, green = central matrix, magenta = symmetric TVD, cyan = FDS TVD with limiter, black = FDS TVD without limiter).

Figure 4.17 Developing laminar boundary layer along channel wall, convergence histories of the density residual: (a) scalar central scheme ($\kappa^{(4)} = 0.1$), (b) symmetric TVD scheme (van Leer limiter) (blue = mesh #2, red = mesh #3, green = mesh #4, magenta = mesh #5, cyan = mesh #6, black = mesh #8, grey = mesh #81).
Figure 4.18  Developing laminar boundary layer along channel wall, convergence histories of the density residual with the first order upwind scheme (blue = mesh #2, red = mesh #3, green = mesh #4, magenta = mesh #5, cyan = mesh #6, black = mesh #8, grey = mesh #81).

Figure 4.19  Blasius solution of the incompressible laminar flat plate: profile of total temperature in the case of an adiabatic wall for five values of the Prandtl number.
Figure 4.20 Coefficients of the dissipation and leading error terms for the $\kappa$-family of upwind schemes (bold line) and for the central scheme with fourth order artificial dissipation ($\kappa(4) = 0.1$) with (light dashed line) and without third order correction (light solid line): (a) $\Delta x^2 \partial_x^3$ term, (b) $\Delta x^3 \partial_x^4$ term, (c) $\Delta x^4 \partial_x^5$ term, (d) $\Delta x^5 \partial_x^6$ term.

Figure 4.21 Developing laminar boundary layer along channel wall: comparison of spatial discretization schemes on mesh #6: (a) total temperature, (b) total pressure.
Figure 4.22  Developing laminar boundary layer along channel wall: comparison of spatial discretization schemes on mesh #81: (a) streamwise velocity, (b) normal velocity, (c) static temperature, (d) total temperature, (e) total pressure.
Figure 4.23  Developing laminar boundary layer along adiabatic channel wall: static temperature profiles on different meshes: (a) first order upwind scheme, (b) scalar central scheme ($\kappa^{(4)} = 0.1$), (c) matrix central scheme ($\kappa^{(4)} = 0.33$), (d) FDS TVD scheme (van Leer), (e) symmetric TVD scheme (van Leer).
Figure 4.24   Developing laminar boundary layer along adiabatic channel wall: total pressure profiles on different meshes: (a) first order upwind scheme, (b) scalar central scheme ($\kappa^{(4)} = 0.1$), (c) matrix central scheme ($\kappa^{(4)} = 0.33$), (d) FDS TVD scheme (van Leer), (e) symmetric TVD scheme (van Leer).
4.7 A Matrix Dissipation for Incompressible Flow

The preceding analysis of the accuracy of the various spatial discretization schemes for a simple laminar boundary layer has pointed to some shortcomings of the simple scalar dissipation models commonly used in industrial CFD codes, one particular weakness being in the prediction of frictional dissipation and resulting temperature profiles. One focus of this dissertation is on the flows of cryogenic liquids in space propulsion components. Owing to the extreme temperature gradients and slim cavitation margins, these flows can be expected to be highly sensitive to accuracy of the calculated temperature field. A similar situation is encountered in a large assortment of industrial heat transfer applications, which often involve fluids in the liquid state, one reason being the high volume-specific heat capacity (thinking, e.g., of cooling liquids). In many cases, it is not mandatory to take into account the small compressibility of the liquid, allowing incompressible calculations to be performed.

4.7.1 Mathematical formulation

A matrix dissipation for the system of preconditioned compressible Navier-Stokes equations has been developed by Hakimi (1997) and implemented in the flow solver Euranus. The applicability of that formulation is, however, limited to ideal gases. In view of the urgent industrial need for highly accurate heat transfer predictions, an analogous dissipation model for strictly incompressible fluids has therefore been developed in the present work and optimized for computational efficiency. It is presented in the following.

We take as a starting point the conservative forms of the mass, momentum and energy equations,

\[ \partial_t \rho + \nabla \cdot (\rho \mathbf{v}) = 0 \quad (4.23) \]
\[ \partial_t (\rho \mathbf{v}) + \nabla \cdot (\rho \mathbf{v} \otimes \mathbf{v}) + \nabla p - \nabla \cdot \mathbf{t} = 0 \quad (4.24) \]
\[ \partial_t (\rho E) + \nabla \cdot (\rho \mathbf{v} E) + \nabla \cdot (\mathbf{v} p) - \nabla \cdot (k \nabla T) - \nabla \cdot (\mathbf{t} \cdot \mathbf{v}) = 0 \quad (4.25) \]

and recall that they describe the motion of any type of fluid, including a strictly incompressible one. For an incompressible fluid, the mass equation reduces to the condition \( \nabla \cdot \mathbf{v} = 0 \), and there is a one-way coupling between the momentum and energy equations: the energy equation depends on the velocity field, while the momentum equation is independent of the internal energy, except for the possible temperature dependence of the molecular viscosity. Combining equations (4.24) and (4.25), and exploiting the identities

\[ \mathbf{v} \cdot (\mathbf{v} \otimes \mathbf{v}) = \mathbf{v} (\mathbf{v} \cdot \mathbf{v}) + (\mathbf{v} \cdot \nabla \mathbf{v}) \mathbf{v} \quad (4.26) \]
\[ \mathbf{v} \cdot \frac{\nabla \mathbf{v}^2}{2} = \mathbf{v} \cdot (\mathbf{v} \cdot \nabla \mathbf{v}) \mathbf{v} \quad (4.27) \]

yields a simple convection-diffusion equation for the internal energy

\[ \partial_t e + \mathbf{v} \cdot \nabla e - \frac{1}{\rho} \nabla \cdot (k \nabla T) = \frac{1}{\rho} \mathbf{v} \cdot \mathbf{v} \quad (4.28) \]

where the dissipation function

\[ \varepsilon_v = \mathbf{v} \cdot (\mathbf{t} \cdot \mathbf{v}) - \mathbf{v} \cdot (\nabla \cdot \mathbf{t}) = (\mathbf{t} \cdot \mathbf{v}) \cdot \mathbf{v} = \mathbf{v} \cdot \nabla \mathbf{v} = \frac{1}{2 \mu} \frac{\mathbf{v}^2}{\mathbf{v}} \quad (4.29) \]

appears as a source term. The temperature field is thus independent of the pressure.

Because of the mentioned one-way coupling between the energy equation and the mass and momentum equations, the solution of the latter two can be considered independently of the energy...
Mesh Dependence Analysis

For this system of equations, we propose a simple, artificial compressibility-like preconditioning, replacing the time derivative of density in the mass equation by that of pressure and keeping the original conservative momentum equation. The system to be solved is

$$\Gamma^{-1} \partial_t Q = \partial_j (F_j^{\text{inv}} - F_j^{\text{vis}})$$

(4.30)

with summation over \( j = x, y, z \) and the usual inviscid and viscous fluxes. The inviscid jacobian matrix \( A \) with respect to a given direction \( \hat{n} = (n_x, n_y, n_z)^T \) is defined as

$$A = n_j \frac{\partial F_j^{\text{inv}}}{\partial Q}$$

(4.31)

For the energy equation, two alternatives are proposed. The simplest possibility is to proceed as with the momentum equation and keep essentially the original conservative energy equation. Given the structure of the energy residual, only one small change suggests itself, which is to solve for \( \rho H \) instead of \( \rho E \). There is then agreement between the convective and time dependent terms. As a consequence, the matrix artificial dissipation of the energy equation is simplified and reduced. For this choice, which should be suitable for most Newtonian liquids, the preconditioning matrix is the following diagonal matrix

$$\Gamma = \begin{bmatrix}
\beta^2 & 0 & \cdots & 0 \\
0 & \frac{1}{\rho} & \cdots & 0 \\
\cdots & \ddots & \ddots & \cdots \\
0 & \cdots & \frac{1}{\rho} & \cdots \\
\cdots & \cdots & \cdots & \frac{1}{\rho}
\end{bmatrix}$$

(4.32)

and the vector of dependent variables is

$$Q = (p, u, v, w, H)^T$$

(4.33)

The eigenvalues of the jacobian matrix \( \Gamma A \) are

$$\lambda_{1, 2, 3} = \hat{\mathbf{v}} \cdot \hat{n} = \lambda^0$$

$$\lambda_{4, 5} = \hat{\mathbf{v}} \cdot \hat{n} \pm \sqrt{\beta^2 \hat{n}^2 + (\hat{\mathbf{v}} \cdot \hat{n})^2} = \lambda^\pm = \lambda^0 \pm \sigma$$

(4.34)

where we have defined

$$\sigma = \sqrt{\beta^2 \hat{n}^2 + \lambda^0}$$

(4.35)

After calculation of the eigenvectors, the dissipation matrix is obtained as
Mesh Dependence Analysis

with the definitions

\[ R|\Lambda|R^{-1} = \frac{1}{\sigma^2} \begin{bmatrix}
\beta^2 \hat{n}^2 \sigma & n_x c_m & n_y c_m & n_z c_m \\
\beta^2 \hat{n}^2 \sigma & n_x d_x + \sigma^2 |\lambda^0| & n_y d_y & n_z d_z \\
\beta^2 \hat{n}^2 \sigma & n_x d_y & n_y d_y + \sigma^2 |\lambda^0| & n_z d_y \\
\beta^2 \hat{n}^2 \sigma & n_x d_z & n_y d_z & n_z d_z + \sigma^2 |\lambda^0| \\
\end{bmatrix} \cdot \] (4.36)

with the definitions

\[ c_c = H(2 \sigma - |\lambda^0|) \quad \quad \quad \quad \quad \quad d_x = c_x (\sigma - |\lambda^0|) + u \sigma \lambda^0 \quad \text{etc.} \]
\[ c_m = \rho \beta^2 \lambda^0 \sigma \quad \quad \quad \quad \quad \quad c_x = \beta^2 n_x + u \lambda^0 \quad \text{etc.} \] (4.37)

\[ b_x = -\frac{1}{\rho} \frac{\sigma^2}{\hat{n}^2} u (\sigma - |\lambda^0|) + n_x \lambda^0 |\lambda^0| \quad \text{etc.} \]

In the analogous definition for \( c_y \), for example, \( n_y \) is to be substituted for \( n_x \) and \( v \) for \( u \). The corresponding 2D matrix is obtained by dropping the fourth line and the fourth column. The matrix (4.36) beautifully shows how the dissipation in each equation is composed of contributions from each of the dependent variables. We first observe that the dissipation of the mass and momentum equations is independent of the total enthalpy, as it should be. Considering further the structure of the dissipation matrix, we see that the dissipation for each equation is composed of a principal part from the respective variable and a deviatoric part from the respective other variables. The principal part has the coefficient \( |\lambda^0| \) in the momentum and energy equations and \( \beta^2 \hat{n}^2 / \sigma \) in the mass equation. In the momentum equations, there is also a deviatoric contribution from the respective velocity component. Note that with the exception of the pressure contribution in the energy equation, the coefficients of the deviatoric dissipation terms may be positive as well as negative. The resulting residual contributions must of course be independent of the flow direction, but of opposite sign for the momentum equations if the flow direction is reversed. Both these conditions are easily verified from (4.24) and (4.25), knowing that the sign of even differences is determined by the sign of the variable and independent of the orientation of the index, as illustrated by the centered second differences of \( p \) and \( u \),

\[ \delta^2 p_i = p_{i+1} - 2p_i + p_{i-1} \quad \text{sign independent of } u \] (4.38)
\[ \delta^2 u_i = u_{i+1} - 2u_i + u_{i-1} \quad \text{sign changes with } u \]

For this choice of dependent variables and preconditioning matrix, the artificial dissipation introduced into the temperature field thus depends primarily on differences of the total enthalpy

\[ H = e + \frac{p}{\rho} + \frac{\varepsilon^2}{2} \] (4.39)

instead of temperature or internal energy, with additional direct contributions from the other dependent variables. The second choice that has been retained avoids both these factors which clearly might in certain cases be detrimental to the accuracy of the temperature field. It is described next.

While it was the aim of the first way of solving the energy equation to keep the method as simple, transparent and straightforward as possible, that of the second way, presented in the following, is to eliminate completely any unnecessary influence, in particular of the pressure, and thereby decouple the energy equation from the others also on the dissipation level. The only way to achieve this is to directly solve equation (4.28) for the internal energy or for the temperature. The dependent variables are thus
A conservative residual corresponding to equation (4.28) is constructed through the following preconditioning matrix

\[
\Gamma = \begin{bmatrix}
\beta^2 & \cdot & \cdot & \cdot \\
\cdot & \frac{1}{\rho} & \cdot & \cdot \\
\cdot & \cdot & \frac{1}{\rho} & \cdot \\
\cdot & \cdot & \cdot & \frac{1}{\rho} \\
\frac{\kappa^2}{\rho} - u & -v & -w & \frac{1}{\rho}
\end{bmatrix}
\]

(4.41)

The resulting energy residual can be viewed as a conservative evaluation of the dissipation function \(\varepsilon_v\), at the discrete level. The convective and diffusive terms of equation (4.28) are of course also included, but most of the complexity comes from \(\varepsilon_v\). The eigenvalues of the Jacobian matrix \(\Gamma A\) remain the same and have been given in equation (4.34). To calculate the new dissipation matrix, only those components of the eigenvectors that depend on the energy equation have to be recalculated. The first four lines of the resulting dissipation matrix, corresponding to the mass and momentum equations, are independent of the energy equation and must therefore remain the same, since the preconditioning of that part of the system has not been changed. The full dissipation matrix is nevertheless repeated here for completeness,

\[
R|\Lambda|R^{-1} = \frac{1}{\sigma^2} \begin{bmatrix}
\beta^2 \rho^2 & n_x c_m & n_y c_m & n_z c_m \\
\cdot & n_x d_x + \sigma^2|\lambda^0| & n_y d_x & n_z d_x \\
\cdot & n_x d_y & n_y d_y + \sigma^2|\lambda^0| & n_z d_y \\
\cdot & n_x d_z & n_y d_z & n_z d_z + \sigma^2|\lambda^0| \\
\cdot & \cdot & \cdot & \sigma^2|\lambda^0|
\end{bmatrix}
\]

(4.42)

where the abbreviations have been given above in equation (4.37). Compared with the system defined by equations (4.30) and (4.33), there are two differences concerning the artificial dissipation. First, the dissipation of the energy equation has no direct contributions from the pressure and velocity components anymore, since the corresponding coefficients are all zero. Second, the remaining principal part now depends on differences of the internal energy which has replaced total enthalpy as the dependent variable.

Note that it is not possible to remove the pressure and kinetic energy from a conservative evaluation of the dissipation function \(\varepsilon_v\). Analytically, their influence cancels out exactly. Numerically, this is not the case for two reasons: (i) the discretization error and (ii) the limited number of digits in the floating point arithmetic. If for any reason some deleterious influence cannot be prevented, due to the presence of very strong pressure gradients for example, it is always possible to evaluate \(\varepsilon_v\) directly and efficiently, but non-conservatively, from the last RHS of equation (4.29). In this case, the two stress related terms in the conservative energy residual defined in equation (4.25) are to be replaced by \(\varepsilon_v\) and the total energy by the internal energy. The last line of the preconditioning matrix (4.41) becomes
\[
\Gamma_{\text{energy}} = \begin{bmatrix}
-\varepsilon & \cdots & \frac{1}{\rho}
\end{bmatrix}
\] (4.43)

where the dots signify zero elements. The dissipation matrix (4.42) remains unchanged.

If the energy equation is conservatively coupled to the mass and momentum equations, errors from the other (non-temperature) variables are introduced in one of two forms: either through the artificial dissipation or through the discretization errors contained in the original conservative mass, momentum and energy residuals. The two proposed energy equations represent the two extremes in this regard. Different choices for \( Q \) and \( \Gamma \) can shift the error from one form to the other, but cannot remove it. There might not be a single choice that is optimal for all conceivable test cases.

In practice, it might not be absolutely necessary to include the deviatoric dissipation terms in the first choice of energy equation. While a more robust scheme is certainly obtained if they are included, we mention the possibility of starting the energy equation with some delay, updating the temperature field only after the mass and momentum system has converged one or two orders.

The central scheme with matrix fourth order artificial dissipation gives the following semi-discretized form of equation (4.30),

\[
\frac{dQ}{dt} = -\Gamma \sum_{\text{faces}} \left( \hat{F}_{\text{inv}} - \hat{F}_{\text{vis}} \right) \cdot \hat{n} - \frac{1}{2} \sum_{\text{faces}} R A R^{-1} \varepsilon^{(4)} \delta^3 Q
\] (4.44)

or, alternatively,

\[
\frac{dQ}{dt} = -\Gamma \sum_{\text{faces}} \left[ \left( \hat{F}_{\text{inv}} - \hat{F}_{\text{vis}} \right) \cdot \hat{n} + \frac{1}{2} R A R^{-1} \varepsilon^{(4)} \delta^3 Q \right]
\] (4.45)

where \( \Omega \) is the cell volume, \( \hat{n} \) the surface normal vector of a cell face, \( \varepsilon^{(4)} \) the constant fourth order artificial dissipation coefficient, and were the third differences of the dependent variables are to be taken in the direction of the respective cell face (for example in the i-direction for \( i = \text{cst faces} \)). In the dissipation matrix (4.36) or (4.42), \( \hat{n} = \hat{S} \). The factor 1/2 is introduced for analogy with upwind dissipation. For the first choice of the energy equation, (4.44) and (4.45) are identical, since \( \Gamma \) is constant, equation (4.32).

We add two final remarks:

(I) The matrix dissipation for the energy equation does not mandatorily have to be combined with a matrix dissipation for the mass and momentum equations. It is, however, recommended to implement them together.

(II) If the preconditioning matrix (4.32) is combined with the set of dependent variables

\[
Q = (p, u, v, w, E)^T
\] (4.46)

that is, if one solves for total energy instead of total enthalpy, the deviatoric dissipation coefficients in the energy equation given in (4.36) and (4.37) would change as follows,

<table>
<thead>
<tr>
<th>Dep. var.</th>
<th>Pressure coefficient</th>
<th>Velocity coefficient (( c_e ))</th>
</tr>
</thead>
<tbody>
<tr>
<td>( H )</td>
<td>( \frac{H \alpha^2}{\rho} (\sigma -</td>
<td>\lambda^o</td>
</tr>
<tr>
<td>( E )</td>
<td>( \frac{H \alpha^2}{\rho} (\sigma -</td>
<td>\lambda^o</td>
</tr>
</tbody>
</table>

The present implementation of the preconditioned energy equation is based on the dependent variables (4.46) and, for \( \alpha = -1 \), on the following last line of the preconditioning matrix,
The resulting deviatoric dissipation coefficients in the energy equation are the same for the unpreconditioned energy equation, given in the above table for the dependent variable $E$, except that the total enthalpy $H$ is replaced by $H - e$, that is

$$H \rightarrow \frac{\dot{\gamma}^2 + P}{\rho}$$  \hspace{1cm} (4.48)$$

The dissipation matrix (4.36) and its multiplication by the fourth differences of the dependent variables is programmed in the following compact way, in algorithmic notation:

$$\delta p = \varepsilon^{(4)} \delta^3 p$$
$$\delta \dot{\gamma} = \varepsilon^{(4)} \delta^3 \dot{\gamma}$$
$$\delta E = \varepsilon^{(4)} \delta^3 E$$  \hspace{1cm} (4.49)

$$\lambda^0 = \frac{\dot{\gamma} \cdot \dot{\gamma}}{\sqrt{\beta \dot{\gamma}^2 + \lambda^0}}$$
$$\sigma = \sqrt{\beta \dot{\gamma}^2 + \lambda^0}$$
$$\delta v_n = (\hat{n} \cdot \delta \dot{\gamma})/\sigma$$
$$\delta p = \delta p/\sigma$$
$$|\lambda^0| = |\lambda^0|/\sigma$$  \hspace{1cm} (4.50)

$$c_1 = \frac{\dot{\gamma}^2 (1 - |\lambda^0|)}{\rho}$$
$$c_2 = \lambda^0 (2 - |\lambda^0|)$$
$$c_3 = (\lambda^0 |\lambda^0|)/\rho$$
$$c_4 = \beta (1 - |\lambda^0|)$$
$$\dot{\beta} = c_1 \dot{\gamma} + c_3 \dot{\gamma}$$
$$\dot{\delta} = c_2 \dot{\gamma} + c_4 \dot{\gamma}$$  \hspace{1cm} (4.51)

Energy equation 1  \hspace{1cm} Energy equation 2  \hspace{1cm} Energy equation 3
$$A = \frac{\dot{\gamma}^2 + P}{\rho}$$
$$A = \frac{E + P}{\rho}$$
$$c_1 = c_1 A + c_2 \lambda^0$$
$$c_2 = c_2 A + c_4 \lambda^0$$
$$c_1 = 0$$
$$c_2 = 0$$

$$f_\rho = \beta (\dot{\gamma}^2 \delta p + \rho \lambda^0 \delta v_n)$$
$$\vec{f}_v = \dot{\beta} \delta p + \dot{\delta} \delta v_n + |\lambda^0| \delta \dot{\gamma}$$
$$f_E = c_1 \delta p + c_2 \delta v_n + |\lambda^0| \delta E$$  \hspace{1cm} (4.53)

In equation (4.52), energy equation 1 means the preconditioned energy equation as it is presently implemented, energy equation 2 the unpreconditioned energy equation with $E$ as the dependent variable and energy equation 3 the proposed preconditioned energy equation with $e$ as the dependent variable.
In the latter case, $E$ is to be replaced with $e$ in equations (4.49) and (4.53). The efficiency of the above implementation can be judged from the fact that it only requires 165 operations (62 multiplications, 13 divisions, 70 additions, and 20 subtractions), to be compared with 315 operations for the (admittedly somewhat more complex) original implementation of the compressible (un-)preconditioned matrix dissipation (4 power functions, 147 multiplications, 7 divisions, 104 additions, and 53 subtractions).

### 4.7.2 Evaluation with the laminar flat plate

The advantage of the incompressible matrix dissipation compared with scalar dissipation is illustrated with the laminar flat plate. The Reynolds number based on plate length is $10^5$, the height of the computational domain equals $10\delta$ ($\delta$ = boundary layer thickness at domain exit), the mesh is uniform in the streamwise direction and has an initial cell height on the plate of $0.005\delta$, it is composed of $32 \times 32 = 1024$ cells. This coarse mesh has been deliberately chosen because it is representative of the resolution of viscous regions in complex industrial applications. The preconditioning parameter $\beta$ is 0.5. The fourth order dissipation coefficients $\varepsilon^{(4)}$ are set to their respective recommended values for both cases, 0.1 for the scalar dissipation model and 0.3 for matrix dissipation. No second order dissipation is applied.

Figure 4.25a shows that the convergence rate achieved with matrix dissipation is comparable to that obtained with the more dissipative scalar dissipation, except for the energy equation, which converges at only about 1/4 the rate of the other equations. The calculations use five multigrid levels in a V cycle and a CFL number of 2. The gain in accuracy with matrix dissipation can be appreciated from the detail of the $x$-velocity profile in Figure 4.25b. Compared with the matrix profile, the scalar profile is somewhat smeared and presents the overshoot known to occur with the scalar second order central scheme.

Figure 4.26 compares skin friction distributions for the two incompressible calculations just discussed (Figure 4.26b) and for two equivalent, unpreconditioned compressible calculations at a free stream Mach number of 0.3 (perfect gas with $\gamma = 1.4$, Figure 4.26a). In either case, there is seen to be a distinct advantage in favor of the matrix dissipation. Regarding the difference between left and right figures, it is easily verified that with the choices made for Mach number in the compressible case and for $\beta$ in the incompressible case, the level of dissipation (as determined by the spectral radius of the respective system) is about four times higher in the two compressible calculations than in the corresponding incompressible calculations, hence the poorer accuracy for the compressible case.

Figure 4.26 is thus also an illustration of the insensitivity of matrix dissipation relative to a disparity in the eigenvalues. In the unpreconditioned compressible case a disparity arises at low Mach numbers, in the preconditioned incompressible case for large values of the preconditioning parameter $\beta$. The value of 0.5 chosen for this example is in fact unrealistically low for most industrial applications, such as for example centrifugal pumps, for which 3 would be a more typical value. In terms of eigenvalue structure, this would be equivalent to compressible flow at a moderate Mach number (of the order of 0.2), and had $\beta = 3$ been used in the present example, Figure 4.26b would resemble Figure 4.26a: the scalar accuracy would be severely degraded while the matrix accuracy would remain largely unaffected. On the other hand, it may be found necessary with demanding test cases calling for more artificial damping to slightly raise the matrix dissipation coefficient $\varepsilon^{(4)}$. Conceptually, it is preferable by far to evenly introduce augmented dissipation in this way rather than through excessive values of the preconditioning parameter. If, as experience with a large number of industrial test cases seems to suggest, the robustness of the numerical method is affected by both the eigenvalue structure and the amount of artificial dissipation, matrix dissipation, apart from being intrinsically less dissipative, presents the additional advantage of giving independent control of each and therefore allowing to explore high-$\beta$ regions which would make the scalar scheme either unstable, if too small a value of $\varepsilon^{(4)}$ is chosen, or excessively dissipative (and ultimately again unstable), for normal or too large values of the dissipation coefficient.
Figure 4.25 —PRELIMINARY— Incompressible laminar flat plate, comparison of scalar dissipation ($\varepsilon^{(4)} = 0.1$) and matrix dissipation ($\varepsilon^{(4)} = 0.3$): (a) convergence history (red = scalar, blue = matrix), (b) detail of velocity profiles (bold = matrix, light = scalar).

Figure 4.26 —PRELIMINARY— Laminar flat plate, skin friction coefficient, comparison of scalar dissipation ($\varepsilon^{(4)} = 0.1$, circles) and matrix dissipation ($\varepsilon^{(4)} = 0.3$, squares): (a) compressible (M = 0.3), (b) incompressible ($\beta = 0.5$).
Chapter 5  An Euler Throughflow Model

5.1 Introduction

In recent years, several authors have proposed throughflow models based on the Euler equations, see for instance Spurr (1980), Nigmatullin and Ivanov (1994), Boure and Gillant (1995), Yao and Hirsch (1995), Damle et al. (1997), and Baralon et al. (1997). Other authors report results obtained with Euler throughflow models, without however giving details of the numerical method, for example Vuillez and Petot (1994), who present an application to unsteady flow, and Broichhausen (1994).

To use the Euler equations instead of the traditional streamfunction or streamline curvature methods as the basis for an axisymmetric throughflow model is indeed appealing from several perspectives: the massflow is a result of the computation, choking of blade rows can be predicted, certain types of shocks can be captured, and the existing, well-developed numerical solution techniques for the Euler equations can be applied. If viscous terms are included, end-wall boundary layers can be computed directly.

Additional benefits arise from integration with a 3D Navier-Stokes solver, namely the possibility to set up computational domains comprised of both throughflow and 3D blade rows, and the utilization of the 3D code’s infrastructure. Dawes (1992) has adapted his 3D Navier-Stokes code in this sense, allowing blades with a given geometry to be modelled either as 3D or as throughflow rows.

In the same spirit, a genuine throughflow model for the design and analysis of turbomachines has been implemented in the multiblock, multigrid 3D Navier-Stokes solver Euranus, Hirsch et al. (1991). The presence of the blades in the inviscid axisymmetric flow is modelled in the classical way through a distributed blade force to produce the desired turning, a blockage factor that accounts for the reduced area due to blade thickness, and a distributed friction force representing the entropy increase due to viscous stresses and heat conduction. The exact blade geometry is not required. All features of the 3D code concerning the physical fluid model, boundary conditions, spatial and time discretization, convergence acceleration techniques, and data visualization are available to the through-flow module. This includes the capability to treat the entire range of relevant Mach numbers, from strictly incompressible (through a preconditioning technique) to supersonic, as well as any number of blade rows in any configuration, including, e.g., bypass engines. Selected elements comprising the through-flow model are discussed, with special emphasis on the blade force and its discretization for which two novel techniques have been developed: a multigrid-accelerated time-dependent discretization and a robust formulation of the analysis mode. Each of these is presented in detail.

Despite the numerous proposed Euler throughflow models, the most important questions of what are the shock capturing properties of such a model and which is the most appropriate throughflow representation of supersonic flows and flows with shock waves has not received much attention and a satisfactory answer is still missing. An attempt is therefore made in the present work to remedy this situation. To this end, the shock capturing properties of the axisymmetric Euler throughflow equations in design mode (imposed swirl) and in analysis mode (imposed flow angle) are examined analytically through derivation of the Rankine-Hugoniot relations, starting from the integral form of the Euler throughflow equations. Isolation of the prescribed streamwise distribution of either $W_\theta$ or $\beta$ as the shock-determining factor leads to the construction of a new mode. Because it combines the off-design analysis capability of the analysis mode with the shock capturing properties of the design mode, this mode is termed hybrid. A survey is then made of experimental as well as numerical evidence for the shock structure in the supersonic outboard part of transonic axial compressor rotors. Based on its findings, the practical implications of the different shock capturing properties are discussed. The pitch-averaged representation of a normal shock in a straight 2D blade passage of high stagger angle is compared analytically with the equivalent flow in the two classical throughflow modes, separating the effects of work input (turning) and losses. The relevance of a single captured normal shock is further evaluated by comparison against a combination of an oblique and normal shocks in function of the shock angle.
Finally, the consequences of the different shock representation in the several modes are illustrated for five characteristic operating points covering the complete design speed performance curve of a transonic axial compressor rotor. Circumferentially averaged 3D Navier-Stokes solutions serve as a reference. A comprehensive comparison of the throughflow and averaged 3D flow fields is presented. Conclusions are drawn and guidelines formulated for the throughflow calculation of relative supersonic flow with shock capturing Euler models, specifically how to achieve a realistic representation of the pitch-averaged 3D flow field and the correct amount of work input and shock losses. The analysis mode, due to captured shocks, predicts a wrong flow field inside the blade passage, yet is reasonably accurate globally. The design and hybrid modes, due to identical shock capturing properties, give near-identical solutions, which are in excellent agreement with the pitch-averaged 3D reference solutions.

Properties of the new methodology are further demonstrated on four additional test cases. A four-stage low speed turbine illustrates the efficiency of the multi-grid convergence acceleration for configurations with a large number of blade rows. This test case also serves to demonstrate the feasibility of two options automatically offered by a throughflow model embedded in a 3D Navier-Stokes solver, namely the possibility to combine throughflow and 3D blade rows in a single calculation and the possibility to perform viscous throughflow calculations in which the end-wall boundary layers are resolved.

When attempting to perform Euler throughflow calculations of trans- and supersonic turbines, two types of problems appeared. First, the classical blade modelling parameters do not provide effective control of the throat area, making it impossible to obtain the correct choking mass flow. Second, in the case of high-turning supersonic turbines, the effective area as seen by the throughflow model was varying erratically in the streamwise direction, presenting local maxima and minima, which led to multiple captured shocks and a gross over-estimation of losses. Another innovation of this work, an analysis mode with throat control and smooth, monotonous distribution of the effective blockage, solves both these problems. The efficacy of this new modelling approach is shown for a transonic turbine vane and a supersonic impulse turbine.

The ability of the Euler throughflow model to predict the distribution of a total mass flow onto two or more flow paths is demonstrated with a bypass turbofan engine.

### 5.2 The Euler Throughflow Model

#### 5.2.1 Governing Equations

The dominant aspect of an Euler throughflow method, in particular when trans- and supersonic applications are considered, is modelling of the turning and the associated distributed blade force. Most authors so far have opted for a straightforward algebraic approach, in which the blade force is effectively determined from the tangential projection of the momentum equation in conjunction with the imposed flow direction or swirl, leaving four differential equations to be solved. This can be viewed as spatial discretization of the blade force (more precisely, its θ-component). Alternatively, it may be treated as an additional time-dependent unknown, Baralon et al. (1997), that is, discretized in time. Here, a time dependent approach is presented. In blade passages, the axisymmetric throughflow model solves the following conservative formulation of the Euler equations:
The presence of the blades is modelled through a distributed blade force, \( \mathbf{f}_B \), to produce the desired turning, through a blockage factor, \( b \), that accounts for the reduced area due to blade thickness and a distributed friction force, \( \mathbf{f}_F \), representing the entropy increase due to losses. The exact blade geometry is not required and the method therefore is suitable for design. In ducts, \( b = 1 = \text{cst} \) and \( \mathbf{f}_F = 0 \). In accordance with the basic code, Eq. (5.1) is formulated in Cartesian coordinates and solved in the relative system. The finite volume spatial discretization employs either the central scheme or upwind TVD schemes, time integration is explicit through a Runge-Kutta scheme with implicit residual smoothing and multigrid. The mesh consists of a single cell in the tangential direction and does not reflect the blade geometry; axisymmetry is expressed through a periodic boundary condition.

5.2.2 The Blade Force

The \( \theta \)-component of the blade force is treated as an additional unknown for which Eq. (5.2) is solved. The other two components are obtained from the conditions that the blade force be orthogonal to the relative velocity vector and tangential to the local axisymmetric stream surface. For stability reasons, this second condition is approximated by orthogonality to spanwise mesh lines. \( \tau \) is a pseudo-time and \( \kappa \) a constant coefficient of the order of 1. Variables with an overbar are mean absolute values over the blade passage. The target relative tangential velocity, \( W^t_\theta \), is either direct external input (the so-called design mode) or locally calculated from the prescribed relative flow angle, \( \beta^t \), and the current meridional velocity (the so-called analysis mode),

\[ W^t_\theta = W_m \tan \beta^t \]  

\( W^t_\theta \) (in design mode) or \( \beta^t \) (in analysis mode) are given as explicit input only at the blade trailing edge (TE). The respective distribution inside the blade passage is recalculated at each Runge-Kutta stage by applying a normalized power function between the current value at the blade leading edge (LE), determined by the oncoming flow, and the imposed TE value.

The discretization and multigrid implementation of Eq. (5.2) are crucial for the efficiency of the following global algorithm. \( \rho f_{B\theta} \) is explicitly corrected at each Runge-Kutta stage,

\[ (\rho f_{B\theta})^{(k+1)} - (\rho f_{B\theta})^{(k)} = \kappa \left( \frac{W^t_\theta - W_\theta}{W_\theta} \right) \]  

The multigrid procedure applied to the blade force differs from the standard full approximation scheme (Brandt, 1977; Wesseling, 1991). For the main variables, the first coarse-grid residual is the restricted fine-grid residual. In symbolic notation for two successive grid levels \( n \) and \( n+1 \), \( U \) representing the pseudo-vector of conservative variables, \( L \) the residual operator, \( R \) and \( \tilde{R} \) restriction op-
erators for variables and residuals, \( F \) the forcing function,

\[
\partial_t U^{(n)} + L(U^{(n)}, f_{\theta B}^{(n)}) = F^{(n)}
\]

\[
\partial_t U^{(n+1)} + L(U^{(n+1)}, f_{\theta B}^{(n+1)}) = L(R(U^{(n)}), R(f_{\theta B}^{(n)})) + \hat{R}(F^{(n)} - L(U^{(n)}, f_{\theta B}^{(n)}))
\]  

(5.5)

where the RHS of the second equation becomes the new forcing function \( F^{(n+1)} \). For the blade force, the residual is directly visible on the coarse grid as the difference between the restricted current solution and the restricted target solution, which tends to zero at convergence. Use of a forcing function and the restricted fine-grid residual is therefore not necessary. Eq. (5.2) is applied on the coarse grids unchanged, writing \( K \) for the right-hand-side,

\[
\partial_t (\rho f_{\theta B}^{(n)}) = K(U^{(n)}, f_{\theta B}^{(n)})
\]

\[
\partial_t (\rho f_{\theta B}^{(n+1)}) = K(U^{(n+1)}, f_{\theta B}^{(n+1)})
\]  

(5.6)

Restriction and prolongation applied to the blade force are the same as those for the main variables. If shocks are captured in analysis mode, only linear prolongation, as opposed to the slightly less expensive constant prolongation, permits satisfactory multigrid convergence. The target on grid level \( n+1 \) is obtained by restricting \( W_\theta^t \) from grid level \( n \). In analysis mode, \( \tan \beta^t \) is calculated immediately after restriction,

\[
(\tan \beta^t)^{(n+1)} = \frac{R((W_\theta^t)^{(n)})}{W_m R(W_\theta^t)^{(n)}}
\]  

(5.7)

and then kept constant.

### 5.2.3 The Friction Force

According to the distributed loss model, Hirsch (1989), losses are introduced in the inviscid flow through a friction force \( F_F \) acting in the direction opposite to the relative velocity vector. Its magnitude is proportional to the streamwise derivative of an imposed loss coefficient, \( \partial_m \psi \), and for the perfect gas its magnitude is given by

\[
\rho f_F = \frac{p}{p_{t \text{rot}}} \rho_{t \text{rot}} \partial_m \psi
\]  

(5.8)

This formulation ensures that a loss coefficient of zero will give exactly zero friction force. The imposed loss coefficient \( \psi \) is defined as

\[
\psi = \frac{p_{t \text{rot}} \text{LE} - p_{t \text{rot}}}{p_{t \text{rot}}}
\]  

(5.9)

where \( p_{t \text{rot}} \) is the total pressure associated with rothalpy,

\[
p_{t \text{rot}} = p \left( \frac{T_{t \text{rot}}}{T} \right)^{\gamma - 1} \quad T_{t \text{rot}} = T + \frac{W^2 - U^2}{2c_p}
\]  

(5.10)

and \( p_{t \text{rot}} \) a reference dynamic pressure, taken at the leading edge (for compressors) or trailing edge (for turbines) on each streamwise mesh line.

For output purposes, true streamlines and spanwise profiles of efficiency, pressure ratio and calculated loss coefficient are defined by connecting points of equal massflow, integrated along spanwise mesh lines.
5.2.4 Treatment of the Blockage Factor

The presence of the blockage factor in the fluxes of the Euler throughflow equations suggests the question if accuracy might not be gained by also including it in the artificial or upwind dissipative fluxes. As a representative example we will consider a scalar 1D transport equation with unknown \( u = u(x, t) \) and flux function \( f = f(u) \).

\[
\partial_t u + \frac{1}{b} \partial_x (bf) = 0 \tag{5.11}
\]

The blockage factor \( b = b(x) \) is a smooth function. Equation (5.11) is mathematically equivalent with

\[
\partial_t u + \partial_x f = -\frac{\partial_x b}{b} \tag{5.12}
\]

showing that the effect of blockage could also be included through a source term, although this is not desirable numerically for conservation reasons. No conceptual error is thus made if the artificial or upwind dissipative fluxes do not include \( b \), even though it is the conservative form (5.11) that is discretized. Defining, in a cell-centered framework, a numerical flux

\[
f_{i+1/2}^* = f_{i+1/2}^C - d_{i+1/2} \tag{5.13}
\]

where \( f_{i+1/2}^C \) is a second-order accurate central flux and \( d_{i+1/2} \) an artificial or upwind dissipative flux, Equation (5.11) can be semi-discretized on a uniform mesh either as

\[
\partial_t u + \frac{b_{i+1/2} f_{i+1/2}^C - b_{i-1/2} f_{i-1/2}^C}{b_i \Delta x} - \frac{b_{i+1/2} d_{i+1/2} - b_{i-1/2} d_{i-1/2}}{b_i \Delta x} = 0 \tag{5.14}
\]

or as

\[
\partial_t u + \frac{b_{i+1/2} f_{i+1/2}^C - b_{i-1/2} f_{i-1/2}^C}{b_i \Delta x} - \frac{d_{i+1/2} - d_{i-1/2}}{\Delta x} = 0 \tag{5.15}
\]

The numerical flux of the linearized convection equation discretized with the first-order upwind scheme is

\[
f_{i+1/2}^* = \frac{1}{2} (u_i + u_{i+1}) - \left| a \right| \frac{1}{2} (u_{i+1} - u_i) \tag{5.16}
\]

To retain the property that the numerical flux depend either only on \( u_i \) or on \( u_{i+1} \), the blockage factor obviously must be introduced in both the central and upwind dissipative fluxes, Equation (5.14). Tests with the Euler equations for the first-order upwind scheme and the flux difference splitting TVD scheme (FDSTVD) have shown, however, that the difference between formulation (5.15) and formulation (5.14) is insignificant. It has not been examined if this remains true for very strong blockage gradients and skew meshes. For the FDSTVD scheme, including \( b \) also in the upwind dissipation increases the computing cost by 5%. Similarly, in an effort to compensate the effect of blockage gradients on the artificial dissipation of the central scheme, one might use differences of the variables

\[
(b \rho, b \rho u, b \rho v, b \rho w, b \rho H)^T \tag{5.17}
\]

This is motivated by considerations similar to those leading to the use of \( \rho H \) instead of \( \rho E \). Given the favorable experience with the simpler formulation (5.14) for the upwind schemes, however, this formulation is also used for the central scheme. The differences are thus based on the usual set of variables.
5.3 A Robust Formulation of the Analysis Mode

5.3.1 Classical Model

In nearly all classical throughflow formulations, including those which solve the Euler equations, the magnitude of the blade force is obtained directly from the tangential momentum equation, which is not solved in blade passages, and is therefore defined by the tangential component of the force vector. As a result, throughflow calculations in analysis mode experience convergence problems if high flow angles are imposed ($\beta > 70$ deg). Similarly, temporary backflow in blade passages usually leads to divergence. Responsible for this lack of robustness are (i) the definition of the flow angle and (ii) the mentioned treatment of the blade force in function of its $\theta$-component, be it time dependent or directly coupled to the $\theta$-momentum equation.

With a time dependent formulation of the blade force, all three momentum equations are solved in blade passages, which allows alternative ways of setting the magnitude of the blade force vector. Similarly, using fixed reference directions for the definition of the flow angle and hence the direction of the blade force vector turns out to have a stabilizing effect. In this section, we first examine the reasons for the problems arising with the classical approach. Based on the insight thus gained, we then describe an alternative, robust formulation of the analysis mode.

Definition of the flow angle

Classically, two definitions of the flow angle are available,

$$\tan \beta = \frac{W_\theta}{W_z} \tag{5.18}$$

and

$$\tan \beta = \frac{W_\theta}{W_m} \tag{5.19}$$

where $W_m$ is the unsigned meridional velocity, $W_m = \sqrt{W_r^2 + W_z^2}$.

Direction and magnitude of the blade force

The blade force is normal to the relative velocity vector and to the lean vector, $\vec{L}$. The lean vector is fixed and points in the direction of spanwise mesh lines. Replacing the fixed lean vector by the normal to the stream surface of revolution was found to be unstable, at least for a transonic compressor test case in conjunction with the time dependent blade force. The direction of the blade force is given by the vector

$$\text{dir}(\vec{f}_B) = \vec{W} \times \vec{L} \tag{5.20}$$

It was found more robust to have the blade force normal to the current velocity rather than the target velocity. The target velocity vector differs from the current velocity vector only in the $\theta$-component. For definition (5.18), with $W_{2D} = \sqrt{W_\theta^2 + W_z^2}$,

$$W_\theta^T = W_{2D} \sin \beta^T \tag{5.21}$$

This definition is more robust than the alternative

$$W_\theta^T = W_z \tan \beta^T \tag{5.22}$$

With (5.22) and at high flow angles, small variations in $W_z$ are amplified to large variations in $W_\theta^T$. For definition (5.19), equations (5.21) and (5.22) respectively become

$$W_\theta^T = W \sin \beta^T \tag{5.23}$$
and

\[ W^T_\theta = W_m \tan \beta^T \]  

(5.24)

where again variant (5.23) presumably is more robust, although this has not been tested systematically. The magnitude of the blade force is determined by its \( \theta \)-component which is iteratively adapted until the current flow direction matches the target flow direction,

\[ \vec{f}_B = f_{B\theta} \begin{bmatrix} n_r/n_\theta \\ 1 \\ n_z/n_\theta \end{bmatrix} \]  

(5.25)

where \( n_r, n_\theta, n_z \) are the components of \( \vec{W} \times \vec{L} \). The correction of \( f_{B\theta} \) is proportional to the error in \( W_\theta \),

\[ \Delta(\rho f_{B\theta}) = \kappa [\rho f_{B\theta}] \frac{W^T_\theta - W_\theta}{W_\theta} \cdot \frac{\Delta t}{\Delta t} \]  

(5.26)

**Limits of the model**

The limits of this model are obvious from equations (5.18)–(5.26). First, if \( \beta^T \) approaches 90 deg, formulations (5.22) and (5.24) for \( W^T_\theta \) fail. (Formulations (5.21) and (5.23) remain usable.) Second, if \( \beta \) approaches 90 deg, formulation (5.26) for the blade force fails since \( n_\theta \to 0 \). Third, if \( \vec{W} \) deviates strongly from the streamwise direction, imposing a flow angle according to definition (5.19) makes no sense. All three situations are likely to occur during the convergency transient, entailing either limit cycles or outright divergence of the iterative solution process.

**5.3.2 Robust Model**

**Definition of the flow angle**

It is proposed to use fixed reference directions for the flow angle, instead of the meridional flow direction, equation (5.19). There are four obvious possibilities, suitable for different types of turbomachines:

- The axial direction, equation (5.18).
- The radial direction.
- The direction of streamwise mesh lines.
- The straight-line connection of LE and TE.

The fixed lean vector could then be chosen normal to the fixed reference direction for the blade-to-blade flow angle, although the previous definition which has it point in the direction of streamwise mesh lines remains of course a valid option, since it, too, is independent of the variable velocity vector.

**Direction and magnitude of the blade force**

The singularity at \( \beta = 90 \) deg can be avoided by working with the magnitude of the blade force, \( f_B \), instead of the tangential component, \( f_{B\theta} \). The fixed lean vetor, \( \vec{L} \), and target flow angle with respect to a fixed direction, \( \beta^T \), allow the local definition of a target flow surface with unit normal \( \vec{l}_n \), Figure 5.1.

\[ \vec{l}_n = \frac{\vec{L} \times \vec{l}}{|\vec{L} \times \vec{l}|} \]  

(5.27)
An Euler Throughflow Model

where \( \hat{t} \) is the target flow direction in a plane that contains the reference direction, e.g., the \( \theta,z \)-plane for definition (5.18),

\[
\hat{t} = \begin{bmatrix} 0 \\ \sin \beta^T \\ \cos \beta^T \end{bmatrix}
\] (5.28)

The measure for the error in equation (5.26), \( \Delta W_\theta = W_\theta - W_\theta^T \), is replaced by the deviation of \( \vec{W} \) from the local target flow surface, defined as

\[
\Delta W = -\hat{1}_n \cdot \vec{W}
\] (5.29)

Equations (5.25) and (5.26) are replaced by respectively

\[
\vec{f}_B = f_B \hat{1}_n
\] (5.30)

and

\[
\Delta(\rho f_B) = \kappa(\rho f_B) - \hat{1}_n \cdot \vec{W} \cdot \frac{\Delta t}{\Delta t}
\] (5.31)

**Multigrid**

Concerning multigrid, the local target flow surface on the coarse grids must contain the restricted target velocity vector, \( \vec{W}^T \). The latter is defined as the projection of \( \vec{W} \) onto the local target flow surface,

\[
\vec{W}^T = \vec{W} - (\hat{1}_n \cdot \vec{W}) \hat{1}_n
\] (5.32)

The normal vector \( \hat{1}_n \) is frozen on the coarse grids and given by

\[
\hat{1}_n = \frac{\vec{L} \times \vec{W}^T}{|\vec{L} \times \vec{W}^T|}
\] (5.33)

A problem arises if \( \vec{W}^T \) coincides with \( \hat{t} \) or vanishes, because the normal vector \( \hat{1}_n \) then is undefined. If ever this exceptional situation should turn out to pose a problem, two possible strategies to cope with it would be the following:
• Detect this situation and blend in a restricted $\vec{l}_n$.

• Define the coarse grid local target flow surface through the restricted normal vector and apply a correction to the error measure in equation (5.31). This correction is given by the deviation of the restricted target velocity from the restricted local target flow surface and should scale with the current target velocity,

$$ C = W_{\text{current}}^T (\vec{l}_n \cdot \vec{W}_{\text{restricted}})/W_{\text{restricted}}^T $$

(5.34)

where $\vec{W}_{\text{current}}$ is given by equation (5.32). Equation (5.31) becomes

$$ \Delta(\rho T) = \kappa[\rho T][\vec{l}_n \cdot \vec{W}] \Delta t $$

(5.35)

The second of these two strategies has been tested and found to work as expected. In practice, however, the described situation is does not appear to require any special treatment other than the usual precaution of preventing divisions by zero. The standard algorithm, Eq. (5.31), is therefore applied successfully also on the coarse grid levels, offering the advantages of simple, uniform formulation and slightly better convergence rates.

**Adaptive flow angle at the leading edge**

With the above measures, all potential sources of instability due to the throughflow model have been eliminated, except one: the adaptive flow angle at the leading edge (LE). If for some reason backflow occurs at the blade leading edge, either because of a poor guess for the initial solution or because the chosen blade design and calculated operating point do not allow a steady state solution with attached flow at both end-walls, the resulting large local fluctuations in flow angle have immediate repercussions over the entire blade passage. Moreover, setting the LE flow angle according to flow emanating from the blade passage is inadmissible conceptually, in much the same way as the extrapolation of flow variables at an exit boundary with backflow, and will lead to divergence following the same mechanism.

An option is therefore offered in which the flow angle is imposed through a spanwise profile both at the TE and at the LE, for a specified number of iterations. Computations therefore may consist of two phases: (1) a phase with fixed LE flow angle to establish the global flow pattern and (2) a phase with adaptive LE flow angle to refine the solution accuracy. The adaptive flow angle at the LE is calculated from the current velocity at (or just upstream of) the LE, for instance according to equation (5.18),

$$ \beta_{\text{LE}} = \tan^{-1}\frac{W_{\theta \text{LE}}}{W_{z \text{LE}}} $$

(5.36)

Additionally, the LE is searched for regions with backflow, in which the flow angle is replaced by values interpolated from neighboring regions which do not experience backflow. If the backflow occurs at end-walls, constant extrapolation is employed.

### 5.4 Other Innovations

#### 5.4.1 An Analysis Mode with Throat Control

When the Euler throughflow model as it has been described so far is applied to transonic and supersonic turbines, difficulties of two sorts are encountered.

First, the independent specification of flow angle and tangential blade blockage generally produces a wrong effective throat area, even if the 2D fields inside the blade passage of both these modelling parameters exactly correspond to the equivalent 3D blade geometry. Or, viewed the other way around, a direct translation of a given blade geometry into tangential blade blockage and turning dis-
tributions fails to yield the throat area of the original blade. As a consequence, the choking mass flow of transonic turbines will be wrong initially and tedious or impossible to adjust correctly.

Second, not only may the described independent modelling give the wrong throat area, it may also yield one or more fictitious throats not present in the actual 3D blade passage. Extensive experimentation with transonic compressors and with a supersonic impulse turbine has shown that the Euler throughflow model in analysis mode tends to capture shocks whenever the flow is supersonic and there are non-smooth variations in the effective passage width

\[ b_{\text{eff}} = b \cos \beta \]  \hspace{1cm} (5.37)

seen by the throughflow, leading to (possibly gross) over-estimation of losses and to subsonic flow where in reality the pitch-averaged flow is clearly supersonic.

As an answer to both these problems, a new input mode has been devised in which it is not the blade thickness but the effective blockage \( b_{\text{eff}} \) which is controlled directly. Instead of providing a spanwise profile and location of the maximum tangential blade thickness and distributing this in the streamwise direction as in the standard analysis mode, a spanwise profile and location are provided for the throat width (the effective blockage at the blade throat, \( b_{\text{eff throat}} \)) and it is the effective blockage rather than the the blade thickness which is distributed along streamwise mesh lines, thus guaranteeing a smooth variation. The turning is prescribed in the usual way through specified leading and trailing edge values of the flow angle in combination with a suitable streamwise distribution.

Although the new throat control mode obviously provides an alternative way of defining, indirectly through Eq. (5.37), a blade thickness, care should be taken when interpreting the such obtained values: just as the true tangential blade thickness turned out to be unsuitable for the purposes of throughflow modelling, at least in the cases which are of interest here, the blade thickness implicitly contained in the throat control model cannot be expected to correspond to a reasonable 3D blade shape. When seeking the blade-to-blade profile corresponding to a given throughflow design employing throat control, it would seem preferable to draw it up directly from the \( \beta \) and \( b_{\text{eff}} \) distributions.

In the practical implementation of the analysis mode with throat control, a fixed flow angle distribution is first calculated by combining in the usual way the streamwise power function with the specified values at LE and TE. Note that the leading edge flow angle, \( \beta_{\text{LE}} \), is always required, as opposed to the classical analysis mode for which it is optional. Assuming zero tangential blade blockage, the LE and TE flow angles are converted to the corresponding values of effective blockage:

\[ b_{\text{eff LE}} = \cos \beta_{\text{LE}} \]
\[ b_{\text{eff TE}} = \cos \beta_{\text{TE}} \]  \hspace{1cm} (5.38)

Concerning the effective blockage at the throat, there are two possibilities:

1. There is no throat. This modelling assumption was found appropriate for supersonic impulse turbines. It is selected by giving a negative value of the throat location. \( b_{\text{eff}} \) then varies monotonously between the LE and TE values according to a power function law.

2. There is a throat, in which case a spanwise profile for the corresponding effective blockage, \( b_{\text{eff thr}} \), is given as input to the solver. A two-piece power function between \( b_{\text{eff LE}} \), \( b_{\text{eff thr}} \) and \( b_{\text{eff TE}} \) is applied. Note that \( b_{\text{eff thr}} = t/s \) (\( t = \) throat width, \( s = \) pitch). The throat location is another input parameter, and in the case of classical reaction turbines a realistic value can be derived from the \( (t, s, \beta) \)-triangle at the TE, for instance (\( d = \) normal blade thickness, \( x = \) axial coordinate),

\[ \text{THRLOC} = 1 - \frac{\left( \frac{1}{2} + d_{\text{TE}} \right) \sin \beta_{\text{TE}}}{x_{\text{TE}} - x_{\text{LE}}} \]  \hspace{1cm} (5.39)

Finally, the blockage factor is calculated from the \( \beta \) and \( b_{\text{eff}} \) distributions,
\[ b = \frac{b_{\text{eff}}}{\cos \beta} \] (5.40)

While smoothing of this \( b \) at LE and TE can be added just as in other mode, extensive use of this option is not recommended since it will adversely affect the monotonicity of \( b_{\text{eff}} \). \( \beta_{\text{LE}} \) used in the above procedure is a guess provided by the user. The target distribution finally imposed in the calculation is allowed to adapt to the actual LE value, determined by the incoming flow, over a fraction of the chord called the LE adaptation zone. The adaptation function is quadratic for a smooth link. As in the standard robust analysis mode, adaptation can be prevented by specifying a large number of iterations with a fixed LE flow angle.

The specified profile for the LE flow angle may contain either \( \beta_{\text{LE}} \) directly, or the blade angle at the end of the LE adaptation zone. In the latter case, \( \beta_{\text{LE}} \) is extrapolated by fitting the power function through this value and \( \beta_{\text{TE}} \).

Concerning the technical aspects of the implementation, there is one slight difference between the classical analysis mode and the analysis mode with throat control when the input is processed: the flow angle distribution is applied between the the first inner cell centers at LE and TE in the classical analysis mode, and between the cell corners at LE and TE exactly with throat control.

5.4.2 Flow angle distribution for high-turning turbines

In the robust analysis mode (both classical and with throat control), the streamwise power function distribution of the flow angle may apply to either \( \beta \) or \( \tan(\beta) \). Applying a linear distribution to \( \tan(\beta) \) will always give a realistic blade loading distribution, while a \( \beta \) distribution requires corrective action through the power function exponent to avoid excessive loading at high flow angles.

5.5 Throughflow Shock Capturing Properties

5.5.1 Preliminary Remarks

This section explains the shock capturing properties of the Euler throughflow equations for an isolated 2D blade-to-blade section with relative supersonic inflow. The governing equations and the Rankine-Hugoniot relations are presented for the 2D Euler equations and the associated 1D Euler throughflow equations. Both the design problem and the analysis problem are considered. For clarity, the discussion is limited to a 2D blade-to-blade section, where the friction force and blockage factor are supposed to vary smoothly across the blade passage. Therefore, they do not enter into the Rankine-Hugoniot relations and will not be considered here. The 2D Euler equations are reviewed first for reference and the associated throughflow equations then analyzed by imposing axisymmetry and adding the blade force in the right-hand side.

5.5.2 2D Euler Equations

The Euler equations in integral form can be written as

\[ \int_V \frac{\partial}{\partial t} \begin{bmatrix} \rho \\ \rho \vec{W} \\ \rho E \end{bmatrix} dV + \oint_S \begin{bmatrix} \rho \vec{W} \\ \rho \vec{W} \otimes \vec{W} + p \mathbf{1} \\ \rho \vec{H} \end{bmatrix} \cdot d\mathbf{S} = 0 \] (5.41)

with the relative velocity vector \( \vec{W} = (W_m, W_\theta)^T \). The Rankine-Hugoniot relations for stationary discontinuity surfaces admitted by system (5.41) are
with the unit normal vector of the discontinuity surface \( \hat{1}_n \). If \( \hat{1}_n \) is parallel to the flow direction, \( \hat{1}_n = \hat{W} \frac{1}{|\hat{W}|} \), one has a normal shock, Fig. 5.2a, otherwise an oblique shock, Fig. 5.2b. A special case of an oblique shock is the axisymmetric shock with \( \hat{1}_n = \hat{1}_0 = (1, 0)^T \), Fig. 5.2c. This is the only possible shock in the axisymmetric flow in ducts.

### 5.5.3 Associated Throughflow Equations

With the assumptions of subsection 5.5.2, of all the throughflow terms of Eq. (5.1) only the blade force remains. Writing the components of the momentum equation explicitly, this gives

\[
[r \hat{W}] \cdot \hat{1}_n = 0
\]

\[
[\hat{W}] r \hat{W} \cdot \hat{1}_n + [p] \hat{1}_n = 0
\]

\[
[H] = 0
\]

with the unit normal vector of the discontinuity surface \( \hat{1}_n = (n_m, n_\theta)^T \). If \( \hat{1}_n \) is parallel to the flow direction, \( \hat{1}_n = \hat{W} \frac{1}{|\hat{W}|} \), one has a normal shock, Fig. 5.2a, otherwise an oblique shock, Fig. 5.2b. A special case of an oblique shock is the axisymmetric shock with \( \hat{1}_n = (1, 0)^T \), Fig. 5.2c. This is the only possible shock in the axisymmetric flow in ducts.

#### Method 1: With blade force and both momentum equations

We begin with the former, because it corresponds to our numerical formulation. Both approaches require the definition of the design and analysis problems. The design problem is defined by a smooth streamwise distribution of \( W_\theta \), thus

\[
[W_\theta] = 0
\]
Likewise, the analysis problem is defined by a smooth streamwise distribution of
\[ \beta = \arctan \frac{W_\theta}{W_m} \quad (5.45) \]
thus \( [\beta] = 0 \) and therefore
\[ [W_\theta] = [W_m] \tan \beta \quad (5.46) \]

In both problems the flow is axisymmetric (\( \partial_\theta = 0 \) and \( dS_\theta = 0 \)). The components of the blade force are coupled by the orthogonality condition
\[ \vec{f}_B \cdot \vec{W} = 0 \quad (5.47) \]
The Rankine-Hugoniot relations for stationary discontinuities admitted by system (5.43) with \( dS_\theta = 0 \) are
\[ [\rho W_m] = 0 \quad (5.48a) \]
\[ \rho W_m [W_m] + [p] = F_{Bm} \quad (5.48b) \]
\[ \rho W_m [W_\theta] = F_{B\theta} \quad (5.48c) \]
\[ [H] = 0 \quad (5.48d) \]

\( F_{Bm} \) and \( F_{B\theta} \) are the components of an impulsive blade force, defined by
\[ F_{B\theta} = \lim_{V \to 0} \int_V \rho f_{B\theta} dV \quad (5.49) \]
where the control volume \( V \) encloses the discontinuity, and
\[ F_{Bm} = \frac{W_{\theta1} + W_{\theta2}}{W_{m1} + W_{m2}} F_{B\theta} \quad (5.50) \]
The subscripts 1 and 2 designate the left and right limit values, for example \( [W_m] = W_{m1} - W_{m2} \). Definition (5.50) replaces condition (5.47), which cannot be applied to \( \vec{F} \) because \( \vec{W} \) is undefined at the discontinuity. Equation (5.50) implies an arbitrary but realistic choice on the direction of \( \vec{F} \); it has no influence on the jump relations of the design and analysis problems.

**Design problem**

In the case of the design problem, it follows from Eq. (5.48c) that \( F_{B\theta} = 0 \) and further with Eq. (5.50) that \( F_{Bm} = 0 \). The resulting set of Rankine-Hugoniot relations is the same as that for the 2D Euler equations (5.42) with \( I_n = (1, 0)^T \),
\[ [\rho W_m] = 0 \]
\[ \rho W_m [W_m] + [p] = 0 \]
\[ \rho W_m [W_\theta] = 0 \]
\[ [H] = 0 \]

The design problem therefore captures axisymmetric shocks, which requires \( M_m > 1 \).

**Analysis problem**

In the case of the analysis problem, Eq. (5.48c) with \( W_\theta = W_m \tan \beta \) becomes
\[ \rho W_m [W_m] \tan \beta = F_{B\theta} \quad (5.52) \]
\[ \tan \beta = \frac{W_\theta}{W_m} = 0 \] by definition so that in Eq. (5.50)
(W_{\theta 1} + W_{\theta 2})/(W_{m1} + W_{m2}) = W_{\theta}/W_{m}, \text{ yielding}

F_{Bm} = (-\rho W_{m})[W_{m}] \tan^2 \beta \tag{5.53}

Inserting $F_{Bm}$ in Eq. (5.48b) gives the jump relation for axial momentum. The resulting set of Rankine-Hugoniot relations is the same as that for the 2D Euler equations (5.42) with

\[ \frac{1_n}{n} = (1, \tan \beta)^T/\sqrt{1 + \tan^2 \beta} = \frac{W}{|W|}, \]

\[ [\rho W_m] = 0 \]
\[ (1 + \tan^2 \beta)\rho W_m[W_m] + [p] = 0 \]
\[ [\beta] = 0 \]
\[ [H] = 0 \tag{5.54} \]

The analysis problem therefore captures normal shocks, which requires $M_{rel} > 1$. It should be stressed that the impulsive force $F_B$ arises as a result of captured shocks in the analysis problem.

**Method 2: With blade force and one momentum equation**

**The Design Problem**

If the blade force is eliminated from system (5.43), the following integral form of the design problem is obtained, cf. Appendix A, using the cartesian notation of this appendix,

\[ \int \rho \left. \begin{array}{c} \rho u \\ \rho E \\ \rho u^2 + p \\ \rho Hu \end{array} \right| d\Omega + \int_{S} \left. \begin{array}{c} 0 \\ -\rho v \partial_x v \end{array} \right| dS = \int_\Omega \left. \begin{array}{c} 0 \\ 0 \end{array} \right| d\Omega \tag{5.55} \]

The Rankine-Hugoniot relations for stationary discontinuities admitted by system (5.55) are the same as those for the 2D Euler equations with the particular choice $1_n = (1, 0)^T$.

<table>
<thead>
<tr>
<th>Design Problem</th>
<th>2D Euler Equations</th>
</tr>
</thead>
<tbody>
<tr>
<td>$[\rho u] = 0$</td>
<td>$[\rho u] = 0$</td>
</tr>
<tr>
<td>$[u] \rho u + [p] = 0$</td>
<td>$[u] \rho u + [p] = 0$</td>
</tr>
<tr>
<td>$[v] = 0$</td>
<td>$[H] = 0$</td>
</tr>
<tr>
<td>$[H] = 0$</td>
<td>$[H] = 0$</td>
</tr>
</tbody>
</table>

The design problem thus admits only shocks normal to the x-direction, so-called axisymmetric shocks, Figure 5.2c. Note that the third relation for the 2D Euler equations, $[v] = 0$, is implicitly contained in the design problem, because $v$ is varying smoothly by definition.

**The Analysis Problem**

The analysis problem can be written in integral form

\[ \int \rho \left. \begin{array}{c} \rho u \\ \rho u^2 + p \\ \rho Hu \end{array} \right| d\Omega + \int_{S} \left. \begin{array}{c} 0 \\ -\rho u \partial_x v \end{array} \right| dS = \int_\Omega \left. \begin{array}{c} 0 \\ \frac{a}{1 + a^2}(\rho u^2 + 2p)/(1 + a^2) \partial_x a \end{array} \right| d\Omega \tag{5.57} \]

where

\[ a = \frac{v}{u} = \tan \beta \tag{5.58} \]

The Rankine-Hugoniot relations for stationary discontinuities admitted by system (5.57) are the same as those for the 2D Euler equations with the particular choice $1_n = (1, a)^T/\sqrt{1 + a^2}$, ob-
An Euler Throughflow Model

Assuming the shock unit normal $\hat{1}_n$ is parallel to $\vec{W}$ and setting $v = au$,

$$
\begin{align*}
[\rho u] &= 0 \\
[u][\rho] + [p]/(1 + a^2) &= 0 \\
[H] &= 0
\end{align*}
$$

The analysis problem thus admits only normal shocks, Figure 5.2a. Note that the third relation for the 2D Euler equations, $[v] = a[u]$, is implicitly contained in the analysis problem through equation (5.58), because $a$ is varying smoothly by definition.

5.5.4 The Hybrid Mode

In the preceding section, a distinction has been made between the design and analysis problems, defined by Eqs. (5.44) and (5.46), respectively. Scaling the smooth distributions of respectively $W_\theta$ or $\beta$ to fit the LE value and an imposed TE value of that same variable defines the design and analysis modes. The shock capturing properties, however, are defined solely by the choice of the smoothly distributed variable, which may differ from the imposed TE variable. Other modes can therefore be constructed by combining a smoothly distributed variable with a possibly different imposed TE variable. We define the hybrid mode as that mode which combines the shock capturing properties of the design mode with the off-design analysis capability of the analysis mode. The imposed TE variable $\beta$ is converted to the distributed variable $W_\theta$ according to

$$
W_{\theta\,\text{TE}} = \left|\vec{W}_{\text{TE}}\right| \sin \beta_{\text{TE}}
$$

5.6 Shock Structure in Supersonic Blade Passages

This section attempts to give an overview of the shock structure in supersonic blade passages based on experimental and numerical evidence collected from the literature and to draw conclusions with regard to suitable representations of those shocks in Euler throughflow calculations. A simple control volume model devised by Freeman and Cumpsty (1992) is also discussed, because it provides valuable insight into the mechanisms behind the performance of transonic compressors, although it has the peculiar advantage of not requiring any assumption on the shock structure. Because transonic compressor rotors presently appear to be the most important application of supersonic blade passages, the following survey and subsequent analytical and numerical throughflow applications will focus on them.

5.6.1 General Discussion

A general description of the flow in compressor cascades with relative supersonic inflow can be found, for example, in the text books by Cumpsty (1989) and Lakshminarayana (1995). The shock structure is determined by

- the geometry of the blades
- the inlet Mach number
- the inlet flow direction
- the back-pressure behind the blade row

According to Cumpsty (1989), operation of compressors is normally unchoked with a detached bow shock and a relatively simple shock pattern. The exit flow is mostly subsonic. On the subject of the “unique incidence” condition he remarks that it is the prevalent choke and operating condition only at very high blade speeds ($\mathbf{M}_1 > 1.5$, corresponding to $\mathbf{U} > 450 \text{ m/s}$ at standard sea level condi-
The bow shock is then oblique and attached to the leading edge (or nearly so); choking occurs where the flow is still supersonic, ahead of the enclosed part of the blade passage. Experimentally determined shock patterns are presented for supersonic blade-to-blade sections of three transonic compressor rotors: the NASA rotors 33 and 67, and a high-speed rotor with negative camber.

A comprehensive classification of cascade operation with respect to inlet and exit Mach number and generic shock patterns have been established by Starken (AGARD-AG-328, 1993) and reprinted by Lakshminarayana (1995). A schematic depiction of the flow in a supersonic compressor cascade passage by Lakshminarayana distinguishes a detached bow shock, an oblique passage shock and a series of reflected shocks. The limiting conditions of choking and unique incidence, and the actual operating range, of a supersonic compressor rotor are illustrated through a diagram reprinted from Schreiber and Starken (1984). Their cascade and rotor experiments confirm operation with a detached, near-normal bow shock rather than at the unique incidence for inlet Mach numbers up to 1.2 to 1.5.

Hirsch (1984) reprints two figures from AGARD-LS-83 (1976), one representing a typical flow pattern in a two-dimensional tip section of a transonic compressor fan as determined experimentally, the other showing a possible model for the complex processes of interacting shocks, expansion waves and boundary layers.

Hah and Wennerstrom (1991), Hah and Puterbaugh (1992), Law and Wadia (1993), Wadia and Law (1993), Copenhaver et al. (1993) and Bloch et al. (1996) extensively analysed the flow in low aspect ratio transonic compressor fans, both experimentally and numerically. The cited references provide information on the shock structure at operating points from stall to choke, mostly through casing static pressure contours.

The most accurate information on shock structure is obtained from laser anemometer measurements. Such measurements were performed in transonic compressor rotors by Dunker (DLR Single Stage Compressor) and by Strazisar et al. (NASA Rotor 67, 1989), both documented in Fottner (1990). For both rotors, the shock structure is visualized for different operating points by means of blade-to-blade Mach contours at different spanwise locations.

Plausible and validated assumptions on the shock structure can also be found in the literature on shock loss models. Such models have been constructed with various degrees of complexity, ranging from the first simple models assuming a single normal shock, e.g., the Miller-Lewis-Hartmann model (Miller et al., 1961; see also Wennerstrom, 1989), over more elaborate two-shock models, e.g.,

---

Figure 5.3  Measured shock structure for 15% rotor immersion (from Bloch et al., 1996; originally from Chima and Strazisar, 1983).
by König et al. (1996) to a model by Broichhausen and Gallus (1987) that includes the effect of shock induced boundary layer separation.

The shock structures measured by Chima and Strazisar at near-stall and maximum flow condition can be taken as representative for a supersonic (outboard) section of a typical transonic compressor rotor. They are shown in Figure 5.3, taken from Bloch et al. (1996) and originally from Chima and Strazisar (1983).

5.6.2 Control Volume Analysis

A lucid analysis of supersonic compressor blade performance has been offered by Freeman and Cumpsty (1992), see also Cumpsty (1989). The simple, essentially one-dimensional method is based on the observation that associated with a given set of fluxes of mass, momentum and total enthalpy is a subsonic as well as a supersonic flow. Assuming thin, lightly cambered blades, the authors apply the conservation laws to a control volume around the inlet region of a supersonic blade passage, choosing supersonic inflow and subsonic outflow, for the same fluxes. With only two refinements compared with inviscid flow in a straight, constant-area duct (which would yield the jump relations for a normal shock), namely allowing a small positive incidence angle and taking into account the maximum blade thickness at the exit of the control volume, the model reproduces all the salient features of supersonic blade passages with remarkable fidelity. It predicts the correct level of pressure ratio and loss, as well as their trends with blade speed, incidence, and blade thickness.

Another merit of the article is to clarify the practical significance of the unique incidence condition, which becomes the choking condition only at very high blade speeds, a rough guide being a relative inlet Mach number of 1.5; below, passage choking occurs before the unique incidence angle is reached.

The control volume analysis finally provides an explanation for the extremely rapid rise of the loss coefficient with incidence. This has startled all previous investigators (see, e.g., Prince, 1980, or Kerrebrock, 1981) and is illustrated in Figure 5.4, taken from Freeman and Cumpsty (1992). It compares the loss coefficient in function of incidence as predicted by the control volume method, by a normal shock after a Prandtl-Meyer expansion through the incidence angle, and by the Miller-Lewis-

![Figure 5.4 Loss coefficient versus incidence for uncambered blades of zero thickness at two speeds; from Freeman and Cumpsty (1992).](image-url)
Hartmann (1961) shock loss model. The curves are for blades of zero thickness inclined at 65 deg to the axial direction and drawn for two non-dimensionalized blade speeds

\[ U / \sqrt{c_p T_1} \frac{T_1}{\text{abs}} \]  

(5.61)

0.8 and 0.65, corresponding to inlet relative Mach numbers of approximately 1.45 and 1.16. At zero incidence, the loss is that of a normal shock. The correct (experimentally observed) rate of increase in loss with incidence is that given by the control volume method. The reason why the other two curves fall short is that they do not contain the shock-boundary layer interaction losses. These are naturally included in the control volume method. The authors therefore offer the view that, quoting from Freeman and Cumpsty (1992), “… the traditional method of predicting profile loss in supersonic blades, the addition of a contribution from the shock and another from the blade boundary layers, is not soundly based. The interaction of the shock waves with the boundary layer is an essential feature in creating loss.” A simple linear formula is given as an approximation to the curves in Figure 5.4 for the total loss in the inlet region.

The model also gives most valuable insight into the relation between blade thickness, minimum incidence, and minimum achievable loss. The blade thickness determines the minimum achievable (choking) incidence. Since this cannot be zero, the “shock losses”, including those due to shock-boundary layer interaction, are therefore considerably higher in practice than those associated with a normal shock at the inlet Mach number.

Finally, the model is apt to predict performance features relevant to operation and stage matching, such as the steepening of the surge line and narrowing of the operating range that occur at high speeds. It should be remembered, however, that this control volume analysis is valid only for the straight, thin blades in transonic and supersonic compressor rotors without shock-free compression and with deceleration to subsonic flow.

### 5.6.3 Recent Numerical Evidence

Bloch et al. (1996) are the first to systematically investigate the shock structure in a transonic compressor rotor along the entire characteristic, at design speed, from near-stall to very low exit pressures. Because the high cost of the requisite measurements precludes any such systematic experimental investigation, they do this numerically, solving the three-dimensional Navier-Stokes equations. Experimentally determined rotor shock structures can, without exception, be fit into the pattern determined by Bloch et al., including all lightly cambered transonic rotors referred to in this and the previous section.

The clarification of the behavior between peak efficiency and fully choked flow, down to fairly low back pressures, is of great practical importance, because transonic compressor rotors in actual engines normally operate along the (highly stable) choked part of the characteristic. This is mandated by the requirement for sufficient surge margin (Kerrebrock, 1981) and stage matching considerations at part speed (Freeman and Cumpsty, 1992). Experimentally determined shock structures in this range are sparse. One notable and oft-cited exception (Gliebe, 1977; Prince, 1980; Kerrebrock, 1981; Cumpsty, 1989) are the LV measurements by Wisler (1977). For a detailed description of the different shock structures encountered between stall and very low back pressures we refer to the paper by Bloch et al. Essentially, three qualitatively distinct patterns can be distinguished:

- Near stall, a single normal shock stands ahead of the closed part of the blade passage.
- Around peak efficiency, this first shock turns oblique and is trailed by a normal shock. Initially (i.e., for higher back pressures) these two shocks coalesce near the suction surface.
- For choked flow, the first, oblique shock is not influenced by the downstream flow anymore. The second, normal shock, by analogy with quasi-1D nozzle flow, positions itself in the diverging blade passage to match the imposed back pressure.

The transition between these structures is continuous and smooth. Bloch et al.’s work confirms conclusions for the first, oblique shock in choked operation at
which Prince (1980) arrived through painstaking analysis of various sources of experiential data, namely that the obliquity is approximately that for maximum flow deflection (which happens to be roughly perpendicular to a line through the leading edges), that downstream Mach numbers are near sonic, and that it turns to normal where it impinges on the suction surface.

Bloch et al.’s sequence ends when the normal shock stands in the exit of the covered part of the blade passage. In some cases (see Wisler, 1977, or any of the citing references given above), it may move even farther aft, extending from the pressure surface into the downstream region, and this may be the operating point on the engine working line.

The steep, non-incidence related increase in loss along the choked part of the characteristic is plausibly explained by the over-acceleration of the supersonic flow ahead of the normal shock. This effect may be amplified by boundary layer separation in the widening passage downstream.

From their findings on the shock structure, Bloch et al. derive a simplified 2D shock loss model consisting of an oblique shock, followed by a normal shock if the downstream Mach number is > 1. How realignment of the flow is handled in this model does not become entirely clear from the description given; no realignment seems to have been chosen. One of Bloch’s conclusions is that the shock systems are the primary mechanism for establishing the shape of the efficiency characteristic.

5.6.4 Conclusions

A fairly complete picture of the shock structure as well as the choking and principal loss mechanisms has been painted, especially if combined with Freeman and Cumpsty’s (1992) findings for non-choked flow. Its implications for the throughflow analysis problem are the following:

1. The captured normal throughflow shock mimicks the second, normal shock for choked operation, including the full dynamic range with back pressure variations.

2. The captured shock may also be a reasonable approximation for the transition from the single- to the two shock-structure, although the shock losses may then be overpredicted. The loss correlations must take this into account.

3. Along the practically important choked part of the characteristics, the modelled part of the loss (that complements the captured shock loss), cannot be correlated with incidence anymore. The loss correlations might have to differentiate between choked and non-choked operation.

4. The captured shock may have the correct meridional inclination, including its turning normal at the casing. The importance of these two aspects was pointed out by Puterbaugh et al. (1997).

5. The Mach number and flow angle upstream of the captured shock must be carefully controlled. Without a first deceleration through the oblique shock, there is the risk of excessive over-acceleration. The mean flow angle downstream of the oblique shock may be different from the local blade angle.

6. A model for the first, oblique shock that is blended in as the captured normal shock moves downstream might achieve or facilitate the control admonished in item 5.

In light of the pronounced dynamic achieved through the normal shock alone over nearly the entire choked part of the characteristic, the LE closure model proposed by Boure and Gillant (1995) must be refined.
5.7 The Relevance of Captured Throughflow Shocks

5.7.1 Possible Interpretations and Modelling Approaches

In the preceding sections, an analysis has been performed about the different shock patterns as seen by the design mode, by the analysis mode and in 3D. We briefly state the main results: In the design and hybrid modes, captured shocks are axisymmetric. Figure 5.5c shows such a shock (unrealistically) placed in the context of a blade-to-blade section of a transonic axial compressor rotor. In most of today’s turbomachinery the flow is axially subsonic, however, and the design and hybrid modes then will not capture any shocks. In analysis mode, captured shocks are quasi-normal on axisymmetric streamsurfaces, obliquity being admitted in the meridional plane. They can be thought of as achieving in axisymmetric flow the effect of a blade-to-blade normal shock, Fig. 5.5a or shock B in Fig. 5.5b. Oblique shocks such as shock A in Fig. 5.5b cannot be represented in any of the three throughflow modes.

Having established the shock capturing properties of the Euler throughflow model, it can now be examined how they are best brought to bear with the important application of transonic axial compressor rotors. The supersonic sections of these are characterized by thin blades with low camber, and subsonic outflow. Information on the shock structure is best gleaned from Laser anemometer measurements, e.g. by Dunker (DLR Single-Stage Compressor) or by Strazisar et al. (1989, NASA Rotor 67), both documented in Fottner (1990). A complete picture, consistent with earlier, more punctual experimental results, of the shock structure as well as the choking and principal loss mechanisms is painted in two papers by Freeman and Cumpsty (1992) and by Bloch et al. (1996).

Since the flow remains axially subsonic, no shock can be captured in design and hybrid mode; the imposed or modelled losses must include the shock losses as in the traditional streamline curvature or matrix throughflow codes. Put another way, existing loss models which often are based on large data bases and have been refined over many yeras can be plugged into the Euler throughflow solver without modification.

In theory, a single normal shock as it can be captured in analysis mode might replace or complement a shock loss model. In the ideal case it would obviate the need for a shock loss model thus leading to great conceptual simplicity. As another potential advantage, it might automatically yield the correct shock obliquity in the hub-to-tip plane. With the exception of Wennerstrom’s model (Wennerstrom and Puterbaugh, 1984; Wennerstrom, 1989; Puterbaugh et al., 1997) this is normally neglected in even the most sophisticated shock loss models, e.g., König et al. (1996), constructed from purely 2D considerations in the blade-to-blade plane, but was demonstrated, amongst others by Strazisar (1985) and by Wood et al. (1987), to have decisive influence on shock strength.

Preliminary experience seems to indicate, however, that the structure and upstream Mach number of shocks captured in analysis mode are difficult to control. Although their response to changes in back pressure is similar to Bloch’s scheme in certain cases, an unequivocal association with the actual 3D shocks over the complete design speed operating range remains yet to be established. The structure of the captured throughflow shocks is determined primarily by the streamwise flow angle distribution, somewhat by the geometrical blade blockage, with minor influence from the imposed losses and their streamwise distribution. Flow angle and thickness distributions must conform to the actual ones found in the pitch-averaged solution because they fix the choking massflow. Therefore, no parameter is left to control the shock pattern, which has to be accepted as is and an attempt is made to exclude the associated losses from the imposed ones.

Clear disadvantages associated with a captured normal shock are the facts that (i) the resulting throughflow does not represent the pitch-averaged 2D blade-to-blade flow and (ii) it is not an appropriate model for blade passages in which the relative flow remains supersonic, for instance supersonic free-vortex designs, used in some real-world turbines. The former shortcoming will be quantitatively analysed for one representative example in the next section.

5.7.2 Pitch-Averaged Representation of a Blade-to-Blade Normal Shock

Conceptually, the axisymmetric throughflow solution is supposed to represent the pitch-aver-
age of the corresponding 3D flow. 3D shocks do not appear as discontinuities in the pitch-average flow unless they are axisymmetric. As Fig. 5.5 illustrates, their meridional projection may actually cover a substantial fraction of the axial chord. The flow variation across this shock zone in the pitch-average solution and in the throughflow will now be examined analytically for a blade-to-blade normal shock.

Figure 5.6a shows a normal shock at high flow angle, as it might occur in a transonic rotor. The x-coordinate has been normalized so that the shock impinges on the pressure surface of the upper blade at $x = 0$ and on the suction surface of the lower blade at $x = 1$. Neglecting blade thickness variation, camber and shock stability questions, the flow upstream and downstream of the normal shock is uniform. The tangential average is then composed of three regions: a uniform region for $x < 0$, a smooth transition region for $0 < x < 1$ and another uniform region downstream for $x > 1$.

Figure 5.7 compares the pitch-averaged 2D blade-to-blade flow with possible throughflow representations for $\beta = 60$ deg and $M_1 = 1.4$: (i) a design mode solution with linear swirl ($W_{\theta}$) and loss distributions, (ii) a first analysis mode solution with constant flow angle ($\beta$) and linear loss distributions, (iii) a second analysis mode solution with a captured shock, arbitrarily placed at $x = 0.3$.

The pitch-averaged solution is defined by the area averaged primitive variables. (For this flow configuration, area and mass averaging are the same.) The effect of a captured shock in analysis mode, shown as a light line in Figs. 5.7a-f, is identical to that of the blade-to-blade normal shock. However, the ensuing discontinuity compares poorly with the smooth variation observed in the averaged solution, considering, for instance, Mach number and static pressure ratio, Figs. 5.7a and b. These same figures also show the agreement between the design mode solution (bold dashed line) and the actual average (bold solid line). In the other analysis mode solution, the flow remains supersonic because the equations associated with $\beta = \text{cst}$ and imposed losses describe one-dimensional adiabatic flow at constant area, which evolves along the Fanno line. In the average and in the design mode solution, deceleration to subsonic is enabled by two different mechanisms. In the latter, the flow angle does not remain constant inside the shock zone, Fig. 5.7c, the maximum deflection amounting to 2.8 deg, while in the former there is a considerable overshoot in the loss coefficient to nearly twice the final

---

**Figure 5.5** 2D compressor blade passage with supersonic relative inflow ($M_1 > 1$): (a) near stall, (b) choked, (c) axisymmetric shock admitted in design mode if $M_1 \cos \beta_1 > 1$. 

![Image of a 2D compressor blade passage with supersonic relative inflow](image-url)
Figures 5.7e and f show the total pressure ratio and efficiency that would result if the shock occurred in a rotor with axial inflow. In analysis mode with a captured shock, both are exact upstream and downstream of the shock zone, but discontinuous, while with imposed losses and constant flow angle both are unrealistically low. The design mode, by contrast, agrees perfectly with the averaged solution in terms of pressure ratio, Fig. 5.7e, and gives an even better approximation of efficiency, Fig. 5.7f, because the loss overshoot in the averaged solution causes a corresponding undershoot in efficiency.

The primitive-variable-averaged 2D solution does not satisfy the 1D conservation laws, Fig. 5.6b. A conservative average can, however, be defined (not shown here for clarity), and this would be practically indistinguishable from the design mode solution, including the flow deflection throughout the shock zone.

Figure 5.6  (a) axial extension of 2D blade-to-blade normal shock at high flow angle, (b) massflow relative error of primitive-variable-averaged solution.
5.7.3 Shock Comparison for Two Representative Operating Points

The objective of this section is to assess how well a captured normal shock would approximate an oblique shock, possibly followed by a normal shock, with or without realignment, the idea being to have the captured normal shock represent the effect of the entire shock system. The assumed sequence of the reference shocks against which a single normal shock is compared is that of Bloch et al.’s (1996) proposed shock loss model, except that over-acceleration between the oblique and the normal shock is taken into account.

Based also on these authors’ classification of shock structures in supersonic compressor blade passages, two operating conditions are selected for comparison of the throughflow model (single normal shock at the inlet Mach number) with an idealized 2D blade-to-blade flow (sequence of oblique and normal shocks): (i) mid-range flow with a near-normal detached shock, (ii) choked flow with an attached oblique shock, possibly followed by a normal shock inside the blade passage.

Calculations involve only textbook gas dynamic relations. Even so, the equations and relations used are summarised in Appendix B to clarify the treatment of flow angles, relative system, and efficiency.

Reduced Mass Flow

The survey on shock patterns in section 5.6 indicates that for operating points from mid-range to near-stall a single detached shock occurs shortly in front of the inlet to the closed part of the blade passage, and that this shock can be approximated by a normal shock. This situation is schematically depicted in Figure 5.8a. For the purpose of this discussion, the normal shock is assumed to occur at

Figure 5.7 Pitch-averaged representation of a blade-to-blade normal shock and analytical comparison with throughflow for $\beta = 60$ deg and $M_1 = 1.4$: (a) Mach number, (b) pressure ratio, (c) relative flow angle, (d) loss coefficient, (e) absolute total pressure ratio, (f) isentropic efficiency.
the inlet Mach number, $M_1$, and inlet flow angle, $\beta_1$. The dotted line in Figure 5.8a represents the average flow direction. Figure 5.8d shows the 1D Euler throughflow model associated with Figure 5.8a. The dot between leading edge (LE) and trailing edge (TE) represents a shock as admitted by the throughflow equations.

**Design problem**
Figure 5.8c shows the representation in the blade-to-blade plane of an axisymmetric, or axis-normal, shock. This is the only type of shock admitted by the design problem. Such a shock could be captured for axially supersonic flow, where the axial Mach number is given by

$$M_{1x} = \frac{u}{c} = M_1 \cos \beta_1$$ \hspace{1cm} (5.62)

The shock relations would be those for an oblique shock with a shock angle

$$\theta = 90 \text{ deg} - \beta_1$$ \hspace{1cm} (5.63)

In praxis, however, the axial Mach number always remains subsonic for contemporary transonic compressor designs. Typical values for a supersonic blade-to-blade section near the blade tip are

$$M_1 = 1.4 \quad \beta_1 = 60 \text{ deg} \quad \rightarrow \quad M_{1x} = 0.7$$ \hspace{1cm} (5.64)

The design problem will therefore not capture any shock. As a consequence, the shock losses must be imposed.

**Analysis problem**
The normal shock can be exactly represented by the analysis problem. It will appear as a discontinuity somewhere inside the blade passage, as suggested by the dot in Figure 5.8d. Since the shock relations of the analysis problem are identical to those for the normal shock in the 2D blade-to-blade plane, an identical downstream flow state will result for the same upstream conditions. The difficulty lies in assuring that the throughflow normal shock occurs at the correct Mach number. The effect of near-isentropic acceleration or precompression induced by the suction side camber of the blades could be modelled in the analysis problem by appropriate turning.

**Choked Flow**
For higher flow rates and choked flow, the first passage shock is oblique (A in Figure 5.8b) and may be followed by a second normal passage shock (B).

**Design problem**
Because only axial Mach numbers < 1 are considered, no shock will be captured in a design computation. The discussion for the reduced flow rate applies unchanged.

**Analysis problem**
Since the analysis problem captures normals shocks, we will first discuss the question if the 2D two-shock configuration should be represented by a sequence of two normal shocks in the throughflow model, as suggested by Figure 5.8e. There may be a second passage shock either if the downstream Mach number of the first shock is > 1 or if the flow reaccelerates to supersonic. Note that the required throat need not be a geometrical one, but can be formed by growing or separating boundary layers in the 2D blade-to-blade flow, or by the viscous losses in the 1D throughflow. Because of the small area variations in typical supersonic compressor blade passages, reacceleration to supersonic flow is realistic only for downstream Mach numbers of the first shock slightly less than 1. Owing to shock obliquity, this may in 2D be satisfied even for high inlet Mach numbers $M_1$. For the throughflow normal shock, however, high $M_1$ necessarily means low $M_2$. Sufficient reacceleration is then implausible. Even if it did occur, the resulting sequence of two normal shocks would most certainly lead to overprediction of the shock loss.
Practically, capturing two normal shocks in the analysis problem would require delicate tuning (or very accurate semi-empirical modelling) of blade thickness, turning and viscous losses, which together determine the effective area available to the flow. (The 1D analysis problem is equivalent to generalized quasi-one-dimensional flow; the latter is discussed in detail by Zucrow and Hoffman, 1976.)

It seems therefore more appropriate to consider only a single normal throughflow shock, thought to represent the effect of the entire 2D shock system. This assumption is corroborated by the control volume analysis outlined in section 5.6.2. We shall thus compare a single normal shock (the throughflow shock, TF) to

- an oblique shock at the same Mach number and
- an oblique shock, followed by a normal shock at the downstream Mach number of the oblique shock if this is > 1,

both labelled 2D. The equations used to calculate the different variables and ratios are given in an appendix.

**Oblique shock.** The effect of shock obliquity, expressed through the shock angle

$$\eta = 90 \deg - \theta$$

see Equation (1.61) and Figures 3b and 5.8b, $\eta = 0$ corresponding to a normal shock, is a reduction of the normal Mach number, $M_{1n}$, and thus shock strength, as well as a deflection of the flow. Figure 5.9a shows the ratio of the downstream Mach numbers,

$$\frac{M_{2 \text{ TF}}}{M_{2 \text{ 2D}}}$$

in function of $\eta$ for four different upstream mach numbers, $M_1$. The curve for $M_1 = 1.1$ ends at $\eta = 24$ deg, above which the normal Mach number, $M_{1n} = M_1 \cos \eta$, becomes $< 1$. For example,
for $\eta = 20$ deg and $M_1 = 1.5$, the ratio $M_{2\text{ TF}}/M_{2\text{ 2D}}$ is 0.8, corresponding to a relative error of 20%.

Figure 5.9b reveals that the ratio of downstream static pressures,

$$
\frac{p_{2\text{ TF}}}{p_{2\text{ 2D}}} = \frac{\xi_{\text{TF}}}{\xi_{\text{2D}}}
$$

(5.67)

is nearly independent of the upstream Mach number; $\xi = p_2/p_1$ is the static pressure ratio. For example, for $\eta = 20$ deg, the ratio $p_{2\text{ TF}}/p_{2\text{ 2D}}$ is approximately 1.15, corresponding to a relative error of 15%.

Given upstream absolute and relative flow angles, the work input through the shock in terms of absolute total pressure ratio,

$$
\Pi_{t\text{ abs}} = \frac{p_{12\text{ abs}}}{p_{11\text{ abs}}}
$$

(5.68)

can be examined. We assume an outboard blade-to-blade section of a transonic compressor fan with axial absolute inflow and a typical inlet relative flow angle of 60 deg,

$$
\alpha_1 = 0 \text{ deg} \quad \beta_1 = 60 \text{ deg}
$$

(5.69)

Figure 5.9d shows $\Pi_{t\text{ abs}}$ in function of $\eta$ for four different upstream Mach numbers. For $\eta = 0$, the 2D oblique shock (solid line) is actually a normal shock and thus identical with the throughflow normal shock (dotted line at constant level, drawn for reference). The values for $M_1 = 1.1, 1.3, 1.5, 1.7$ are respectively $\Pi_{t\text{ abs}} = 1.18, 1.64, 2.21, 2.87$. The associated loss coefficients,

$$
\psi = \frac{p_{11\text{ abs}} - p_{12\text{ abs}}}{p_{11\text{ abs}} - p_1}
$$

(5.70)

Figure 5.9e, are 0.002, 0.032, 0.097, 0.181 and the isentropic efficiencies,

$$
\eta_{is} = \left(\frac{p_{12\text{ abs}}}{p_{11\text{ abs}}}\right)^{\gamma - 1} - 1
$$

(5.71)

Figure 5.9f, are 0.994, 0.957, 0.906, 0.851.

Work input through the 2D oblique shock happens through two mechanisms: deceleration, which is the only mechanism for the normal throughflow shock, and deflection. For the assumed sense of shock inclination, Figures 3b and 5.8b, which is the one observed experimentally, both tend to increase the absolute tangential velocity and thus $\Pi_{t\text{ abs}}$. The shock polar shows that, as the shock angle $\eta$ is increased from zero, initially the deflection angle $\delta$ grows rapidly, while the shock strength remains almost constant. Therefore, work input through the 2D oblique shock initially rises above that through the normal throughflow shock, Figure 5.9d, at lower losses, Figure 5.9e, and thus higher efficiency, Figure 5.9f. Beyond a certain point, it drops again, due to both reduced shock strength and diminishing deflection angle. For $M_1 = 1.5$, for example, $\Pi_{t\text{ abs TF}} = \Pi_{t\text{ abs 2D}}$ again for $\eta = 19.2$ deg, Figure 5.9d. Then, $\psi = 0.063$ (0.097 for the normal shock), Figure 5.9e, and $\eta_{is} = 0.938$ (0.906), Figure 5.9f.

Figure 5.9e shows the ratio of downstream absolute total pressures,

$$
\frac{p_{12\text{ abs TF}}}{p_{12\text{ abs 2D}}} = \frac{\Pi_{t\text{ abs TF}}}{\Pi_{t\text{ abs 2D}}}
$$

(5.72)

The maximum negative error ($\Pi_{t\text{ abs TF}} < \Pi_{t\text{ abs 2D}}$) occurs for $M_1 = 1.7$ at $\eta = 12.6$ deg: it is 6.5%, with a ratio $\Pi_{t\text{ abs TF}}/\Pi_{t\text{ abs 2D}}$ of 0.935. If the maximum positive error is to be kept below 10%, the shock angle $\eta$ must stay below 19.5 deg, 23.1 deg, 26.6 deg, 30.0 deg for respectively $M_1 = 1.1, 1.3, 1.5, 1.7$. 

\[ \]

\[ \]

\[ \]

\[ \]

\[ \]

\[ \]

\[ \]

\[ \]
**Oblique shock followed by a normal shock.** The effect of an additional normal shock after the 2D oblique shock if \( M_{2\,\text{oblique\,2D}} > 1 \) is to abruptly reverse the trend of the error. Figure 5.14a shows the ratio of downstream Mach numbers, Figure 5.14b the ratio of downstream absolute total pressures. The kink in the curves marks the point where \( M_{2\,\text{oblique\,2D}} = 1 \). For smaller shock angles \( \eta \), \( M_{2\,\text{oblique\,2D}} < 1 \); the curves are then identical to those in Figures 5.9a and 5.9c. For bigger shock angles, the strength of the second, normal shock (B in figure 5.8b) increases as that of the first, oblique shock (A) decreases. In the limit of vanishing strength of shock A, shock B occurs at the original upstream Mach number, \( M_{1} \), and becomes thus identical to the throughflow normal shock. For \( M_{1} = 1.1 \) this happens at \( \eta = 24 \) deg.

The oblique shock as assumed in the preceding analysis deflects the flow. The deflection angle \( \delta \) is plotted in Figure 5.15 in function of the shock angle \( \eta \) for four different upstream Mach numbers. At upstream Mach numbers \( M_{1} = 1.1, 1.3, 1.5, 1.7 \) the maximum deflection \( \delta = 1.5 \) deg, 6.7 deg, 12.1 deg, 17.0 deg occurs at shock angles \( \eta = 13.7 \) deg, 20.6 deg, 23.5 deg, 24.7 deg. Typical supersonic compressor blade passages, however, have only negligible turning, especially in the front part where the oblique shock is located. The required realignment with the original flow direction appears to be isentropic, in the sense that there are no additional shock losses. A more realistic model for the 2D blade-to-blade passage is therefore one that includes this realignment.

In this straight passage model, the shock angle and upstream Mach number determine the entropy rise across the oblique shock. Together with the conservation of mass and energy, this determines the downstream flow state. (Since the flow direction remains constant, the problem is one-dimensional and there is only one velocity component; thus, three variables suffice to describe the flow.)

Each combination of mass flux, enthalpy and loss admits both a supersonic and a subsonic solution. For the supersonic solution, a two-shock model with an additional normal shock can again be constructed. All three possibilities will now be compared to a single normal shock.

**Oblique shock with realignment—subsonic solution.** By the subsonic solution of an oblique shock with realignment we mean the deceleration of 2D supersonic flow to subsonic with no change in the flow direction, conserving mass and energy, and with losses ranging between those for a normal shock for \( \eta = 0 \) to no losses for \( \eta = \eta_{\text{max}} \), the shock angle at which \( M_{1n} = 1 \). The latter case corresponds to shock-free deceleration, as it may be realized (imperfectly) with inverse-cambered blades at a single operating point. The error of approximating this type of flow by a normal shock is small for low upstream Mach numbers (\( M_{1} = 1.1 \)) and similar in magnitude to that made for a simple 2D oblique shock for higher upstream Mach numbers (\( M = 1.7 \)), but of opposite sign. Figures 5.10a, b and c show respectively the ratio of downstream Mach numbers, \( M_{2\,\text{TF}}/M_{2\,\text{2D}} \), the ratio of downstream static pressures, \( p_{2\,\text{TF}}/p_{2\,\text{2D}} \), and the ratio of downstream absolute total pressures, \( p_{2\,\text{abs}\,\text{TF}}/p_{2\,\text{abs}\,\text{2D}} \), the latter as before for \( \alpha_{1} = 0 \) and \( \beta_{1} = 60 \) deg. For \( \eta = 0 \), the 2D flow undergoes a normal shock; all ratios are 1. Increasing shock angle means lower losses and thus more efficient compression in 2D than by the normal throughflow shock. As a consequence, the downstream Mach number is higher after the throughflow normal shock than after the low-loss 2D compression (losses drive both super- and subsonic adiabatic constant-area flow towards Mach 1, as described by the Fanno line), the error at \( \eta = 30 \) deg being around 10% for \( M_{1} = 1.5 \). The static and total pressure ratios are lower for the throughflow normal shock, with errors, at \( \eta = 30 \) deg and \( M_{1} = 1.5 \), of respectively 10% and 12.5%. For the 2D flow, there is a monotonous increase in absolute total pressure ratio and efficiency, Figures 5.10d and f, solid line; the dotted line marks the constant level of the throughflow normal shock for comparison. For instance, at the level of losses associated with an oblique shock at \( \eta = 30 \) deg, efficiency is 94% for \( M_{1} = 1.7 \), compared with only 85% for a normal shock, and even 99.5% for \( M_{1} = 1.3 \), compared with 96%. Beyond \( \eta_{\text{max}}, M_{1n} < 1 \); an oblique shock in the 2D blade-to-blade flow is no longer possible and the efficiency becomes 1.

**Oblique shock with realignment—supersonic solution.** If the supersonic branch is selected, the losses associated with a normal (for \( \eta = 0 \)) or oblique shock (for \( \eta > 0 \)) alone hardly achieve any deceleration; the flow remains supersonic close to the original Mach number, with a small rise in both
static and absolute total pressure. In the 2D blade-to-blade plane it would correspond to an oblique shock, followed by realignment with the original flow direction, and acceleration back to supersonic if the downstream Mach number of the oblique shock is < 1. This type of 2D flow clearly cannot be approximated by a normal shock, as the ratios of downstream Mach numbers, $M_{2\text{TF}}/M_{2\text{2D}}$, downstream static pressures, $\rho_{2\text{TF}}/\rho_{2\text{2D}}$, and downstream absolute total pressures, $\rho_{t2\text{abs TF}}/\rho_{t2\text{abs 2D}}$ in Figures 5.11a, b, and c illustrate. In the limit of vanishing shock strength and thus zero loss, the downstream 2D flow is identical to the upstream flow; the comparison then is that between upstream and downstream states of a normal shock. Compared with a normal shock, the work input is minuscule, Figure 5.11d, although it also increases with increasing upstream Mach number. Efficiency is extremely low, Figure 5.11f, except for vanishing losses close to $\eta_{\text{max}}$.

A better approximation for this type of 2D blade-to-blade flow (assuming it is realizable) would be a normal shock followed by reacceleration to supersonic. The analysis of such a combination will be given in the section after the next.

**Oblique shock with realignment—supersonic solution, followed by a normal shock.** If the realigned supersonic flow downstream of an oblique shock is followed by a normal shock, the normal throughflow shock at the same upstream Mach number as the first, oblique shock becomes a reasonable approximation again. At the extreme where the first, oblique shock in 2D is a normal shock ($\eta = 0$) at $M_1$, the second, normal shock in 2D occurs at a lower Mach number. The Mach number downstream of the throughflow normal shock, occurring at $M_1$, is therefore lower than the corresponding 2D Mach number, and the pressure ratio higher. At the other extreme, where the first, oblique shock in 2D vanishes ($\eta = \eta_{\text{max}}$), the second, normal shock occurs at the original upstream Mach number and is thus identical with the throughflow normal shock. The transition between the two limits is monotonous. The maximum error made in the approximation of this 2D blade-to-blade flow through a single normal shock at the same upstream Mach number occurs at $\eta = 0$. For instance, for $M_1 = 1.5$ one finds a ratio of downstream Mach numbers, $M_{2\text{TF}}/M_{2\text{2D}}$, of 0.925, corresponding to an error of 7.5%, Figure 5.12a, a ratio of downstream static pressures, $\rho_{2\text{TF}}/\rho_{2\text{2D}}$, of 1.088 (8.8%, Figure 5.12b), and a ratio of downstream absolute total pressures, $\rho_{t2\text{abs TF}}/\rho_{t2\text{abs 2D}}$, of approximately 1.105 (10.5%, Figure 5.12c). The errors in all variables grow with increasing $M_1$. The throughflow normal shock overpredicts the work input, Figure 5.12d, but underpredicts the losses, Figure 5.12e, resulting in too favorable efficiency, Figure 5.12f. The solid lines are for the 2D blade-to-blade flow, the dotted lines for the throughflow. For instance, for $M_1 = 1.5$ and $\eta = 0$, the absolute total pressure ratio in 2D would be about 2, whereas a single normal shock gives 2.21, Figure 5.12d. The loss coefficient is 0.14 in 2D, but only 0.1 in the throughflow model, Figure 5.12e. The resulting efficiency as given by the throughflow normal shock is 5 percentage points too optimistic, being 91% instead of 86% for the 2D blade-to-blade flow.

**Oblique shock with realignment—supersonic solution, compared with a normal shock + reacceleration to supersonic.** If the 2D blade-to-blade flow downstream of the first, oblique shock, realigned with the original flow direction, remains or reaccelerates to supersonic, the normal throughflow shock was seen to be an inadequate model. A better throughflow model for what is conceptually one-dimensional supersonic flow with losses is a normal shock followed by reacceleration to supersonic. It will exactly match the 2D blade-to-blade flow if the shock there is also a normal shock ($\eta = 0$). In the simple model considered here, the effect of increasing shock obliquity in 2D is merely a decrease in loss, which becomes zero at $\eta_{\text{max}}$. The loss coefficient of the oblique shock has been compared to that of a normal shock in Figure 5.9e. Deceleration, and thus work input, is only through the losses (which may be interpreted as the result of friction) and therefore (i) very small and (ii) tending to zero as $\eta \rightarrow \eta_{\text{max}}$, Figure 5.13d. The absolute total pressure ratios for $\eta = 0$ range from 1.03 at $M_1 = 1.3$ to 1.11 at $M_1 = 1.7$. The relative error of $\Pi_{t2\text{abs TF}}$ (dotted line at constant level in Figure 5.13d) with respect to $\Pi_{t2\text{abs 2D}}$ (solid line) becomes so big for higher shock inclinations that the throughflow normal shock, despite the subsequent reacceleration, ceases to be a useful approximation. If the relative error in $\Pi_{t2\text{abs}}$ taking the 2D blade-to-blade flow as the reference, is to remain below 20%, $\eta$ must not exceed 5.8 deg, 9.5 deg, 11.1 deg, 11.9 deg for $M_1 = 1.1$, 1.3, 1.5, 1.7; for $\eta = 30$ deg and $M_1 = 1.5$, the error is 347%. The corresponding efficiencies, Figure 5.13f, while reasonable at very
low upstream Mach numbers (89% for $M_1 = 1.1$ and $\eta = 0$), drop rapidly and are of the order of 40–60% for $M_1 = 1.3–1.7$. The lower losses in 2D for truly oblique shocks mean that the downstream Mach number is higher than for the normal throughflow shock, Figure 5.13a, and the static and absolute total pressure ratios lower, Figures 5.13b and c. At a shock inclination of 30 deg and upstream Mach numbers of 1.3, 1.5, and 1.7, for instance, one reads ratios $M_2^{TF}/M_2^{2D}$ of approximately 0.96, 0.93, and 0.9, ratios $p_2^{TF}/p_2^{2D}$ of 1.05, 1.10, and 1.14, and ratios $p_{t2}^{abs\ TF}/p_{t2}^{abs\ 2D}$ of 1.03, 1.05, and 1.07.

5.7.4 Conclusions

The preceding investigation into the shock capturing properties of the Euler throughflow equations, supplemented by a survey of the shock structure and loss mechanisms in supersonic compressor rotors, together with the above quantitative analysis, suggests the following conclusions:

1. The design problem does not capture shocks (for axially subsonic flow). The shock losses must be imposed using some form of shock loss model.

2. For certain applications, in particular supersonic blade passages with subsonic outflow as they occur in transonic compressor rotors, it is possible to approximately reproduce the effect of the actual shock structure by a single normal shock that can be captured by the analysis problem.

3. It is difficult to give a general idea of the error to be expected from a captured normal shock on pressure ratio, losses and work input. In most cases, it should be below 20%, but may also exceed 50%, in particular for the shock loss coefficient and efficiency.

4. A captured normal shock at the inlet Mach number would be compatible with a linear loss correlation derived by Freeman and Cumpsty (1992) from a control volume analysis.

5. Control of the shock upstream Mach number is an essential factor for the usefulness of a captured normal shock.

6. Realistic performance prediction cannot be expected without loss correlations with incidence (and deviation correlations, although this has not been the topic of this investigation).

7. The loss correlation by which a captured normal shock (for the analysis problem) or a modelled shock (for the design problem) has to be complemented must include the effects of shock-boundary layer interaction and subsequent mixing, responsible for the steep increase in loss with incidence.

8. Especially with the analysis problem, it might prove necessary to exploit information on the cascade geometry. For example, flow at the unique incidence could be imposed or the acceleration and precompression due to suction side camber and incidence incorporated into the throughflow model. The leading edge flow angle has, for example, been taken into account by Nigmatullin and Ivanov (1994) and, in connection with unique incidence considerations, by Boure and Gillant (1995).
Figure 5.9 Comparison of 2D (oblique) and throughflow (normal) shock in function of the shock angle $\eta = 90 \text{ deg} - \theta$ for different upstream Mach numbers: (a) ratio of downstream Mach numbers, $M_{2,\text{TF}}/M_{2,\text{2D}}$, (b) ratio of downstream static pressures, $p_{2,\text{TF}}/p_{2,\text{2D}}$, (c) ratio of downstream absolute total pressures, $p_{t2,\text{TF}}/p_{t2,\text{2D}}$, (d) absolute total pressure ratio, $\Pi_{t\text{abs}} = p_{t2,\text{abs}}/p_{t1,\text{abs}}$, (e) loss coefficient, $\psi = (p_{t1} - p_{t2})/(p_{t1} - p_{1})$, (f) isentropic efficiency, $\eta_{\text{is}}$. In (d), (e), (f), solid line = 2D (oblique) shock, dotted line = TF (normal) shock; (c), (d), (f) for inlet flow angle $\beta_1 = 60$ deg.
Figure 5.10  Comparison of 2D (oblique with realignment, subsonic solution) and throughflow (normal) shock in function of the shock angle $\eta = 90 \text{ deg} - \theta$ for different upstream Mach numbers: (a) ratio of downstream Mach numbers, $M_2\text{TF}/M_2\text{2D}$, (b) ratio of downstream static pressures, $p_2\text{TF}/p_2\text{2D}$, (c) ratio of downstream absolute total pressures, $p_\text{t2 abs TF}/p_\text{t2 abs 2D}$, (d) absolute total pressure ratio, $\Pi_\text{t abs} = p_\text{t2 abs}/p_\text{t1 abs}$, (e) loss coefficient, $\psi = (p_{\text{t1}} - p_{\text{t2}})/(p_{\text{t1}} - p_1)$, (f) isentropic efficiency, $\eta_{\text{is}}$. In (d), (e), (f), solid line = 2D (oblique) shock, dotted line = TF (normal) shock; (c), (d), (f) for inlet flow angle $\beta_1 = 60 \text{ deg}$. 
Figure 5.11 Comparison of 2D (oblique with realignment, supersonic solution) and throughflow (normal) shock in function of the shock angle \( \eta = 90 \text{ deg} - \theta \) for different upstream Mach numbers: (a) ratio of downstream Mach numbers, \( M_2^\text{TF}/M_2^\text{2D} \), (b) ratio of downstream static pressures, \( p_2^\text{TF}/p_2^\text{2D} \), (c) ratio of downstream absolute total pressures, \( p_{t,\text{abs}}^\text{TF}/p_{t,\text{abs}}^\text{2D} \), (d) absolute total pressure ratio, \( \Pi_{\text{abs}} = p_{t,\text{abs}}^\text{TF}/p_{t,\text{abs}}^\text{2D} \) (e) loss coefficient, \( \psi = (p_{t,1} - p_{t,2})/(p_{t,1} - p_1) \), (f) isentropic efficiency, \( \eta_{\text{is}} \). In (d), (e), (f), solid line = 2D (oblique) shock, dotted line = TF (normal) shock; (c), (d), (f) for inlet flow angle \( \beta_1 = 60 \) deg.
Figure 5.12  Comparison of 2D (oblique with realignment, supersonic solution + normal) and through-flow (normal) shock in function of the shock angle $\eta = 90 \text{ deg} - \theta$ for different upstream Mach numbers: (a) ratio of downstream Mach numbers, $M_2\text{TF}/M_2\text{2D}$, (b) ratio of downstream static pressures, $p_2\text{TF}/p_2\text{2D}$, (c) ratio of downstream absolute total pressures, $p_2\text{abs}\text{TF}/p_2\text{abs}\text{2D}$, (d) absolute total pressure ratio, $\Pi_t = p_2\text{abs}/p_2\text{abs}$, (e) loss coefficient, $\psi = (p_{t1} - p_{t2})/(p_{t1} - p_1)$, (f) isentropic efficiency, $\eta_{\text{is}}$. In (d), (e), (f), solid line = 2D (oblique) shock, dotted line = TF (normal) shock; (c), (d), (f) for inlet flow angle $\beta_1 =$
Figure 5.13  Comparison of 2D (oblique with realignment, supersonic solution) and throughflow (normal) shock with reacceleration to supersonic in function of the shock angle $\eta = 90 \text{ deg} - \theta$ for different upstream Mach numbers: (a) ratio of downstream Mach numbers, $M_{2,TF}/M_{2,2D}$, (b) ratio of downstream static pressures, $P_{2,TF}/P_{2,2D}$, (c) ratio of downstream absolute total pressures, $P_{t2,abs,TF}/P_{t2,abs,2D}$, (d) absolute total pressure ratio, $\Pi_{2,abs} = P_{t2,abs}/P_{t1,abs}$, (e) loss coefficient, $\psi = (P_{t1} - P_{t2})/(P_{t1} - P_t)$, (f) isentropic efficiency, $\eta_{is}$. In (d), (e), (f), solid line = 2D (oblique) shock, dotted line = TF (normal) shock; (c), (d), (f)
Figure 5.14  Comparison of 2D (oblique + normal) and throughflow (normal) shock in function of the shock angle $\eta = 90\,\text{deg} - \theta$ for different upstream Mach numbers: (a) ratio of downstream Mach numbers, $M_{2\,TF}/M_{2\,2D}$, (b) ratio of downstream absolute total pressures, $p_{t2\,abs\,TF}/p_{t1\,abs\,2D}$. (b) for inlet flow angle $\beta_1 = 60\,\text{deg}$.

Figure 5.15  Flow deflection angle $\delta$ through a 2D oblique shock in function of the shock angle $\eta = 90\,\text{deg} - \theta$ for different upstream Mach numbers.
Chapter 6  Transonic Axial Compressor Rotor

6.1  General Presentation of the Test Case

Figure 6.1  Throughflow blade modelling parameters.

Figure 6.2  NASA Rotor 67: $32 \times 72 = 2304$-cell mesh.
In the preceding chapter, the shock capturing properties of the Euler throughflow equations have been analysed theoretically, with the principal conclusion that the type of captured shock depends on the mode of the calculation. The design mode captures axisymmetric shocks, requiring axially supersonic flow, while the analysis mode captures normal shocks which may occur in relative supersonic flow. With the additional insight that it is not the variable which is held fixed at the trailing edge but the variable for which a smooth streamwise distribution is imposed across the blade passage that determines the type of shock to be captured, a new hybrid mode has been defined which combines the analysis capability of the analysis mode with the shock capturing properties of the design mode.

The (thus far hypothetical) throughflow shocks have been contrasted with the shock patterns actually observed in supersonic blade passages. Possible consequences of captured throughflow shocks on flow structure, work input, and losses have been discussed and a first quantitative assessment has been given at the hand of simplified models.

In this chapter, the theoretical predictions are verified and different modelling options explored in practice with a transonic compressor rotor. The test case that has been selected for this purpose, NASA Rotor 67 (Strazisar et al., 1989; Fottner, 1990), is a typical transonic axial compressor rotor, with the design parameters listed in Table 6.1.

<table>
<thead>
<tr>
<th>Table 6.1: NASA Rotor 67 design parameters.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Absolute total pressure ratio</td>
</tr>
<tr>
<td>Isentropic efficiency</td>
</tr>
<tr>
<td>Inlet tip relative Mach number</td>
</tr>
<tr>
<td>Tip stagger angle</td>
</tr>
<tr>
<td>Number of blades</td>
</tr>
<tr>
<td>Aspect ratio</td>
</tr>
<tr>
<td>Mean hub/tip radius ratio</td>
</tr>
</tbody>
</table>
This test case has been chosen in part because it is exceptionally well documented and representative of transonic compressor rotors used in practice, for example in aircraft jet engines, with a subsonic inboard part and supersonic outboard part, reflecting also the trend towards wide-chord designs. Moreover, a complete set of numerical data in the form of eleven 3D Navier-Stokes solutions covering the design speed performance curve had been obtained previously. Those computations had been performed with the same flow solver which now also contains the throughflow model, on a mesh of circa 550 000 cells, including tip clearance and with a two-equation turbulence model.

The flow path and meridional projection of the blade leading and trailing edges are represented exactly in the throughflow mesh of $32 \times 72 = 2304$ cells, shown in Figure 6.2. Figure 6.1 schematically shows the other blade modelling parameters.

The throughflow results will be presented in two parts, reflecting the chronological order in which they have been obtained. In a first step, attention has been focussed on one operating point only, the aim being to gain experience with transonic throughflow calculations, experiment with the various blade modelling parameters, and assess the sensitivity of transonic throughflow solutions containing captured shocks with regard to the spatial discretization scheme and mesh resolution. The peak efficiency operating point has been chosen for this purpose. The results of this preliminary analysis are presented in section 6.2.

In a second step, capitalizing on the experience accumulated in step 1, the behavior of the Euler throughflow model in the three modes (design, analysis, and hybrid) is examined for five operating points, evenly distributed along the design speed line between nearly stalled and deeply choked. For these selected operating points, the 3D Navier-Stokes solutions are circumferentially averaged and systematically analyzed with regard to the throughflow blade modelling parameters. Drawing on both the lessons from the preliminary study of step 1 and the insight afforded by the pitch-averaged 3D solutions, each of the five selected operating points is then calculated in each of the three modes and the solutions synoptically compared amongst each other and against the respective pitch-average. This comprehensive analysis is presented in section 6.3.

### 6.2 Preliminary Analysis at Peak Efficiency Operation

To get the same results in a design run as in an analysis run, the losses imposed in the design run must be the losses imposed in the analysis run plus the shock losses captured in the analysis run:

$$\psi_{\text{design}}^{\text{imposed}} = \psi_{\text{analysis}}^{\text{imposed}} + \text{captured shock losses}$$  \hfill (6.1)

We first performed an analysis run with a viscous loss profile, shown in Fig. 6.3. The captured shock appears clearly in the isolines of relative Mach number, Fig. 6.4a; the bold isoline marks $M_{\text{rel}} = 1$. The computed trailing edge values for swirl and loss coefficient were then imposed in an otherwise identical design run, Fig. 6.4b. Although—in conformance with theory—no shock is captured, nearly the same massflow, pressure ratio and efficiency are predicted as in the analysis run, table 6.2.

<table>
<thead>
<tr>
<th>Mode</th>
<th>$\dot{m}$ [kg/s]</th>
<th>$\Pi_t$</th>
<th>$\eta_{\text{is}}$ [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>analysis</td>
<td>30.25</td>
<td>1.831</td>
<td>89.87</td>
</tr>
<tr>
<td>design</td>
<td>30.37</td>
<td>1.844</td>
<td>89.51</td>
</tr>
</tbody>
</table>

*Table 6.2: Comparison of analysis mode and design mode.*
Conceptually, solutions calculated with an axisymmetric throughflow method are an approximation to the pitchwise averaged 3D flow. Their accuracy should therefore be assessed by comparison with the latter. 3D Navier-Stokes calculations have been performed with the same flow solver on a mesh of 550,000 cells including tip clearance and using a two-equation turbulence model. Figure 6.5 shows isolines of relative Mach number in the meridional average, to be compared with Figs. 6.4a and b. Globally, the design solution represents the pitch averaged 3D flow field more faithfully than the analysis solution. The 3D solution corresponds to the peak efficiency operating point on the calculated characteristic (\( \dot{m} = 33.65 \, \text{kg/s}, \Pi_t = 1.58, \eta_{is} = 89.0 \% \)). Because the primary goal of the throughflow calculations had been the comparison between design and analysis mode, simple choices have been made for the different parameters that model the blade (e.g., linear distribution of flow angle and losses, standard parabolic thickness distribution). In particular, the choice of a deviation angle of only 5 deg over the entire span constitutes a considerable underestimation. The pressure ratio is therefore too high (\( \Pi_t = 1.83 \)). The throughflow operating point should nonetheless occupy a similar position on the characteristic, shifted to a higher pressure ratio; the massflow of 30.25 kg/s equals 99.15 % of the throughflow choking massflow, versus 98.25 % for the 3D calculation, efficiency is 89.9 % versus 89.0 % in 3D. The 3D calculation determines a slightly oblique shock at the blade passage entrance which appears as a smooth deceleration in the meridional average. This is realized in
design mode, whereas the analysis solution first shows substantial acceleration upstream of the captured normal shock. Only in the subsonic region near the blade root does the analysis solution appear to be of higher fidelity than the design solution: the weaker deceleration (and even reacceleration at the hub), leading to an inflection point in the isolines, is present, albeit overestimated in its severity, whereas the design solution misses this effect. The obliquity of the analysis shock provokes considerable deflection of the meridional streamlines. Figure 6.6a compares analysis mode (solid bold lines) and design mode (dashed bold lines), shown superimposed on streamwise mesh lines. The reference 3D solution, Fig. 6.6b, confirms the smooth design mode streamlines. Despite the difference inside the blade passage, the downstream flow fields are nearly identical, witnessed by the profiles of $\beta$ and absolute total pressure ratio, Figs. 6.7a and b.

An advantage of the analysis mode over the design mode is the ability to run performance curves by mere variation of exit pressure, including the prediction of choke. (Simultaneous variation of exit swirl would be required in design mode to maintain the correct exit flow angle, which is not practicable.) The choking massflow turns out to be highly sensitive to some of the blade modelling parameters. The blade thickness is defined through a spanwise profile of maximum tangential thickness $d_{\text{max}}$, the location of $d_{\text{max}}$ as a fraction of chord, an exponent for the streamwise power function distribution and the number of cells over which to smooth $b$ at LE and TE. The turning is prescribed through a spanwise profile at the trailing edge of $W_{\theta}$ or $\beta$ and an exponent for the streamwise power function distribution. Finally, a spanwise profile at the trailing edge for $\psi$ as well as an exponent for the streamwise power function distribution define the imposed losses.

Table 6.3: Sensitivity of the choking massflow to modelling parameters.

<table>
<thead>
<tr>
<th>Case</th>
<th>Modification</th>
<th>$\dot{m}_{\text{chok}}$ [kg/s]</th>
<th>$\Pi_t$</th>
<th>$\eta_{\text{IS}}$ [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td>30.8</td>
<td>1.53</td>
<td>78</td>
</tr>
<tr>
<td>2</td>
<td>$d_{\text{max}}$ at 0.75 c</td>
<td>32.7</td>
<td>1.52</td>
<td>86</td>
</tr>
<tr>
<td>3</td>
<td>$d_{\text{max}}$ at 0.25 c</td>
<td>27.2</td>
<td>1.53</td>
<td>66</td>
</tr>
<tr>
<td>4</td>
<td>no smoothing</td>
<td>28.9</td>
<td>1.53</td>
<td>72</td>
</tr>
<tr>
<td>5</td>
<td>$\delta - 5 \deg$</td>
<td>32.6</td>
<td>1.63</td>
<td>78</td>
</tr>
<tr>
<td>6</td>
<td>$\delta + 5 \deg$</td>
<td>28.6</td>
<td>1.44</td>
<td>79</td>
</tr>
<tr>
<td>7</td>
<td>$\beta$-exponent 2</td>
<td>34.8</td>
<td>1.55</td>
<td>89</td>
</tr>
<tr>
<td>8</td>
<td>$\beta$-exponent – 2</td>
<td>24.7</td>
<td>1.52</td>
<td>63</td>
</tr>
<tr>
<td>9</td>
<td>$\psi$-exponent – 5</td>
<td>31.0</td>
<td>1.53</td>
<td>79</td>
</tr>
<tr>
<td>10</td>
<td>$\psi$-exponent 5</td>
<td>30.1</td>
<td>1.52</td>
<td>76</td>
</tr>
<tr>
<td>11</td>
<td>$d_{\text{max}}$ – 20 %</td>
<td>32.2</td>
<td>1.54</td>
<td>82</td>
</tr>
<tr>
<td>12</td>
<td>$d_{\text{max}}$ + 20 %</td>
<td>29.3</td>
<td>1.51</td>
<td>75</td>
</tr>
<tr>
<td>13</td>
<td>$d_{\text{max}}$-exponent 1.5</td>
<td>31.8</td>
<td>1.53</td>
<td>81</td>
</tr>
<tr>
<td>14</td>
<td>$d_{\text{max}}$-exponent 3</td>
<td>29.1</td>
<td>1.52</td>
<td>75</td>
</tr>
</tbody>
</table>

The sensitivity of the choking massflow to these parameters has been investigated by applying sensible variations to each in turn, keeping a constant exit pressure of 100 kPa at $r = 14$ cm. The results are summarized in Table 6.3 ($\Pi_t =$ total pressure ratio, $\delta = 5 \deg$ indicates a reduction of the reference deviation angle by 5 deg). In terms of the blade modelling parameters, the reference case, 1, is defined as follows: $d_{\text{max}}$, rather than being deduced exactly from the blade geometry, is assumed to be co-located with the directly available maximum normal blade thickness, $d_{\text{n max}}$, and thus approximated as $d_{\text{max}} = d_{\text{n max}} / \cos \beta_{\text{n max}}$, where $\beta_{\text{n max}}$ is the blade angle at the location of $d_{\text{n max}}$. The streamwise thickness distribution seen by the throughflow computation is quadratic (exponent 2) with $d_{\text{max}}$ at mid-chord (0.5 c) and the resulting $b$ smoothed over $\pm 4$ cells at LE and TE. The imposed trailing edge flow angle and the loss coefficient are those measured at peak efficiency at a station downstream of the blade row. Both are distributed linearly (exponent 1), $\beta_{\text{TE}}$ is corrected by the cal-
culated variation between the measurement station and the TE. Maximum variations of choking massflow and total pressure ratio are highlighted by bold print. The greatest influence on choking massflow is exerted by the flow angle distribution. Figure 6.8 compares the actual distributions, extracted from the meridional average of the 3D solution at 13 %, 52 % and 88 % span from the hub, with the power function distributions assumed in the throughflow module. At the hub and near mid-span the simple power function, with a suitable exponent, fits the actual distributions remarkably well. The tip distribution follows the same general trend of more turning in the forward region, but shows a defect around mid-chord. The location of $d_{\text{max}}$ and the $\beta$-exponent have also been modified either only for the tip-section or only for the hub-section, with a linear variation along the span towards no modification at the respective other end of the blade. The choking massflow turns out to be more sensitive to modification of the tip section, but only slightly so (by 15 % for the $\beta$-exponent and by 9 % for the location of $d_{\text{max}}$).

![Figure 6.6 NASA Rotor 67: streamlines: (a) throughflow (solid bold lines = analysis mode, dashed bold lines = design mode, light lines = streamwise mesh lines), (b) meridional average of 3D Navier-Stokes calculation.](image)

![Figure 6.7 NASA Rotor 67: downstream profiles of (a) relative flow angle and (b) absolute total pressure ratio (solid line = analysis mode, dashed line = design mode).](image)
No control or limiting of the leading edge flow angle has been performed in these calculations. The choking mechanism for all blade sections is thus passage choking, as opposed to choking at the unique incidence angle. At the experimental choking massflow (35.0 kg/s, adjusted with, e.g., $d_{max}$ at 0.6 c, a $d_{max}$-exponent of 1.6 and a $\beta$-exponent of 1.5) the throughflow computation indicates zero incidence ($\pm 0.5$ deg) with respect to the suction side blade angle for the outer 40 % span.

The multigrid algorithm described in section 5.2 is highly effective in the absence of shocks. This is demonstrated by an analysis run at 50 % speed, Fig. 6.9, for which a convergence level of 4 orders, sufficient for practical purposes, is reached 14 times faster (in terms of CPU time) with the multigrid algorithm than on a single grid. If shocks are captured, multigrid still provides an advantage.
both in terms of robustness and convergence rate for all operating points except choked ones. In the latter case, a slight slowdown compared with single grid is observed in terms of CPU time. Single grid then only requires around 300 iterations for engineering accuracy (residual reduction of 5 orders), however. Multi-stage machines might favour multigrid even in those cases.

The computations discussed so far use the second-order central scheme with a Jameson-type blend of fourth- and second-order artificial dissipation (Jameson et al., 1981; Hirsch, 1990), in combination with a four-stage Runge-Kutta scheme and time-step dependent residual smoothing (Zhu, 1996) with a coefficient of 2. The CFL number is 6 and the coefficient $\kappa$ in Eq. (5.4) is 5. The analysis run has been repeated with a first-order upwind scheme (Roe, 1981) (1st OU, with simplifications such as variable-averaging) and a second-order TVD extension of this scheme (Hirsch, 1990) (FDSTVD for flux difference splitting TVD scheme) using the minmod limiter. All three schemes yield comparable convergence rates, giving a residual drop of 4 orders in about 500 iterations. Although, apart from pressure ratio, the first-order upwind scheme appears to agree well with the FDSTVD scheme globally, table 6.4, inspection of the radial profiles of the calculated loss coefficient, Fig. 6.10a, reveals that this agreement is fortuitous: insufficient strength of the strongly diffused shock in the outboard region, Fig. 6.11, is balanced by excessive numerical dissipation in the inboard region; at the hub, the numerically generated losses reach 10%. The downstream absolute flow angle, $\alpha_{TE}$, is another sensitive variable, 6.10b. Here, the maximum difference, occurring at the casing, is 4.5 deg. The two second-order schemes predict practically the same pressure ratio, but differ by 2% in mass-flow and 1.2 percentage points in efficiency. Referring again to the spanwise profiles, Fig. 6.10, this divergence might be caused, or at least exacerbated, by the captured shock: the solutions are nearly identical in the subsonic region below about 40% span, but differ by up to 2.5% in loss coefficient and 1.5 deg in $\alpha_{TE}$ in the supersonic region. The shocks captured with the central and FDSTVD schemes do occur at exactly the same position, but, because of the lower massflow, the FDSTVD solution predicts a higher incidence angle and therefore increased turning and thus acceleration of the supersonic flow, leading to a higher shock upstream Mach number and a stronger shock, Fig. 6.11b.

A mesh dependence study performed in analysis mode with the central scheme indicated negligible dependence on the spanwise direction (16, 32 and 64 cells tested), but considerable dependence on the streamwise direction (16, 32 and 64 cells between LE and TE tested). In the latter case, the differences were again limited to the supersonic outboard region and similar in magnitude to those between the central and FDSTVD scheme.

The observed mesh and scheme dependence appears to be shock-related and is not present in design mode.

---

**Table 6.4: Influence of the spatial discretization.**

<table>
<thead>
<tr>
<th>Scheme</th>
<th>$\dot{m}$ [kg/s]</th>
<th>$\Pi_t$</th>
<th>$\eta_{ls}$ [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>central</td>
<td>30.25</td>
<td>1.831</td>
<td>89.87</td>
</tr>
<tr>
<td>FDSTVD</td>
<td>29.67</td>
<td>1.830</td>
<td>88.65</td>
</tr>
<tr>
<td>1st OU</td>
<td>29.63</td>
<td>1.804</td>
<td>88.36</td>
</tr>
</tbody>
</table>

The two second-order schemes predict practically the same pressure ratio, but differ by 2% in mass-flow and 1.2 percentage points in efficiency. Referring again to the spanwise profiles, Fig. 6.10, this divergence might be caused, or at least exacerbated, by the captured shock: the solutions are nearly identical in the subsonic region below about 40% span, but differ by up to 2.5% in loss coefficient and 1.5 deg in $\alpha_{TE}$ in the supersonic region. The shocks captured with the central and FDSTVD schemes do occur at exactly the same position, but, because of the lower massflow, the FDSTVD solution predicts a higher incidence angle and therefore increased turning and thus acceleration of the supersonic flow, leading to a higher shock upstream Mach number and a stronger shock, Fig. 6.11b.
6.3 Detailed Analysis of the Design Speed Operating Line

6.3.1 Analysis of the Pitch-Averaged 3D Flow
Figure 6.12  Computational grids: (a) meridional projection of the 3D Navier-Stokes mesh, (b) the Euler throughflow mesh.
Five characteristic operating points were selected: near stall (NS), mid-range (MR), peak efficiency (PE), near choke (NC) and choked at a low pressure ratio (CH).

The pitch-averaged solution is defined by the mass averaged primitive variables. Figure 6.13 shows the spanwise profiles of incidence angle, deviation angle, exit swirl and loss coefficient extracted from the pitch-averaged 3D Navier-Stokes solutions for the five operating points. Because normal shocks are expected to be captured in analysis mode, the associated losses (the captured shock losses) were approximatively removed from the loss profiles imposed in analysis calculations by subtracting the losses of a normal shock at the inlet Mach number, the latter being assumed to vary linearly from 1 at 40% span (measured from the hub) to 1.4 at the casing. Near stall, a vortex forms at the LE tip, Fig. 6.14a (caution: the figure does not show the flow in the tip gap proper but the circumferentially averaged flow in the tip region; the blade contours are drawn to aid orientation.) The ensuing recirculation zone extends along the casing beyond the TE, causing considerable end-wall blockage. Corrections therefore had to be made to the swirl and loss profiles for this operating point.
near the casing (the bold straight line segments in Figs. 6.13c and d), smoothing out strong local variations to give profiles close to the mixed out ones one chord downstream. The LE tip vortex is negligible for the four other operating points, Fig. 6.14b; in particular, there is no backflow at the TE. The extracted profiles could therefore be used directly.

Figure 6.15 compares the streamwise distributions for $\beta$, $W_\theta$ and $\psi_{rot}$ extracted from the pitch-averaged 3D solutions at three spanwise positions (12 %, 52 % and 88 % span from the hub) with power functions for selected exponents (5, 2, 1.2, 1, –1.2, –2, –5). Four of the five selected operating points are shown: NS, PE, NC and CH. In the throughflow model, the same power function distribution applies to all spanwise locations. Figure 6.15 indicates that this is a good approximation in most cases. S-shaped distributions, such as the choked swirl distribution, Fig. 6.15k, and overshoots, such as in the choked flow angle distribution, Fig. 6.15j, cannot be represented by the current simple power function model. Based on these figures and past experience, a $\beta$-exponent of 2 was chosen for all operating points (analysis mode) and $W_\theta$-exponents of 2, 1.75, 1.5, 1 and again 1 for the near stall, mid-range, peak efficiency, near choke and choked at low pressure ratio operating points, respectively (design and hybrid modes). The loss distribution, within the reasonable bounds observed in Figs. 6.15c, f, i and l, is not a sensitive parameter and was therefore chosen linear for all cases ($\psi$-exponent 1).
Figure 6.15 3D pitch-averaged streamwise distributions of relative flow angle ($\beta$, left column), relative tangential velocity ($W_{\theta}$, middle column), and loss coefficient ($\psi_{rot}$, right column). Top row = near stall (NS), second row = at peak efficiency (PE), third row = near choke (NC), bottom row = choked (CH). Triangles = near the casing (88 % span), circles = at mid-span (52 %), squares = near the hub (13 % span). Solid lines = power functions with exponents 5, 2, 1.2, 1, -1.2, -2, -5.
An exception had to be made for the choked operating point at low pressure ratio (CH). Contrary to the design mode, the analysis mode has a built-in choking mechanism. A linear loss distribution leads to losses in the forward part of the blade passage which, near the hub and at mid-span, are too high compared with the actual values, Fig. 6.15. With the ensuing diminished total pressure, the blade passage throat can pass only a reduced massflow; compared with the NC operating point, the choking massflow obtained with a linear loss distribution is underpredicted by 2%. A consistent value is obtained with the power function exponent \(-2.5\) suggested by Fig. 6.15. The analysis of the pitch-averaged 3D flow field thus provides both qualitative and quantitative information on the streamwise distributions of flow angle or swirl, and losses. Although they have little impact on the downstream flow, they are important for the flow field inside the blade passage and the choking massflow in analysis mode.

In accordance with the analysis performed in section 6.3.1, features of the pitch-averaged streamwise distributions of Fig. 6.15 can be associated with the various oblique and normal shocks in the blade-to-blade plane. As an example, Fig. 6.16 shows pressure and Mach contours at the same three spanwise positions. At 88% span, top row of Fig. 6.16, expansion of the supersonic flow along the slightly cambered suction surface together with the deflection through the oblique bow shock cause significant overturning even in the pitch-average, witnessed in Fig. 6.15j. This is undone by the oblique passage shock which emanates from the suction side of the trailing edge and which is reflected off the pressure surface. This reflected shock and another weak oblique shock originating from the trailing edge extend into the downstream region. Work input, Fig. 6.15k, is smooth and continuous. Losses rise steeply where shock strength is high (at and immediately downstream of the leading edge) and where the percentage of axial chord covered by the circumferential projection of oblique shocks is small (the passage shock is of roughly the same strength as the bow shock—both have an upstream Mach number of 1.45 and similar obliquity—but its projected length, between \(x \approx 0.7\) and 1, is smaller than that of the bow shock, between \(x = 0\) and 0.8). Similar reasoning can be applied to the mid-span and hub sections, middle and bottom rows of Fig. 6.16. Here, it is the normal or near-normal passage shock that accounts for most of the losses, Fig. 6.15l, with only minimal losses upstream.
6.3.2 Throughflow Calculation Strategy

The mesh is the same quasi-uniform mesh used for the preliminary calculations of the preceding section, with 32 x 32 cells in the blade passage, the mesh sensitivity study having shown this to be an adequate size, guaranteeing a sufficiently low level of numerical dissipation. Side-by-side comparison with a meridional projection of the 2D Navier-Stokes mesh shows that the spatial resolution of both is indeed comparable.

The spanwise profile of maximum blade thickness, \( d_{\text{max}} \), was obtained from the blade geometry. The location of \( d_{\text{max}} \) is at 60% axial chord at the tip and 50% at the hub, with a linear transition. The streamwise blade thickness distribution is quadratic (power function with exponent 2).

Each of the five selected operating points was calculated in each of the three throughflow modes as described in the following.

In design mode, the imposed exit swirl fixes the work input (up to the effect of radial streamline displacement with varying back pressure). Because the flow remains axially subsonic, the design mode does not have a built-in choking mechanism. Variation of the exit pressure therefore results in horizontal displacement of the operating points in the performance diagrams: the massflow adapts, at nearly constant pressure ratio and efficiency.

Both analysis mode and hybrid mode do have a built-in choking mechanism: the fixed flow angle limits the effective area available to the relative flow, expressed by the effective blockage factor, \( b_{\text{eff}} \), according to

![Figure 6.16 For the choked operating point (CH), contours of (a) static pressure coefficient and (b) relative Mach number at three spanwise positions (top row = near the casing, middle row = at mid-span, bottom row = near the hub).](image-url)
Even though in 3D the situation is complicated by AVDR variation for individual blade-to-blade sections, the same reasoning applies for the blade passage as a whole. In analysis mode, the CH and NC operating points were run to the correct pressure ratio. The three non-choked operating points (PE, MR and NS) were then run to the correct fraction of the choking massflow. The same procedure was followed in hybrid mode, except for NC, which was run to the correct fraction of choking massflow because its own choking massflow (i.e., that obtained with the NC profiles instead of the CH profiles, but with the same streamwise distributions) turned out to be higher than that of CH.

Figure 6.17 compares the resulting performance curves with those of the reference 3D Navier-Stokes calculations.

Also shown are the measured performance curves. A discussion of possible reasons for the discrepancy between the 3D calculation and the experimental data is beyond the scope of this paper. The reference for the throughflow calculations are the calculated 3D results, because this is from where the input profiles have been taken.

Each curve has been normalized with respect to its own choking massflow. All lie within 2% of the experimental value. While the design and hybrid modes agree remarkably well with the reference 3D calculation, the analysis mode overestimates both pressure ratio and efficiency of the non-choked operating points. This is explained by the presence of multiple shocks, to be discussed below, and the modified loss profiles. The shock pattern is insensitive to the imposed losses. Even though one might be tempted to conclude that the captured shock losses have been overestimated when removing them from the extracted ones, it should not be forgotten that the presence of multiple shocks has of course a profound impact on AVDR. The correct exit angle does therefore not guarantee the correct exit swirl and thus work input.

\[
b_{\text{eff}} = b \cos \beta
\]  

Figure 6.17 NASA Rotor 67 performance curves: (a) absolute total pressure ratio, (b) isentropic efficiency. Open circles = experiment, open squares = 3D calculation, filled squares = design mode, filled triangles = analysis mode, filled diamonds = hybrid mode.
6.3.3 Comparison with Pitch-Averaged 3D Flow Fields

Figures 6.18 to 6.25 compare the flow fields obtained in analysis mode, in design mode, and in the new hybrid mode with the pitch-averaged calculated 3D flow field through relative Mach number (\(M_{rel}\), Figures 6.18 and 6.19), static pressure coefficient (\(C_p\), Figures 6.20 and 6.21), relative flow angle (\(\beta\), Figures 6.22 and 6.23), and relative tangential velocity (\(W_{\theta}\), Figures 6.24 and 6.25). Rows correspond to modes (from top to bottom: the analysis mode, the pitch-averaged 3D solution, the design mode, and the new hybrid mode), columns correspond to different operating points (from left to right: near stall (NS), peak efficiency (PE), near choke (NC), and choked (CH)).

A first overview suggests that, except for the flow angle, Figures 6.22 and 6.23, design mode and analysis mode differ dramatically inside the blade passage and that the design mode better approximates the pitch-averaged 3D flow field.

In accordance with theory, the design mode does not capture shocks, because the axial Mach number remains subsonic. An exception is the CH operating point, for which a weak shock appears to be captured at the blade root at 2/3 chord, Figs. 6.19g and 6.21g. Incidentally, this is also the location of the passage shock in 3D, Fig. 6.16. Between the hub and mid-span, this passage shock is nearly axisymmetric and consequently appears as a near-discontinuity in the pitch-average, Figs. 6.19f and 6.21f.

The analysis mode, independently of the operating point, captures a normal shock at the blade leading edge, e.g., Figs. 6.18a and e, and 6.19a and e. For all operating points except the NS one, this is followed by a second normal shock at or near the trailing edge. Going along the performance curve from stall to choke (the sequence of figures 6.18a, 6.18e, 6.19a, 6.19e), the reacceleration to supersonic flow and the strength and inboard extension of the second shock intensify. This process is associated with a downstream displacement of the second shock, which gradually moves off the blade, thereby allowing the required supersonic exit flow in the outboard region at choked operation, Fig. 6.19e.

Despite the differences inside the blade passage, the downstream flow fields in design and analysis mode are nearly the same, and in very good agreement with the reference 3D solution. This observation holds unreservedly for Mach number and pressure coefficient, for the NS and PE operating points. Near stall, the averaged 3D flow has a low velocity/recirculation zone at the blade tip that gradually mixes out, Fig. 6.18b, see also Fig. 6.14a. It is caused by phenomena in the blade-to-blade plane and interaction with the tip leakage flow and does therefore not arise in the throughflow calculations, Figs. 6.18a, c, and d. Near choke, distinctions begin to appear in the downstream field, in the form of additional expansion/acceleration in the lower 60–70 % span, which is not observed in the averaged 3D solution and more pronounced in analysis mode than in design mode, Figs. 6.19 and 6.21.

Another interesting feature of the averaged 3D flow on which both throughflow solutions necessarily miss out is the effect of wake blockage and mixing, cf. Fig. 6.16b. This is most visible at the blade root, immediately downstream of the trailing edge, from the hub to about 30 % span, Figs. 6.18 to 6.21. Here, the throughflow isoxline patterns of both \(M_{rel}\) and \(C_p\) are ‘centered’ about the trailing edge, while in the averaged 3D solution the pattern is shifted some way downstream. This also has consequences on the flow angle, Figs. 6.22 and 6.23. The representation of wake blockage in the throughflow might therefore be desirable because it would allow a more realistic throughflow representation. On the other hand, it would add an additional modelling parameter to be tuned or correlated.

The fact that the smooth compression in the forward part of the blade passage, Figs. 6.18b,f and 6.20b,f, is so well represented in the design solution, Figs. 6.18c,g and 6.20c,g, is due to the realistic choice of streamwise \(W_{\theta}\)-distribution that could be made for these cases, Figs. 6.15a and d. For distributions which cannot be well represented by the simple power functions, e.g., those near choke (NC), Figs. 6.15g and 6.19b, 6.21b, agreement is less good, Figs. 6.19c, 6.21c. S-shaped \(W_{\theta}\)-distributions prevail in this range, with steeper gradients and therefore isoxline clustering in the middle of the blade, while the best-compromise linear distribution of the throughflow produces uniform deceleration. If this kind of information is available, it might be worth while to allow more detailed throughflow modelling via independent control of the hub, mid-span and tip regions, and via two-
However, the error made by simplified streamwise $W_\theta$-distributions in design mode is much smaller than that caused by the captured shocks in analysis mode, and its consequences on the downstream flow are presumably negligible. The latter is not true of the analysis mode, where the flow angle field is visibly distorted downstream of the second, trailing edge shock, Figs. 6.22e and 6.23a,e. The best agreement, for all operating points, is given by the hybrid mode, Figs. 6.22d,h and 6.23d,h. Conformity with 3D is even particularly good for choked operation at low pressure ratio, Figs. 6.23f and h, where the unturning at about 3/4 chord is well reproduced, also, in slightly milder form, in design mode, Fig. 6.23g. This effect is captured without any tuning of the $W_\theta$-distribution, which is linear for the NC and CH operating points. In the analysis solution it is necessarily absent, Fig. 6.23e. Inside the blade passage, design mode and hybrid mode yield correct swirl fields, Figs. 6.24 and 6.25, while the analysis mode deviates strongly where shocks are captured, from about 50 % span to the casing.

To avoid arbitrary modification of the extracted profiles, it was left to the Euler throughflow mesh to filter out end-wall boundary layer effects. The high flow angles near the stationary casing extend far enough inward to be partially visible on the throughflow mesh, however. As a consequence, both the analysis and the hybrid solutions have a zone of low and even negative turning at the casing, Figs. 6.22a,d,e,h and Figs. 6.23a,d. Owing to diffusion on the Euler mesh, the low-turning zone spreads radially inwards downstream of the blade row.

### 6.4 Conclusions

The shock capturing properties of the Euler throughflow equations have been determined through analysis of the Rankine-Hugoniot relations. The design mode captures axisymmetric shocks, the analysis mode quasi-normal shocks.

The shock capturing properties were found to depend only on the streamwise distributed variable, not on the variable for which a trailing edge profile is imposed. This enabled the construction of a novel mode, termed the hybrid mode, which combines the imposed trailing edge flow angle of the analysis mode with the shock capturing properties of the design mode.

For an isolated 2D blade-to-blade section, analysis mode and design mode have been analytically compared with the pitch-averaged representation of a normal shock at high flow angle, showing the deviation of the analysis mode from the averaged solution.

3D Navier-Stokes solutions of a transonic axial compressor rotor for different operating points have been analyzed in detail with regard to the profiles and distributions required as input to the throughflow model. Throughflow calculations in the three modes were then performed with the extracted swirl (design mode) or flow angle (analysis and hybrid modes) and loss profiles and the results compared to the 3D pitch-averaged solution, with the following conclusions:

1. The analysis mode captures a normal shock at the leading edge and, depending on the operating point, a second normal shock near the trailing edge.
2. As a consequence of this, the blade passage flow field in the upper half-span is not correctly represented in analysis mode.
3. The smooth, continuous deceleration in the pitch-averaged 3D solution is correctly represented by the design mode and by the hybrid mode.
4. Despite the erroneous solution inside the blade passage, the downstream flow field in analysis mode is largely correct.
5. Design mode and hybrid mode converge two to four times faster than the analysis mode, for transonic flows with captured shocks.
Figure 6.18 NASA Rotor 67: isolines of relative Mach number ($M_{rel}$, increment 0.05). Left column = near stall (NS), right column = at peak efficiency (PE). Top row = analysis mode, second row = 3D pitch-average, third row = design mode, bottom row = hybrid mode.
Figure 6.19  NASA Rotor 67: isolines of relative Mach number ($M_{rel}$, increment 0.05). Left column = near choke (NC), right column = choked (CH). Top row = analysis mode, second row = 3D pitch-average, third row = design mode, bottom row = hybrid mode.
Figure 6.20 NASA Rotor 67: isolines of static pressure coefficient ($C_p$, increment 0.025). Left column = near stall (NS), right column = at peak efficiency (PE). Top row = analysis mode, second row = 3D pitch-average, third row = design mode, bottom row = hybrid mode.
Figure 6.21  NASA Rotor 67: isolines of static pressure coefficient ($C_p$, increment 0.025). Left column = near choke (NC), right column = choked (CH). Top row = analysis mode, second row = 3D pitch-average, third row = design mode, bottom row = hybrid mode.
Figure 6.22 NASA Rotor 67: isolines of relative flow angle ($\beta$ in deg, increment 2.5 deg). Left column = near stall (NS), right column = at peak efficiency (PE). Top row = analysis mode, second row = 3D pitch-average, third row = design mode, bottom row = hybrid mode.
Figure 6.23  NASA Rotor 67: isolines of relative flow angle ($\beta$ in deg, increment 2.5 deg). Left column = near choke (NC), right column = choked (CH). Top row = analysis mode, second row = 3D pitch-average, third row = design mode, bottom row = hybrid mode.
Figure 6.24 NASA Rotor 67: isolines of relative tangential velocity ($W_\theta$ in m/s, increment 20 m/s). Left column = near stall (NS), right column = at peak efficiency (PE). Top row = analysis mode, second row = 3D pitch-average, third row = design mode, bottom row = hybrid mode.
Figure 6.25 NASA Rotor 67: isolines of relative tangential velocity ($W_\theta$ in m/s, increment 20 m/s). Left column = near choke (NC), right column = choked (CH). Top row = analysis mode, second row = 3D pitch-average, third row = design mode, bottom row = hybrid mode.
Chapter 7  Other Throughflow Test Cases

7.1  Four-stage low speed turbine

The multistage capability of the throughflow module is demonstrated with test case E/TU-4 of Fottner (1990), a four-stage low speed turbine (nominal power 703 kW, 7500 rpm, massflow 7.8 kg/s, pressure ratio 2.5, efficiency 91 %, degree of reaction 0.5, hub ratio at exit 0.525, constant hub radius 135 mm). Figure 7.1 shows the mesh.

An important benefit of the multigrid technique is the mesh size-independent convergence rate. Applied to multi-stage configurations this means that even for machines with a large number of stages there is no significant slow-down compared with individual components. Figure 7.2 shows the convergence histories of density (the lower curves) and axial momentum for rotor 1 computed alone (dashed lines) and embedded in a computation of the complete machine (solid lines). For this test case, a residual drop of 3 to 4 orders and stabilization of the massflow are reached after 30 to 70 multigrid cycles, depending on the initial solution and on the operating point. Four-level multigrid in a V-cycle with 1, 2, 3 and 4 smoothing sweeps on respectively grid levels 1, 2, 3 and 4 were used. The computations were performed with the central scheme, the coefficient $\kappa$ in Eq. (5.4) is 1 and the CFL number is 6.

Figure 7.3 shows contours of relative Mach number at the design point. Contrary to transonic test cases, blade thickness and the flow angle distribution are not critical parameters. Deviation angles and loss coefficients are the same for all stages and were adjusted to match massflow and efficiency at the design operating point with the experimental boundary conditions at machine inlet (total pressure and temperature) and exit (static pressure) and kept constant. Figure 7.4 shows the resulting performance curves.

![Figure 7.1  Turbine E/TU-4: computational grid.](image)
Figure 7.2 Turbine E/TU-4: comparison of convergence of rotor 1 if computed alone (dashed lines) and in a computation of the complete machine (solid lines), curves are for density and axial momentum.

Figure 7.3 Turbine E/TU-4: contours of relative Mach number at design point.
7.2 Transonic Turbine Vane

The effectiveness of both the robust analysis mode and throat control are demonstrated with the stator of a steam turbine last stage taken from the AGARD-AR-275 collection of validation test cases (Fottner, 1990, test case E/TU-2). The published results concern tests with air performed on a 1/6 scale model, preserving both Mach and Reynolds similarity. The inlet total pressure is 80 kPa, the inlet total temperature 300 K. Figure 7.5a shows the mesh, containing 32 cells radially and 72 cells axially (32 upstream of, 32 inside, and 8 downstream of the blade passage, for a total 2304 cells). The inlet to the computational domain was placed some way upstream of the blade row at a location with low end-wall curvature. The inlet boundary condition imposes total pressure and total temperature, and the flow direction through two angles. Inflow is in the meridional plane (yaw angle $\tan(V_\theta/V_z) = 0$), and parallel to the end-walls at hub and casing, with a linear variation of the pitch angle $\tan(V_r/V_z)$. The exit of the computational domain has been placed approximately 13 mm (between 1/2 and 1/3 axial chords) downstream of the trailing edge to coincide with the downstream radial measurement traverse. In the experiment, the presence of the rotor is simulated through a perforated plate located 32 mm (approximately 1 axial chord) downstream of the trailing edge. At the exit boundary, the static temperature and cylindrical velocity components are extrapolated with first order, the static pressure is imposed. The perforated plate appears to have a rather drastic influence on the exit pressure profile. If an exit pressure is imposed around midspan in conjunction with simplified radial equilibrium, the obtained exit static pressure profile is substantially different from the experimental one. The experimental static pressure profile (available in five points evenly spaced between hub and casing) was therefore imposed directly, as suggested by the authors (Ball et al., 1990).

The E/TU-2 test case presents the essential features of a last stage low pressure steam turbine nozzle. Citing from Ball et al. (1990): a low hub-to-tip ratio (0.484 at the TE), high turning ($\cos(t/s) = 69.61$ deg for the hub profile and 75.56 deg for the tip profile), high casing flare (with a maximum of 55 deg according to the authors, 52.3 deg according to the provided flow path geometry), and supersonic flow at the hub (M $\approx 1.5$) and subsonic flow at the tip (M $\approx 0.8$). The blade count is 38.

The blade geometry is defined by two sections at constant radius, located outside of the flow path, at $-12.5\%$ span and $151.0\%$ span at the LE hub and tip, and $-9.2\%$ span and $109.5\%$ span at

---

**Figure 7.4** Turbine E/TU-4: performance curves: (a) absolute total pressure ratio, (b) isentropic efficiency (solid lines = computed, dashed lines = experiment, D.P. = design point).
the TE hub and tip. This blade profile geometry was processed as follows to extract the throughflow modelling parameters. It was assumed that the throat of the actual 2D blade-to-blade section is formed by the TE point on the pressure side and by the suction side of the next blade and the throat width defined as the shortest distance between the two. In the computational model, the throat is located at 83% chord at the hub and at 87% chord at the casing. (The exact positions would be 0.8323 at the hub profile radius and 0.8684 at the casing profile radius.) The streamwise distribution of the effective blockage is quadratic (power function with exponent 2), with throat (and therefore minimal) values of 0.3484 at the hub and 0.2494 at the casing profile radii.

To avoid arbitrary assumptions on the deviation angle, the experimental exit flow angle profile, measured along the traverse about 1/3 axial chord downstream, was imposed at the blade trailing edge. The leading edge flow angle is zero to match the swirl-free inflow, the streamwise flow angle distribution between LE and TE is linear. The loss coefficient was calculated from the definition and the measured exit total pressure profile. Losses are linearly distributed between LE and TE. All modelling parameters given only at hub and casing are linearly interpolated along spanwise mesh lines.

Of the two reported test section pressure ratios (0.3 and 0.42, defined as the ratio of downstream casing static pressure to upstream stagnation pressure and abbreviated TSPR), only the lower one of 0.3 was considered, since it is more likely to represent a choked operating point.

The initial solution is uniform, with \( p = 60 \text{ kPa}, T = 270 \text{ K}, V_t = 25 \text{ m/s}, V_0 = 0, \) and \( V_z = 50 \text{ m/s} \). The fluid is a perfect gas with \( R = 287 \text{ J/kg/K} \) and \( \gamma = 1.4 \) (\( c_p = 1004.5 \text{ J/kg} \)). The blade force is normal to the target velocity (as opposed to the current velocity). The central scheme with the standard values of the dissipation coefficients is used (VIS2 = 1, VIS4 = 0.1), in conjunction with 4-stage Runge-Kutta time integration (coefficients 0.125, 0.306, 0.587, and 1). The artificial dissipation is recalculated on the first and second Runge-Kutta stages only. Implicit residual smoothing employs a time-step-dependent coefficient (Zhu, 1997). Four multigrid levels in a V-cycle are used, with 1, 2, 3, and 4 smoothing sweeps on respectively grid levels 1, 2, 3, and 4 (1 denoting the finest grid level). Restriction and prolongation are both linear.

This test case is robust and converges rapidly (CFL = 6, \( \kappa = 1 \)). Figure 7.5b shows the convergence history. Engineering accuracy is reached in 100 multigrid cycles. With the described direct control of the nozzle throat, the correct massflow was obtained immediately: 10.37 kg/s (should be \( 10 \pm 0.2 \text{ kg/s} \) according to the experiment). This is to be compared with 11.98 kg/s in classical analysis mode with prescribed flow angle and tangential blade thickness.

Compare here: virtual tangential blade thickness of throat control mode with actual tangential blade thickness, or effective blockage resulting in classical analysis mode with controlled effective blockage.

A number of tests were conducted to assess the robustness of the new numerical model. Reversing the direction of the initial velocity vector (\( V_z = -70 \text{ m/s} \)) does not provoke divergence, nor do extreme imposed TE flow angles (between 85 and 89 deg). The only observed mechanism for divergence was (uncontrolled) backflow in the exit section.

An impression of the flow is given in Figure 7.6 through meridional streamlines (Figure 7.6a) and contours of Mach number (Figure 7.6b). Note the strong variation in radial streamline position and consequent streamtube area variation inside and especially downstream of the blade row. It would appear, however, that for some unknown reason the pitch angle is slightly overestimated. Due to a captured shock located at the trailing edge, the downstream flow angle is difficult to control. For example, at the hub, the flow angle increases first discontinuously through the trailing edge shock from the imposed 56.5 deg to 68 deg, and then smoothly to 77 deg at the exit.

Losses from the captured shock are added to the imposed losses. As a consequence, the calculated exit total pressure is slightly below the experimental one, Figure 7.7a. The amount of additional losses depends on shock strength, which in turn depends on shock upstream Mach number, Figure 7.6a, and shock obliquity. Hence, there are no captured shock losses at the casing, where the flow remains entirely subsonic, and the calculated and experimental total pressure exactly coincide. A strong increase in losses is observed over the inner 30% span, both in the experiment and the calculation. The fact that the calculation overpredicts the losses by up to 70% suggests that most of the losses
there are shock losses. In an actual design application, the shock losses would have to be excluded from the imposed losses, at least approximately, because they will be captured automatically by the Euler throughflow model in analysis mode.

Ball et al. (1990) point to the radial distribution of Mach number as the “most relevant parameter for comparison.” As seen from Figure 7.7b, it is well predicted by the calculation, except at the blade root, where the discussed excess of losses leads to an underprediction of about 7%. The global trend is however perfectly reproduced.

**Figure 7.5** Euler throughflow calculation of E/TU-2: (a) meridional mesh (32 x 72 = 2304 cells), (b) convergence history with robust analysis mode and direct control of the throat width.
Figure 7.6 Euler throughflow calculation of E/TU-2 with direct control of the throat width: (a) meridional streamlines, (b) isolines of Mach number (increment 0.05).

Figure 7.7 Euler throughflow calculation of E/TU-2 with direct control of the throat width: exit spanwise profiles of (a) total pressure and (b) Mach number.
7.3 Supersonic Impulse Turbine

The applicability of the Euler throughflow model is not limited to transonic blade rows. Fully supersonic blade rows can be treated as well, and this is demonstrated here with the supersonic impulse turbine shown in Figure 7.8a. Such turbines are used in space propulsion to drive the fuel and oxidizer turbo-pumps of rocket engines, where they provide the required power at minimum structural mass. The poor efficiency inherent in this design is outweighed by the performance advantage in terms of a high power density.

The selected turbine is composed of a first nozzle guide vane, the so-called distributor, which receives the high-enthalpy gas from a toroidal inlet plenum and accelerates it to supersonic relative velocity, followed by the first-stage rotor and the second-stage stator and rotor, all three bladed with classical symmetric impulse buckets of approximately constant flow path width, designed to maintain supersonic flow. A final OGV serves to deswirl the flow which is then ejected through a nozzle to exploit its residual energy in the form of a small amount of additional thrust.

The OGV being only mildly supersonic with a conventional design and therefore of no particular interest, only the first four blade rows are modelled in the presented calculation. Each blade row is meshed with 16 cells radially and 48 cells axially, giving a total mesh size of 3072 cells. Experience has shown that meshes of this size yield solution accuracy commensurate with the hypothesis of axisymmetric flow. Because simple-wave discontinuities are expected to be captured at the blade leading and trailing edges, mild clustering is employed there to improve spatial resolution.

The working fluid of this particular turbine is a mixture of hydrogen and water vapor, resulting from the combustion of LH2 with LOX at an excess of LOX. When the reported calculations were performed, the tabular state equation had not yet been available. The mixture of combustion gases was therefore modelled as a perfect gas with a constant ratio of specific heats, \( \gamma = 1.38 \), and a specific gas constant \( R \approx 2170 \text{ J/kg/K} \) (isobaric specific heat capacity \( c_p \approx 7900 \text{ J/kg/K} \)).

The massflow is 4.1 kg/s, the shaft speed 12 600 rpm, and the shaft power 5.1 MW. Loss coefficients for the individual blade rows range between 0.6 and 0.8 (\( \psi_{rel} \), defined in Eq. XX), which means that between 60% and 80% of the exit (TE) dynamic pressure is lost to heat through shock waves and friction. For comparison, losses in conventional turbines are of the order of 10%. The streamwise distribution of the losses is linear in all elements, except in the distributor, where only 5% of the total losses are assumed to occur in the subsonic part, between the inlet and the nozzle throat, and 95% in the supersonic part, with a linear distribution in each part.

The static and total pressure ratios of the modelled configuration are respectively about 30 and about 12. A static pressure ratio of about 8 is accomplished by the distributor alone, reflecting the acceleration from near stagnation conditions to a Mach number around 1.7. The work load and thus the total pressure drop are evenly distributed on the two stages (ratios of 3.8 and 3.1, respectively). What is remarkable, though, is the fact that the drop of total pressure in the stators (ratios of 1.7 and 1.5, caused only by losses), is of the same order as that in the rotors (ratios of 2.3 and 2.1).

At the inlet, constant profiles of total pressure and temperature are specified. The flow direction is purely axial. The exit boundary condition imposes the static pressure at mid-height, the radial distribution is dynamically calculated assuming simplified radial equilibrium. A slip condition is applied at the end-walls, where the wall pressure is set from linear extrapolation.

All four blade rows are modelled in robust analysis mode with throat control. The tangential blade thickness is thus not imposed independently but derived from the imposed flow angle distribution and throat width. However, only in the distributor is there a physical throat (located at 1/3 axial chord form the leading edge). In the three other blade rows, no throat is specified. The respective option of the code then applies a monotonous distribution of the effective blockage between the specified leading and trailing edge values. In the distributor, the two branches (upstream and downstream of the throat) of the \( b_{eff} \)-distribution are quadratic power functions to ensure slope continuity at the throat, while in the three other blade rows the \( b_{eff} \)-distribution is linear. In the general case, imposing the flow angle at the leading edge engenders a discontinuity, because the calculated exit flow angle of the preceding blade row is not (and should not) be controlled directly and will therefore be different. Although, thanks to a continual strive for robust model and algorithm formulations, the solver can handle this situation without problems, one might in some cases question the physical signifi-
Other Throughflow Test Cases

The deviation follows a quadratic power function which is slope continuous with the specified fixed flow angle distribution. A linear streamwise distribution is applied to $\tan \beta$. At the high exit flow angles typical of turbine vanes, applying a linear distribution to the flow angle itself results in excessive tangential acceleration very close to the trailing edge (and in the case of the present impulse blading a corresponding deceleration at the leading edge). Although in 3D the turning is indeed biased towards the blade trailing edge, a linear flow angle distribution overestimates this trend, and without suitable clustering the resulting strong gradients would be poorly resolved numerically, with the obvious consequence of diminished accuracy and increased dissipation.

Note that whenever it is possible to do so, distributions are pre-calculated and stored during the start-up phase (in dimensional or in normalized form), which avoids redundant evaluation of the costly trigonometric functions in the repetitive iteration phase.

The calculation proceeds from an approximate, spanwise constant initial solution with linear distributions of the primitive variables in each block. The magnitude of the blade force is initialized algebraically to be consistent with the initial velocity and density fields.

For this supersonic throughflow test case, a CFL number of 6 could be used, but the step-like coefficient $\kappa$ in the blade force equation had to be reduced slightly from its default value for sub- and transonic flows (from 1 to 0.5). The convergence history is presented in Figure 7.9. For this difficult case, convergence is obtained in only 300 multigrid cycles. Note how the convergence of the time-dependent blade force exactly parallels that of the other variables, as one would expect from a tightly coupled implementation which applies to the artificial blade force equation the exact same numerical techniques as to the basic Euler equations.

Mach number contours of the converged steady state solution are shown in Figure 7.8b, and these most impressively highlight the utility of meridional solutions to quickly and inexpensively assess designs at an early stage. In view of the low aspect and high hub ratios of the examined blading (respectively between about 0.5 and 1.3 and between 0.8 and 0.9), one might expect to find little spanwise variation in the flow. Figure 7.8b proves this reasoning to be wrong, revealing surprisingly strong departures from the mean line solution. While the Mach number level indicated by the Euler throughflow simulation is consistent with that predicted by a 1D mean line analysis, and while the flow in the distributor is indeed very homogeneous, the abrupt widening of the flow path causes substantial spanwise non-uniformity in rotor 1, witnessed by a minimum Mach number around 1.4 near the casing, slightly upstream of the blade centerline, and a maximum around 1.9 near the hub, slightly downstream. A corresponding distortion appears in the meridional streamlines, shown in Figure 7.8c, which in rotor 1 visibly are no longer parallel. Remarkably, these modulations do not reflect the symmetry of the flow path, suggesting a high sensitivity with respect to the spanwise flow angle profiles (exactly constant at the blade leading edges and nearly constant at the blade trailing edges).

The rather peculiar flow pattern in stator 2 is easily explained with gas dynamic theory: the convex corner in both end-walls at the leading edge gives rise to a Prandtl-Meyer expansion, followed by compression waves emanating from the concavely curved sections of the end-walls, and an oblique shock attached to the final straight part. In analysis mode, the flow inside blade passages corresponds to an equivalent 3D flow in the limit of infinite solidity (or blade number). The design mode would see the other extreme of vanishing solidity, in which case the behavior of the flow at the kink in the end-walls at the LE would be opposite: the flow would decelerate in the widening passage, at
least as long as it remains axially subsonic.

In this context, another difficulty associated with the throughflow calculation of supersonic flows must be mentioned, namely the high sensitivity with regard to the modelling parameters: seemingly insignificant changes may lead to completely different solutions in parts of the flow field, for example subsonic instead of supersonic flow, and stator 2 of this turbine is one example. If, instead of allowing the described adaptation zone at the LE, the flow angles from a preliminary 1D mean-line design are imposed throughout the blade passage, a solution with a normal shock at the blade passage entrance and subsequent subsonic flow is obtained. It is interesting to know that a 3D Euler calculation (without additional imposed losses), in a geometry as seen by the throughflow calculation and on a comparable mesh, also predicts a predominantly subsonic solution, with a quasi-normal shock at the blade passage entrance, which in the pitch-average is similar but not identical to the subsonic throughflow solution. While the supersonic solution presented here slightly overpredicts the shaft power, the subsonic solution (in stator 2 only, supersonic in all other blade rows) results in a comparable underprediction (5 MW at a mass flow of 4.4 kg/s). The discussed extremes therefore appear to correctly bracket the actual 3D flow.

Figure 7.10 shows the streamwise evolution between inlet and exit of relative flow angle, relative Mach number, and effective blockage, at two spanwise positions (15% span from the hub, triangles, and mid-span, circles). These figures allow a number of observations regarding both the test case and the throughflow model.

To begin with, the flow angle in Figure 7.10a illustrates the extremely high turning of about 135 deg (between about +70 deg and –65 deg). It also shows the effect of distributing linearly between LE and TE $\tan \beta$ rather than the flow angle itself, namely a high rate of change at the center ($\beta \approx 0$), and low rate of change at ends of the blade passage ($\beta \approx 60$ deg), which in a first approximation yields a uniform distribution of swirl velocity and therefore blade loading, without the need to tune the exponent of the power function distribution. Mild discontinuities are observed at the trailing edges of both stators, but not, or to a lesser extent, at the rotor trailing edges. A possible explanation for this might be sought in the stabilizing effect exerted by the conical widening of the flow path immediately downstream of the stator trailing edges. (Shocks are stable only in diverging ducts.)

On the other hand, the evolution of Mach number in Figure 7.10b is an example for the fact that discontinuities captured at the blade edges need not be shocks: at the TE of the distributor, the Mach number actually increases (at the hub from about 1.65 to about 1.85). First and foremost, however, this figure is an impressive illustration of the fact that with the proposed Euler model, throughflow calculations can now be performed for supersonic flows, both relatively and axially. The highest Mach numbers (of up to 2.2) are observed in stator 2, which also experiences the largest spanwise variations.

In compressible flow, losses and the concomitant gradients in total pressure act as driving potential in the same way as area variation. (See, for example, Zucrow and Hoffman, 1976, for an exhaustive discussion.) In ordinary sub- and transonic turbine and compressor applications, this effect is negligible for all practical purposes, and in particular for working with the effective blockage. (Strictly speaking, losses would have to be taken into account when working with the effective blockage, but so would streamtube area variation, which cannot be known a priori. The effective blockage thus is an approximate 2D concept, but was found extremely helpful in controlling Euler throughflow solutions.) In the present case, however, the imposed losses are high enough to cause a perceptible shift in the location of the effective throat. This effect is clearly visible in the distributor, where $M = 1$ does not occur at the geometrical throat as seen by the throughflow calculation, but some way downstream in the diverging part of the nozzle, compare the Mach number and effective blockage distributions in respectively Figures 7.10b and c.

In rotor 1 and stage 2 discrepancies between the assumed LE flow angle (spanwise constant) and actual incoming flow angles imply a deviation from the imposed linear distribution, which is smoothly reduced to zero as explained above.

In the inter-row spaces, the flow is supersonic in both the absolute and relative systems, but axially subsonic. In the diverging ducts which connect each stator to its respective rotor, the flow should therefore decelerate, and this is indeed what one observes, both on the isolines of Mach number in Figure 7.8b and on the streamwise evolution of Figure 7.10b.
The global performance of the turbine predicted by the throughflow calculation is realistic and comes very close to the actual values, as far as they are known to the author. (For reasons of industrial confidentiality, only limited information was released, and references cannot be given.) The mass flow of 4.1 kg/s is fixed by the distributor and hence can be matched exactly by imposing the correct width of the choked throat. In the absence of detailed information on the blade geometry, this was in the present case done by means of four manual iterations. With a value of 29.77, the static pressure ratio is very well predicted, which is to be expected, since the exit static pressure is imposed and the correct mass flow implies the correct inlet static pressure. The total pressure ratio, on the other hand, depends on the internal flow and with a value of 12.46 is minimally over-estimated, as is the shaft power, predicted to be 5.501 MW. As an internal consistency check, power was calculated from both total enthalpy and swirl, separately for each rotor, finding perfect agreement between the two formulations. This mechanical output corresponds to an efficiency of 0.391 (isentropic) or 0.314 (polytropic).

Figure 7.8  Supersonic impulse turbine: (a) computational grid (16 x 192 = 3072 cells), (b) isolines of relative Mach number (range from 0 to 2.2, increment 0.05), (c) meridional streamlines.
Figure 7.9  Supersonic impulse turbine: convergence history (solid line = density, dashed line = tangential momentum, symbols = blade force).
Figure 7.10  Supersonic impulse turbine: streamwise evolution of (a) relative flow angle ($\beta = \arctan \frac{W_\theta}{V_z}$), (b) relative Mach number ($M_{rel}$), and (c) effective blockage ($b_{eff} = b \cos\beta$) at 15% span (triangles) and at mid-span (circles).
7.4 Bypass Turbofan Engine

One decisive advantage of the Euler throughflow model is the possibility to directly impose as boundary conditions at inlet and exit of the calculated domain variables which would be controlled in an analogous experiment, or which are known from the preliminary design phase and characterize operation under certain conditions, for example, cruising at a certain altitude and Mach number in the case of an aeronautical jet engine. Typically, these are the total pressure and total temperature at the inlet and the static pressure at the outlet. The massflow, which had to be imposed in the streamline curvature and matrix throughflow methods, is a result of the calculation. For bypass configurations, this means that the distribution of the inlet massflow onto the bypass and engine core is controlled by the respective back pressures and therefore accessible to direct prediction.

This unique capability of the Euler throughflow methodology is demonstrated with a representative example of a civil transport jet engine. The engine of the 50 kN thrust class has a fan diameter of about 1.2 m and a bypass ratio of 7 (which is a high value for such a relatively small engine; current production engines in this class typically have bypass ratios around 5). Engines of this size power regional jets with a seating capacity between 40 and 80 in a tail-mounted twin-engine configuration.

The throughflow calculations presented here model the forward part of the engine’s compression system which comprises the fan, a bypass OGV, a bypass strut, and the front part of the core engine consisting of an IGV, a two stage booster compressor with tandem blades in the rotors, and a strut. Table 7.1 gives an overview of the different elements, with an indication of the number of blades and of the mesh size. For a global view of the modelled part of the engine we refer forward to Figure 7.15a, in which the blade rows are visualized through their respective mesh. Comparing with Table 7.1, we observe that the number of blades in each row is inversely proportional to the axial chord of the blade, thus ensuring a reasonable value of the solidity for all rows.

The results discussed in the following were obtained on a relatively fine mesh of 24 651 grid points, chosen out of a concern of the engine’s designers to minimize numerical dissipation and to get a detailed picture of the flow at the leading edge of the divider between the core and bypass flow paths. Figure 7.11 shows this mesh. (Only every other mesh line is drawn for clarity.) Each blade row occupies one block divided into an upstream region, the blade passage, and a downstream region, with the exception of the two core rotors for which each of the two tandem blades is modelled independently and therefore assigned a block of its own, without however any upstream and downstream regions. (The idea behind tandem blades is to achieve higher diffusion factors by re-initializing the suction side boundary layer and by reducing secondary flows.) The grid point distribution is essentially uniform, with mild refinement at the edges of some blades dictated by the small inter-row gaps in the core and the desire to allow at least four levels of multigrid. In the fan, the mesh is somewhat denser in the rear part where the captured shock is expected at design operation. The most conspicuous aspect of the mesh arguably is the radial redistribution of grid points downstream of the fan. In the Euler throughflow model, contrary to the streamline curvature method, there is no intrinsic link between streamlines and streamwise mesh lines, which may therefore be distributed radially according to other imperatives. In the present application this degree of freedom is exploited to (i) cluster the mesh around the splitter leading edge and (ii) distribute the equi-spaced mesh lines from the fan between core and bypass in a ratio which ensures comparable spanwise resolution but which does not reflect the absolute heights of the respective flow paths. (An equivalent effect could be obtained with the non-matching connecting boundary condition presented in section XX, although selective refinement at the splitter leading edge would then require the insertion of additional blocks.) A conventional arrangement with streamwise mesh lines as quasi-streamlines is retained inside blade rows where streamwise distributions are applied along mesh lines and where the latter may serve as a fixed reference direction for the blade-to-blade flow angle.

All geometrical and operational data, including empirical estimates for losses and deviation, were provided by the engine’s designers and accepted unquestioningly for the purposes of this demonstration of the Euler throughflow methodology. The design operating point represents steady flight at Mach 0.816 at an altitude of 9307 m, which corresponds to an inlet stagnation pressure of 45 575 Pa and an inlet total temperature of 257.98 K (based on the standard atmosphere). The shaft speed is
6436 rpm and the total mass flow 99.81 kg/s, which is also the choking mass flow of the transonic fan. The back pressures at the exits of the modelled parts of the bypass and core are respectively 56 072 Pa and 70 713 Pa.

The calculations presented here are classical analysis calculations, meaning that the flow angle is imposed at the trailing edge only; inside the blade passage, it follows the imposed streamwise distribution but is allowed to vary with the flow angle at the leading edge, which is a result of the calculation.

Figure 7.13 shows the spanwise profiles of the loss coefficient imposed at the trailing edge of five selected blade rows (the fan, the bypass OGV, the front and rear blades of rotor 1, and stator 1), defined with respect to the relative stagnation pressure, $\Psi_{rel}$. Losses range between 3% and 5% at mid-span and in rotor 1 increase to values of the order of 10% towards the end-walls. The reason for the absence of a similar increase in the stators has not been communicated by the engine’s designers, nor has the difference between the bypass OGV and stator 1 (nearly 4 percentage points, a factor of two at mid-span). At the root of the fan, the estimated losses attain 20%, due to the large blade thickness (37% of pitch) and very high turning (from 40 deg to near-axial). The 15% loss at the fan tip does not include the shock loss, which will be captured by the Euler throughflow analysis calculation. Tangential blockage factors in the other blade rows range from 0.8 to 0.95, Figure 7.12.

Transonic fans operate at or very close to choked conditions. To perform meaningful calculations of the considered engine, it is therefore essential that the choking mass flow of the fan as seen by the throughflow model can be controlled reliably. The turbine test cases presented in the preceding sections have shown that one possible way of doing so is to use the geometrical throat width as a blade modelling parameter, relinquishing direct control of the blade thickness. An equally practicable alternative for transonic fans (which have only low turning in the outboard, supersonic sections) is to make realistic choices for the secondary traditional blade modelling parameters (the primary parameters being the spanwise distributions of blade thickness, exit flow angle, and loss coefficient). In the present case, these are the location of the maximum blade thickness (at 0.69 axial chords) and the exponents of the streamwise power function distributions of blade thickness (1.5) and flow angle (2). For comparison, the default values of these three parameters, used for all other blade rows ($d_{max}$ at 0.5 c, thickness and flow angle exponents of respectively 2 and 1), yield a choking mass flow of only 80.1 kg/s.

As a parenthesis, these considerations revive the more general question of what information on the blade geometry is most relevant for throughflow calculations. From the experience accumulated with the present Euler throughflow model, it would appear that for trans- and supersonic blade rows, the most realistic flow and shock patterns are obtained by controlling the throat width, either directly or through a collocated description of blade thickness and flow angle, and by a monotonous streamwise variation of effective blockage. A faithful reproduction of the actual geometrical blade thickness and thus tangential blockage distribution seems least important, and in some cases even detrimental to throughflow solution accuracy.

Standard numerical settings are used for this test case (central scheme with second and fourth order dissipation coefficients of respectively 1 and 0.1, 4-stage Runge-Kutta time integration with time-step dependent implicit residual smoothing, 4-level multigrid in a V cycle with 1, 2, 8, and 32 smoothing sweeps on respectively grid levels 1, 2, 3, and 4). The inlet boundary condition imposes total pressure and temperature, and the flow direction (axial), axial velocity is extrapolated. At the two exits, constant values of static pressure are imposed, the absolute velocity vector and absolute total enthalpy are extrapolated. Calculations start directly on the finest grid from a uniform initial solution (static pressure 55 kPa, static temperature 260 K, purely axial velocity of 100 m/s). The CFL number is 5, the time step-like coefficient $\kappa$ in the blade force equation is 1. During the first ten iterations, $\kappa$ is reduced to 0.1 and the target flow angles to 85% of their final value.

Figure 7.14 shows the resulting convergence histories. Machine accuracy is reached in 250 iterations for deeply choked flow (first curve, exit pressures of 50 kPa in the core and 40 kPa in the bypass), and in 750 iterations at the design point (second curve). With the described choices for the secondary blade modelling parameters, the envisaged choking mass flow of 99.8 kg/s is obtained exactly.

Figure 7.15 gives a first global impression of the predicted flow. Meridional streamlines are
perfectly attached to all end walls, and the core and bypass appear to be well matched to the fan and to each other, Figure 7.15b. In particular, the flow is well attached to the splitter, Figure 7.15e. The leading edge of the splitter was of major concern to the engine’s designers because spill-over either from the core into the bypass or vice versa with ensuing flow separation along the hub of the bypass or along the casing of the core due a large angle of attack would degrade the performance of respectively the bypass OGV or the core IGV. Even though the actual flow is of course circumferentially non-uniform due to the wakes and secondary flows generated by the fan, the Euler throughflow model (and only it) does provide valuable information in the early stages of design when the blade number and precise blade geometry are still undetermined. Figure 7.15c suggests that the stagnation point is shifted slightly towards the bypass, leading to mild over-speeds on the core side. The severity of this situation and the risk of leading edge separation could be assessed by performing a viscous throughflow calculation. Before relying on results of such a simulation, however, it would be advisable to verify the accuracy of the viscous meridional solution through comparison with pitch-averaged 3D solutions for similar cases.

In the axisymmetric throughflow solution, supersonic relative flow occurs only in the fan. Of all the other blade rows, it is the the front blades of the first tandem rotor in the core which experience the highest Mach numbers, reaching values of 0.86. Predicted inlet relative Mach numbers of the fan range from 0.69 at the hub to 1.42 at the casing. The bold isoline in Figure 7.15c marks Mach 1. Deceleration and hence work input almost exclusively occur in the rear part of the fan, where the blade-to-blade passage described by the imposed blade thickness and flow angle distributions forms a diverging channel. In the supersonic sections along the outer 60% of span, this involves a quasi-normal shock. Between 40% and 75% span, this captured shock is entirely contained within the blade passage, while it coincides with the blade trailing edge between 75% and 100% span and is thus only one part of the more general solution of the Riemann problem between the blade passage (analysis mode) and the following duct (design mode). On the outermost 5% span, the exit relative flow even remains supersonic.

The captured shock in the fan most clearly appears in the entropy contours of Figure 7.15d. Otherwise, entropy increases smoothly across blade rows according to the imposed losses. In the ducts between blade rows, the flow is isentropic (apart from the effect of numerical dissipation) and entropy contours therefore are streamlines. Increases range between $\Delta s = 10.4$ to 42.2 J/kg K in the bypass and between 28.1 to 47.4 J/kg K in the core, with the higher values occurring near the end walls.

At the design point, the Euler throughflow calculation predicts a bypass ratio of 6.7. The single transonic stage formed by the outboard part of the fan and the bypass OGV and strut achieves a total pressure ratio of 1.51 at a polytropic efficiency of 84.75%, while the subsonic stage formed by the inboard part of the fan, the core IGV, and the two stages of the booster compressor plus strut produce an overall total pressure ratio of 2.16, corresponding to an average per-stage pressure ratio of 1.29, at a polytropic efficiency of 86.70%.

Figure 7.16 presents a synoptic view of color contours for six relevant variables, relative Mach number (a), absolute Mach number (b), static pressure (c), absolute total pressure (d), relative flow angle (e), and entropy (f), allowing additional observations. Upstream of the engine nacelle and in the intake duct, the flow must decelerate from the free stream Mach number of 0.816 to values between 0.445 and 0.536 predicted at the inlet of the throughflow computational domain and corresponding to static pressure ratios between 1.352 and 1.274. Fan total pressure ratio as predicted by the present throughflow calculation monotonously increases from 1.34 at the hub to 1.64 at about 80% span and then decreases again to values around 1.35 at the shroud. It is also at the casing between the fan trailing edge and bypass OGV leading edge that the largest flow angles occur, both relative (75 deg) and absolute (55 deg), accompanied by diverging meridional streamlines and low absolute Mach numbers, Figures 7.15b and c. Finally, the change in radius of the core flow at constant flow path height approximately doubles the Mach number, from 0.35 downstream of stator 2 to 0.7 at the exit of the core computational domain.

The ability of the Euler throughflow model to predict the distribution of the inlet mass flow on the core and on the bypass is further illustrated with a mistuned operating point defined by bypass and core exit pressures of respectively 70 kPa and 60 kPa. Although this calculation does not con-
verge to machine accuracy but enters into a limit cycle, Figure 7.17, the solution is sufficiently stabilized to allow a meaningful interpretation (mass flow fluctuations less than ±0.3%). The total predicted mass flow is 89.1 kg/s, with a bypass ratio of only 5.6. As a consequence, the core has to capture its mass flow from a larger portion of the total area, Figure 7.12a, leading to high incidence at the splitter and subsequent separation in the core, Figure 7.12b. It is the interaction of these vortices with the dynamic blade force which causes the limit cycle. A detailed view of the velocity field around the nose of the splitter is given in Figure 7.18.

Table 7.1: Modelled elements of the bypass turbofan engine with number of blades and mesh size (total number of grid points 24 651).

<table>
<thead>
<tr>
<th>Block number</th>
<th>Element</th>
<th>Number of blades</th>
<th>Mesh size I x J</th>
<th>Number of grid points</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Fan</td>
<td>33</td>
<td>81 x 89</td>
<td>7209</td>
</tr>
<tr>
<td>10</td>
<td>Bypass OGV</td>
<td>85</td>
<td>49 x 97</td>
<td>4753</td>
</tr>
<tr>
<td>11</td>
<td>Bypass strut</td>
<td>10</td>
<td>49 x 65</td>
<td>3185</td>
</tr>
<tr>
<td>2</td>
<td>Core IGV</td>
<td>122</td>
<td>33 x 73</td>
<td>2409</td>
</tr>
<tr>
<td>3</td>
<td>Core rotor 1 front</td>
<td>78</td>
<td>33 x 17</td>
<td>561</td>
</tr>
<tr>
<td>4</td>
<td>Core rotor 1 rear</td>
<td>78</td>
<td>33 x 17</td>
<td>561</td>
</tr>
<tr>
<td>5</td>
<td>Core stator 1</td>
<td>135</td>
<td>33 x 41</td>
<td>1353</td>
</tr>
<tr>
<td>6</td>
<td>Core rotor 2 front</td>
<td>72</td>
<td>33 x 17</td>
<td>561</td>
</tr>
<tr>
<td>7</td>
<td>Core rotor 2 rear</td>
<td>72</td>
<td>33 x 17</td>
<td>561</td>
</tr>
<tr>
<td>8</td>
<td>Core stator 2</td>
<td>74</td>
<td>33 x 41</td>
<td>1353</td>
</tr>
<tr>
<td>9</td>
<td>Core strut</td>
<td>10</td>
<td>33 x 65</td>
<td>2145</td>
</tr>
</tbody>
</table>

Figure 7.11 Bypass turbofan engine: throughflow mesh (24 651 grid points, every other grid line shown).
Figure 7.12  Bypass turbofan engine: tangential blade blockage.

Figure 7.13  Bypass turbofan engine: imposed trailing edge profiles of the loss coefficient for five selected blade rows.
Figure 7.14  Bypass turbofan engine: convergence histories of (a) the density residual and (b) the inlet and exit mass flows for the design operating point.

Figure 7.17  Bypass turbofan engine: convergence histories of (a) the density residual and (b) the inlet and exit mass flows for mistuned operation (increased core mass flow with casing separation).
Figure 7.15 Bypass turbofan engine, design point operation: (a) calculated configuration, (b) meridional streamlines, isolines of (c) relative Mach number and (d) entropy, (e) velocity field at the splitter leading edge.
Figure 7.16   Bypass turbofan engine, design point operation: color contours of (a) relative Mach number, (b) absolute Mach number, (c) static pressure, (d) absolute total pressure, (e) relative flow angle, (f) entropy.
Figure 7.18  Bypass turbofan engine: streamline pattern for mistuned operation with increased core mass flow: (a) global view, (b) detail at the splitter leading edge and in the engine core.
7.5 Conclusions

A 3D Navier-Stokes solver has been extended to include a throughflow model based on the Euler equations. Blades are described through distributions and profiles for blade thickness, flow angle or swirl, and losses. The efficiency of the multigrid-accelerated Euler throughflow algorithm with a time-dependent blade force was demonstrated for multi-stage machines. It allows the use of an Euler through-flow solver in the design cycle at acceptable turn-around times of a few minutes. The algorithm is equally effective for subsonic and for supersonic flows. An investigation of the shock capturing properties of the Euler throughflow equations showed that quasi-normal shocks are captured in analysis mode and axisymmetric shocks in design mode. The implications for the important class of applications of transonic compressor rotors have been pointed out. The choking massflow in analysis mode was found to be highly sensitive to the distribution of thickness and, especially, flow angle across the blade. Analysis of a 3D Navier-Stokes solution in the meridional (pitchwise) average shows that the actual evolution of the flow angle is well described by simple power functions. The throughflow model may thus serve as a tool to predict the effect of design changes in those parameters. Although due to the captured shock the analysis mode flow field can differ from the design mode flow field, which more closely resembles the pitch-averaged 3D flow field, the differences downstream of the blade row appear to be small.
Chapter 8  Condensable Fluids

The 3D Navier-Stokes solver Euranus, representative of contemporary CFD technology, has been equipped with a tabular thermodynamic properties formulation based on reference equations of state. Any required thermodynamic variables are directly interpolated from dedicated tables. Compared with the perfect gas, the new method degrades neither robustness nor convergence rate and results in only a minor increase in CPU time (between 5% and 30%). This chapter first presents general tabular properties implementation, Section 8.1. The performance of the new method is then illustrated for two concrete fluids, namely steam in Section 8.2 and liquid hydrogen (LH2) in Section 8.3.

8.1 Thermodynamic Interpolation Tables

8.1.1 Methodology

In the non-chemistry version, the working fluid is a pure substance and its thermodynamic state therefore defined by two variables. In the conservative formulation employed by Euranus, the density and internal energy emerge as a natural choice, because these are the variables that are first and mandatorily obtained when the solution is updated at each Runge-Kutta stage. The evaluation of the fluxes, on the other hand, also involves the pressure in the momentum and energy equations and the temperature in the energy equation. For a perfect gas, both are readily calculated from the state equation; the conversion between different pairs of independent thermodynamic variables is in fact so facile that only the pressure and density are stored and other variables recalculated from these whenever they are needed.

For a real fluid, on the other hand, the equation of state may be a complicated expression and given implicitly, adding complexity in the code and generating unacceptable computational overhead, as has been illustrated in the introduction at the example of LH2. Similarly, when an equilibrium mixture of vapor and droplets (wet steam, for instance) is considered as a single fluid, the calculation from the saturation properties of thermodynamic variables corresponding to a given density and internal energy must be done iteratively. In addition to the state equation for the vapor phase, a saturation table or equation would also be required.

Addressing these two problems, an option has been added to Euranus in which the classical concept of a state equation, in the sense of one equation directly implanted into the code, is abandoned in favor of the interpolation of all needed thermodynamic variables from dedicated tables. Concretely, the density, $\rho$, and internal energy, $e$, are chosen as the stored dependent variables (in addition to the three velocity components). The pressure, $p$, and the temperature, $T$, are then interpolated from $p(e, \rho)$ and $T(e, \rho)$ tables, respectively. The speed of sound, $c$, which enters the numerical solution process through the spectral radius in the time step and in the residual smoothing and artificial dissipation coefficients, is also required. It is interpolated from a $c(e, \rho)$ table. Additional tables may be required to apply the inlet and exit boundary conditions. The tables are generated outside and independently of the solver, and loaded from files when a computation is started.

Careful optimization of the interpolation technique and the strategic choice of interpolated and dependent thermodynamic variables have lead to an efficient method. Depending on the selected options and test cases, overheads between 10% and 30% have been measured in steam applications, with respect to the perfect gas. Compared with the direct implementation of a complex equation of state, the tabular fluid properties thus offer substantial savings. To treat other fluids, the according interpolation tables must be generated, either from an equation of state or from tabular data. In the present project, this task is undertaken for liquid hydrogen.

8.1.2 Implementation

The interpolation tables required for a basic implementation of the above described concept can be read directly from the governing equation, e.g., in semi-discretized form,
Examination of Equations (8.1) and (8.2) shows that in addition to the internal energy and the density, which are obtained directly when the solution is updated at each new time level, the pressure is the only other thermodynamic variable that appears in the inviscid part of the equations. Hence, a single table giving pressure in function of internal energy and density,

\[ p = p(e, \rho) \quad (8.3) \]

would in principle be sufficient to perform Euler calculations. Navier-Stokes calculations additionally require the temperature, again in function of energy and pressure, and also the transport coefficients thermal conductivity and dynamic viscosity, if tables are to be used for these,

\[ T = T(e, \rho) \quad \mu = \mu(e, \rho) \quad k = k(e, \rho) \quad (8.4) \]

It must be emphasized, however, that the molecular transport properties of a substance are in no way directly related to its thermodynamic properties. Separate constituent laws are required, which have to be determined in dedicated experiments, and the philosophy behind the formulation of the resulting empirical equations as well as the structure of typical resulting equations both differ from those employed for the thermodynamic properties.

While the tables listed in (8.3) and (8.4) above are all that is required for a complete physical description of the flow inside the computational domain, the boundary conditions usually involve additional thermodynamic variables, or the same basic variables with different dependencies (e.g., \( p(e, \rho) \)) or in different combinations (e.g., \( p(T, \rho) \)). Although it would in principle be possible to iteratively invert tables as required during the numerical solution process, and even reconstruct certain other thermodynamic variables from the basic tables, all in a thermodynamically consistent manner, one would thereby lose two of the principal advantages of the tables, namely their efficiency and robustness. Additional tables are therefore required to apply the boundary conditions, and in the present work the following have been used:

\[ p(h, s) \quad p(h, s) \quad s(h, p) \quad p(e, \rho) \quad e(T, \rho) \quad (8.5) \]

While the physical description of the flow problem is now complete, the time-marching procedure for the numerical solution of Equation (8.1) relies heavily on the wave propagation properties, which inevitably involve the speed of sound,

\[ c^2 = \left( \frac{\partial p}{\partial \rho} \right)_s \quad (8.6) \]

It enters the solution process through the eigenvalues of the flux Jacobian and its spectral radius (in the time step, the artificial or upwind dissipation, and implicit residual smoothing) and is either interpolated from a dedicated table, for example in function of internal energy and density,

\[ c = c(e, \rho) \quad (8.7) \]

or calculated from the partial derivatives of the \( p(e, \rho) \) table,

\[ c^2 = \left( \frac{\partial p}{\partial \rho} \right)_e + \frac{p}{\rho^2} \left( \frac{\partial \rho}{\partial e} \right)_p \quad (8.8) \]
No additional tables are required to set the wall boundary conditions. At Euler walls, \( \rho_E \) is extrapolated with zero or first order. Its only influence on the solution is through the artificial dissipation of the energy equation. At Navier-Stokes walls, both \( \rho \) and \( \rho_E \) are extrapolated with zero order. The temperature in the dummy cell is set from the condition of a zero normal gradient for adiabatic walls.

In multigrid, the set of variables that is restricted and prolongated is

\[
(\rho, \hat{\rho}, \rho_E)
\]

The pressure and the temperature are then recalculated according to (8.3) and (8.4).

Two inlet and one exit boundary condition applicable to tabular condensable fluids are listed in Table 8.1. An (in the present work) commonly used inlet condition for steam flows imposes total enthalpy; entropy and two flow angles, the axial velocity component is extrapolated with zero or first order. An older option, limited to wet steam and based on a pressure saturation table, imposes the dryness fraction instead of entropy and extrapolates the pressure. In addition to the tables already listed above, it requires a pressure saturation table which in our implementation contains the variables

\[
T, \rho', \rho'', h', h'', s', s'', e', e''
\]

in function of \( p \) (8.9)

where a prime marks the saturated liquid value and a double prime the saturated vapor value.

### Table 8.1: Inlet and exit boundary conditions applicable to condensable fluids.

<table>
<thead>
<tr>
<th>Type</th>
<th>ID #</th>
<th>Imposed</th>
<th>Extrapolated</th>
</tr>
</thead>
<tbody>
<tr>
<td>subsonic inlet</td>
<td>1</td>
<td>H, x, dir(( \hat{\rho} ))</td>
<td>( p )</td>
</tr>
<tr>
<td>subsonic inlet</td>
<td>2</td>
<td>H, s, dir(( \hat{\rho} ))</td>
<td>( v_z )</td>
</tr>
<tr>
<td>subsonic exit</td>
<td>3</td>
<td>( p )</td>
<td>H, ( \hat{\rho} )</td>
</tr>
</tbody>
</table>

As an example for applying boundary conditions with tabular condensable fluids, the schema below describes in detail the entropy-based inlet boundary condition (ID # 2), indicating the sequence of operations, where they are performed (on the boundary cell face or in the dummy cell center) and the tables involved:

\[
\hat{\rho} = \text{function of } v_z \text{ and } \text{dir}(\hat{\rho})
\]

\[
\begin{align*}
    h &= H - \frac{\hat{\rho}^2}{2} \\
    \rho &= \rho(h, s) \\
    p &= p(h, s) \\
    \rho E &= \rho H - p
\end{align*}
\]

on the boundary

\[
T = T(\rho, e)
\]
in the dummy cell

For a subsonic exit boundary condition imposing the pressure and extrapolating (with zero or first order) the total enthalpy and the velocity vector (ID # 3 in Table 8.1), the sequence of operations would be the following:
The dynamic viscosity and thermal conductivity can be interpolated from tables in function of \( e \) and \( \rho \). The possibility is provided to use a variable \( c_p \) in the turbulent thermal conductivity, \[ k_t = \frac{\mu_{t} c_p}{Pr_t} \] where, for example, \( c_p = c_p(e, \rho) \). We note, however, that \[ c_p = \frac{\partial h}{\partial T} \rho \] is undefined in the mixture region, because \( p \) and \( T \) are not independent there. The behavior near the saturation lines should be continuous with that of the dominating single phase. A simple solution therefore is to proceed as for the molecular viscosity, \( \mu \), and thermal conductivity, \( k \), and use an average of the saturated liquid and vapor values. The type of average remains to be chosen. (The analogy with \( \mu \) and \( k \) suggests a volume weighted average. On the other hand, the heat capacity is a mass specific quantity, which would suggest a mass weighted average.) Equation (1) is the only place where \( c_p \) enters into the computation.

The speed of sound is either interpolated directly from a \( c(e, \rho) \) table or calculated consistently from the \( p(e, \rho) \) table (in which case a speed of sound table is not required). The void fraction \( \alpha \) is interpolated from a dedicated \( \alpha(e, \rho) \) table.

The initial solution is composed of two parts: the three velocity components and two thermodynamic variables. Uniform initial solutions in terms of static enthalpy and entropy, pressure and density, and internal energy and density have been used successfully.

### 8.1.3 Interpolation and Table Types

Experience has been gathered with two interpolation types and two types of tables. The two interpolation types are bilinear and bicubic. Bicubic tables are constructed by passing cubic splines through a set of data points on an orthogonal mesh.

The two types of tables accepted by the EURANUS are single and double tables. Figure 8.1 illustrates the concept, taking as an example the \( p(e, \rho) \) table. The thermodynamic surface is assembled from two parts: the stable single phase region above the saturation line, and a synthetic surface representing the properties of the equilibrium two-phase mixture of saturated vapor and liquid below. By definition, these two surfaces match up at the saturation line, but they are not slope-continuous. An isochore in the p,e-diagram, which one may then think of as a cut through the \( p(e, \rho) \) surface at some constant density, therefore has a kink where it crosses the saturation line, Fig. 8.1a.

In a so-called single table, no distinction is made between the mixture and single phase regions; the compound true thermodynamic surface is represented by a single interpolating surface (the ‘table’). As a consequence, the sharp transition between the two regions is smeared out. Figure 8.1b schematically shows the result for a piece-wise bilinear table. It is seen that cells which bridge the slope discontinuity at the saturation line will in general have errors that are much larger than in the smooth regions. Bicubic tables should be avoided in this case because slope discontinuities tend to introduce oscillations in the spline functions, which may cause large, uncontrolled errors, not only at the saturation line but also away from it.
To avoid this problem, the concept of a double table was created. Here, each of the two regions (single phase and mixture) is represented by its own interpolating surface. A double table thus actually contains two tables: one, \( p_1(e, \rho) \), for the single phase region, and another, \( p_2(e, \rho) \), for the mixture region. Although each of these covers the whole range of independent variables, valid data is only required in the respective region, and in a narrow zone of overlap along the saturation line. Additional information, also contained in the table, tells the code which surface, \( p_1 \) or \( p_2 \), to use for a given pair of independent variables. The result is a clean transition, as shown in Fig. 8.1c.

The generation of a double table requires that the data can be split into the two regions exactly, or that the data are provided separately for each. Missing data in the respective other region (where the two surfaces overlap along the saturation line) can be generated by extrapolation.

### 8.1.4 Combinations of Variables

An summary of some typical interpolation tables and their use in Euranus is given in Table 8.1. To enable a flexible table management system, a separate table is made for each dependent variable. Each table is known to Euranus by a three-letter code, composed of the first letters of respectively the dependent variable and the two independent variables, in this order. In the few cases where this scheme leads to conflicts, priority is given to the more common variable, and a meaningful alternative is sought for the other. A filled circle indicates that the table is mandatory, while a light circle means that there are alternatives that use different tables or no table at all.

All tables use consistent SI units (\( p \) [Pa], \( T \) [K], \( \rho \) [kg/m\(^3\)], \( e \) [J/kg], \( c \) [m/s], \( \mu \) [Pa s], \( k \) [W/m/K]).

<table>
<thead>
<tr>
<th>Name</th>
<th>Code</th>
<th>Used for(^a)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( p(e, \rho) )</td>
<td>PER</td>
<td>● – ● o</td>
</tr>
<tr>
<td>( T(e, \rho) )</td>
<td>TER</td>
<td>– ● ● o</td>
</tr>
<tr>
<td>( c(e, \rho) )</td>
<td>CER</td>
<td>o – – o</td>
</tr>
<tr>
<td>( \rho(h, p) )</td>
<td>RHP</td>
<td>– – o –</td>
</tr>
<tr>
<td>( \rho(e, p) )</td>
<td>REP</td>
<td>– – o –</td>
</tr>
<tr>
<td>( e(T, p) )</td>
<td>ETR</td>
<td>– – o –</td>
</tr>
<tr>
<td>( \mu(e, \rho) )</td>
<td>MER</td>
<td>– o – o</td>
</tr>
<tr>
<td>( k(e, \rho) )</td>
<td>KER</td>
<td>– o – o</td>
</tr>
<tr>
<td>( \alpha(e, \rho) )</td>
<td>AER</td>
<td>– – – o</td>
</tr>
</tbody>
</table>

\(^a\) IF = inviscid fluxes and numerical algorithm, VF = viscous fluxes, BC = boundary conditions, CFV = CFView output
Figure 8.1 The concept of single and double tables illustrated for the \( p(e, \rho) \) thermodynamic surface: (a) cut through the surface at constant density, (b) approximation of the true thermodynamic surface by a single bilinear surface (single table), (c) approximation by two overlapping bilinear surfaces (double table).
8.2 Application to Steam

In the context of steam flows, and in particular the economically most important application to steam turbines, the proposed tabular thermodynamic properties formulation allows to address three issues simultaneously. First, a (perhaps definitive) answer is given to the problem of how to best account for the real gas properties of steam in a Navier-Stokes code. Second, equilibrium wet steam is accessible in exactly the same way as dry steam. Third, spontaneous condensation (condensation shock) can be modelled algebraically by switching between dry and equilibrium wet steam at the correct level of subcooling. This section presents the required tables, with indications on their structure, size, accuracy and effective use in the numerical solution procedure. A throughflow calculation of a condensing four-stage low-pressure turbine serves to illustrate the benefits of the basic method. Features of the switched condensation model are examined for supersonic nozzle flow and refinements proposed in the form of a convected switch and a two-stage transition process.

8.2.1 Introduction

Steam turbines are one of the most economically important turbomachinery applications. Until recently, however, numerical flow simulations of steam turbine components usually used the perfect gas assumption, especially in industrial design practice. This assumption is inaccurate near the saturation line, for wet steam and for expansion processes with high pressure ratio. The latter occurs if several stages are calculated simultaneously, e.g., in throughflow calculations, but increasingly also in 3D. Where the increased cost was acceptable, for instance for research work, various truncated virial equations of state (Young, 1988; Singh, 1999; Chmielniak et al., 1999) or empirical compressibility correction factors (Young, 1992; White and Young, 1993) have also been employed. The principal drawbacks of these formulations are the limited range of validity and, in the case of those valid also at higher pressures, the high computational cost, iterative inversion of the state equation being frequently required.

Further, in the case of wet steam, for which the perfect gas assumption is clearly inadequate, the effort required for the detailed modelling of nucleation and condensation is excessive for most practical applications. In many cases it is also unjustified, for instance if only the global thermodynamic behaviour is of interest or in the steady state analysis of turbomachinery blade passages, where in reality flow unsteadiness plays a dominant role (Gyarmathy, 1976; Guha, 1998). Downstream of the initial spontaneous condensation, the two-phase mixture is close to thermodynamic equilibrium and slip between the fine fog and the vapour is negligible (Gyarmathy, 1976), except behind shocks. However, even the simple alternative of treating wet steam as a homogeneous fluid with the properties of the equilibrium two-phase mixture used to increase computing times considerably because it requires the iterative evaluation of state equation and saturation data. The lack of efficient, accurate and generally accepted dry and wet steam formulations for CFD codes also hampers validation efforts when comparison with actual steam data is required (e.g., Dejean et al., 1999).

When comparing the various possibilities of modelling dry and wet steam with regard to range of validity, accuracy, and speed of evaluation, direct interpolation of the required thermodynamic variables from tables seems to offer the best compromise. Memory requirement for storing the tables is small compared with that for a typical 3D Navier-Stokes computation and therefore not a criterion any more, as it might have been a few years ago.

In this section, we show how spontaneous condensation can be modelled algebraically on the basis of tabular properties for dry and wet steam, Section XX. Results are then presented for two test cases, a four-stage low-pressure turbine illustrating the basic equilibrium wet steam implementation, Section XX, and a Laval nozzle demonstrating the potential of the switched condensation model, Section XX.

8.2.2 Steam Tables and Steam Properties Formulation

The steam tables are based on the current reference IAPWS-95 (IAPWS, 1996; Wagner and Pruss, 1997; Span and Wagner, 1997; Harvey et al., 1997; Wagner and Pruss, 2002) which describes the thermodynamic properties of fluid water substance through a single fundamental equation for the
Helmholtz free energy in function of temperature and density. Figure 8.2 shows the composite stable \( p(e, \rho) \) surface calculated from this formulation. Relevant states for steam turbine applications are contained within a band of about 1 000 kJ/kg around the vapour saturation line. An initial wish to have smooth, continuous, differentiable tables led to the selection of bicubic interpolation. Systematic checks on a 30 x 30 \( p(e, \rho) \) table covering states from the triple line to supercritical indicate an accuracy better than 1% and an RMS error below 0.1%. Memory requirement is about 400 kB per table in ASCII format. As an example, the \( T(e, \rho) \) table is shown in Figure 8.3. Every fifth grid line delimits a bicubic table patch, the four intermediate lines are interpolated. Note the sharp intersection between the stable single-phase and equilibrium two-phase mixture surfaces. Table accuracy improves rapidly with resolution and can be increased arbitrarily. Finer tables do not entail any measurable slowdown.

### 8.2.3 Switched Condensation Model

Compared with perfect gas or dry steam computations, the single-fluid equilibrium two-phase model of wet steam flow represents a major improvement. It lacks, however, one important feature of initially dry condensing flows, namely the fact that, depending on the pressure and expansion rate, a significant level of subcooling may be reached before condensation spontaneously occurs (condensation shock).

To describe the nucleation and condensation processes, traditional Eulerian condensation models require at least two (e.g., McCallum and Hunt, 1999) and more commonly four additional equations (Hill, 1966; Sejna, 1993; Chmielniak et al., 1999; Stastny and Sejna, 1999) with stiff source terms, resulting in considerably longer computing times than in the absence of condensation. Lagrangian models require an iterative procedure between the Lagrangian condensation module and the Eulerian flow solver (Guha and Young, 1991; Young, 1992; White and Young, 1993; Singh, 1999), resulting in costs of 3 to 5 times those of a perfect gas calculation.

Adopting an idea recently put forward by Hill et al. (1999), it was therefore examined if spontaneous condensation (a non-equilibrium process) could be represented, at least qualitatively, in the context of a single-fluid equilibrium two-phase model for steam by delaying the switch from the dry to the equilibrium wet formulation until the appropriate subcooling has been reached, as illustrated in Figure 8.4. Although the IAPWS-95 formulation nominally covers only the stable fluid region, the official release (IAPWS, 1996) states that it produces reasonable results when extrapolated into the

---

Figure 8.2  Pressure in function of internal energy and density, \( p(e, \rho) \), for steam calculated from IAPWS-95 (100 x 100 data points), stable single-phase and equilibrium mixture regions.
The implications of switching from meta-stable dry to equilibrium wet steam were analysed theoretically for pressure and temperature, which are the only table-interpolated variables away from external (inlet and exit) boundaries. It was concluded that interpreting a blend between the values furnished by the two formulations as a local average over the non-equilibrium condensing mixture does not lead to thermodynamic inconsistencies.

Practically, the transition between the meta-stable and equilibrium steam properties is accomplished through a space- and time-dependent switch, \( \alpha \), that takes values between 0 (dry) and 1 (wet). In the transition zone, pressure and temperature are calculated as a blend of the subcooled dry and equilibrium wet steam properties associated with the given conservative variables (density and internal energy),

\[
\begin{align*}
    p &= (1 - \alpha) p_{\text{dry}} + \alpha p_{\text{wet}} \\
    T &= (1 - \alpha) T_{\text{dry}} + \alpha T_{\text{wet}}
\end{align*}
\]

where \( p_{\text{dry}} = p_{\text{dry}}(e, \rho) \) and \( p_{\text{wet}} = p_{\text{wet}}(e, \rho) \), with analogous expressions for \( T \). The transition location depends on the a priori unknown flow and must be linked to the current solution dynamically according to the known dependence of the limiting subcooling on the pressure and on the expansion rate. Also, in the framework of a multiblock solver, an essential demand on the condensation switch and its determination is that the procedure be independent of mesh topology. The field of the condensation switch is therefore obtained from a scalar convection-diffusion equation solved in parallel with the main flow equations,

\[
\frac{\partial \alpha}{\partial t} + (\hat{\nu} \cdot \nabla) \alpha - \nabla \cdot (k \nabla \alpha) = q(\alpha, \hat{\nu}, p, \dot{\rho}, T)
\]

Only convective transport is considered in the present implementation \((k = 0)\). The rise of \( \alpha \) from 0 to 1 is effected by the source term \( q \), which is non-zero only in the condensation zone and in the present implementation determined by four parameters \( K_N \), \( K_W \), \( q_1 \), and \( q_2 \) involving the subcooling \( \Delta T = T_{\text{sat}}(p) - T \) and the limiting subcooling \( \Delta T_{\text{lim}} = f(p, \dot{\rho}) \). The latter is a function of the pressure and of the expansion rate \( \dot{\rho} = -(1/p)(dp/dt) \). \( K_N \) is the value of \( \Delta T_{\text{lim}} - \Delta T \) at a point \( N \) by which the nucleation rate has become so large that wetness cannot be neglected any more and the thermal process line begins to deviate from the dry subcooled expansion isentrope. \( K_W \) is the value of \( \Delta T_{\text{lim}} - \Delta T \) at the Wilson point \( (W) \) and normally zero. \( q_1 \) is the (presently constant) value of \( q \).
between points N and W, q2 the value between points W and E. Point E marks the end of the transition zone.

In principle, the dependence of $\Delta T_{\text{lim}}$ on $p$ and $\dot{p}$ could be mapped out by repeated numerical integration of the nucleation and droplet growth equations. More convenient, however, is an implicit analytical solution for the limiting subcooling recently obtained by Huang and Young (1996), who combined classical theory with a Taylor series expansion of $\Delta T(t)$ about the Wilson point. The present implementation uses a biquadratic surface fit to this solution given by Hill et al. (1999).

Outside of the narrow transition zone, the accuracy of the switch equation is irrelevant, and inside it, any upstream numerical influence of the source terms is undesirable. With these criteria, the switch equation is one of the few applications well served by a first order upwind discretization, which has therefore been adopted. Other numerical techniques are the same as for the five Euler equations.

Presently, a two-step procedure is employed in which the equations are first solved in dry steam mode to a predetermined level of convergence. Based on this solution, the field of source terms $q$ is set as described above in function of the subcooling criterion. The Navier-Stokes equations and Equation (8.14) are then solved in parallel to final convergence, keeping $q$ frozen. Attempts at dynamically linking $q$ to the flow solution so far have failed, due to the difficulty of evaluating the subcooling criterion once condensation has occurred.

### 8.2.4 Ansaldò 4-stage LP Turbine

The tabular wet steam properties implementation has been validated with a low-pressure steam turbine. The selected throughflow test case (Hirsch and Denton, 1981) represents the last four stages of the 6-stage LP cylinder of a 320 MW, 3 000 RPM power generation turbine. Only the last four stages are calculated. These have an inlet mass flow of about 100 kg/s, a pressure ratio of 48, and an 89% efficiency. Rotor hub radius ratios are 0.84, 0.74, 0.64, and 0.50, stage pressure ratios 1.95, 2.20, 2.14, and 5.23. The throughflow model accounts for an 8.1% bleed mass flow evacuated through a permeable casing wall between stages 2 and 3 (last stage = stage 4). The imposed inlet entropy and total enthalpy are $s = 7 \, 287.81 \, J/kg/K$ and $H = 2 \, 904.55 \, kJ/kg$ (corresponding to dry, superheated steam of $h = 2 \, 895 \, kJ/kg$ and $p = 3.176 \, bar$), the imposed exit hub static pressure is 0.0671 bar (with
Condensable Fluids

radial equilibrium). An equivalent perfect gas calculation serves as a reference \( c_p = 2000 \text{ J/kg/K}, \gamma = 1.3, R = 461.5 \text{ J/kg/K}, p_{t,in} = 3.30025 \text{ bar}, T_{t,in} = 492.172 \text{ K} \). Constant deviation and loss profiles of respectively 5 deg and 5% are applied to all eight blade rows.

The Euler throughflow model does not call for very fine meshes. The presented results were obtained on a uniform mesh of 12 blocks with 16 cells axially and 32 cells radially each (6144 cells in total), Figure 8.5a. The perfect gas and tabular equilibrium wet steam convergence rates are strictly identical, as illustrated in Figure 8.5b through the RMS residual of density. In terms of CPU time, the steam calculation costs 11% more. The reason for the slightly higher final residual level is not known; with other test cases, the opposite situation has also been observed. Both calculations use 3 multigrid levels in a V-cycle and a CFL number of 4. The initial solution provides a rough, spanwise constant estimate for pressure and flow angles on the block boundaries, which coincide with blade row leading and trailing edges.

Figure 8.6 shows the predicted meridional field of steam quality, indicating a realistic wetness level of about 8% at machine exit. In agreement with a Lagrangian analysis by Guha (1998), condensation is predicted in stages 3 and 4 only \((x < 1)\), stages 1 and 2 operating with dry steam \((x = 1)\). This same reference also indicates that subcooled states are first reached around the trailing edge of stator 2, with substantial subcooling in rotor 2. Spontaneous condensation only occurs, however, when the limiting subcooling is first attained in stator 3. Subcooled states cannot be represented by an equilibrium wet steam model. Instead, wet steam is signalled immediately upon crossing of the saturation line, which in this throughflow simulation correctly and tentatively occurs in stage 2.

Design features of LP steam turbines (low hub ratio; high casing flare, blade speeds, and pressure ratio) imply very high relative Mach numbers at the nozzle root and rotor tip (Hirsch and Denton, 1981). Exit Mach numbers up to 1.8 are predicted by the present throughflow calculation in the last stage, and values between 1.5 and 1.6 in the first three stages. One advantage of the Euler throughflow model is its ability to capture shocks. In this turbine application, it automatically finds an axisymmetric representation of the complex trailing edge shock system consistent with the governing conservation laws, satisfying of course radial equilibrium.

The equilibrium two-phase model responds immediately to the heating associated with these captured aerodynamic shocks, which explains the reversion to lower wetness observed in several places in Figure 8.6. Downstream of rotor 2, the steam in the pitch-average is still superheated, restored from subcooled (in reality) or wet states (in the present equilibrium model). At the tip of stators 3 and 4, steam wetness at the trailing edge (= rotor leading edge) is locally reduced to values of about 0.5% and 2%, from respectively 3% and 5%. The maximum wetness of about 10.5% occurs down-
stream of rotor 4 tip, but is immediately reduced by a weak axisymmetric aerodynamic shock that extends from the casing a short way into the flow path. In reality, aerodynamic shocks are followed by inertial and thermal relaxation zones. For a representative droplet diameter of 0.1 µm (Young, 1992; Guha, 1998), their thickness (measured in the flow direction) behind a 45 deg oblique shock at Mach 1.8 and 10% wetness is respectively 2 mm and 20 mm (Moore, 1976), to be compared with an axial length of the Ansaldo last stage of 380 mm. Equilibrium results may thus not be entirely implausible. The throughflow test case description regrettably does not report wetness data.

Global performance data, summarized in Table 8.3, allow a number of observations. First, the mass flow in trans- and supersonic turbines most strongly depends on throat width, over which the Euler throughflow model therefore offers direct control via coupled tangential blade thickness and flow angle distributions. The predicted mass flows are in excellent agreement with the experimental value, without any adjustments to either throat width, turning, losses, or exit pressure. Differences at exit are −0.9% for the perfect gas and +0.3% for steam. Second, the perfect gas and steam calculations predict the same pressure ratio (to within 0.2%), in excess of the measured value by 1.3%. Efficiency and power furnish a beautiful illustration of characteristic attributes of wet steam turbines. Third, therefore, the difference of 3.6 percentage points between the perfect gas and steam calculated efficiencies is in good agreement with the Baumann rule, which for this turbine with two dry stages and two wet stages of 4% and 8% steam wetness would suggest an efficiency drop in the vicinity of 3%. (The lower efficiency level compared with the Baumann rule, which for this turbine with two dry stages and two wet stages of 4% and 8% steam wetness would suggest an efficiency drop in the vicinity of 3%. (The lower efficiency level compared with the Baumann rule, which for this turbine with two dry stages and two wet stages of 4% and 8% steam wetness would suggest an efficiency drop in the vicinity of 3%. (The lower efficiency level compared with the Baumann rule, which for this turbine with two dry stages and two wet stages of 4% and 8% steam wetness would suggest an efficiency drop in the vicinity of 3%. (The lower efficiency level compared with the Baumann rule, which for this turbine with two dry stages and two wet stages of 4% and 8% steam wetness would suggest an efficiency drop in the vicinity of 3%. (The lower efficiency level compared with the Baumann rule, which for this turbine with two dry stages and two wet stages of 4% and 8% steam wetness would suggest an efficiency drop in the vicinity of 3%.) Finally, because of latent heat release, wet steam has a higher work capacity than dry steam. Predicted shaft power of the two wet stages (P_{34}) consequently is 6.8% higher with steam than with the perfect gas. A 3.8% power excess for the two dry stages (P_{12}) in favour of the perfect gas is attributed to deviation of actual steam properties from the chosen perfect gas approximation. Total predicted power with steam is higher by 1.7%.

Figure 8.7 compares exit axial Mach numbers, $M_x$, and absolute flow angles, $\alpha$. Although, due to the simplified assumptions on imposed losses and deviation, care must be exercised when assessing absolute accuracy, the steam solution obviously is in much better agreement with the measured profiles than the perfect gas solution. Compared with steam, the perfect gas underpredicts $M_x$ by between 12% near the hub and 20% near the casing. The two calculated $\alpha$ match at the casing, but differ by 20 deg at the hub. Along the inner 50% span, agreement of steam is very good (for a throughflow
solution). A sudden increase of $M_x$ (from 0.7 to 0.8) and $\alpha$ (from $-10\,\text{deg}$ to 0) occurring at mid-span is not reflected in the throughflow, however, which between $50\%$ and $90\%$ span maintains the general trend, being roughly parallel to the experimental profile, but shifted by a small amount. Over the outermost $10\%$ span, the measurements indicate a sharp reduction in $M_x$ (from 0.75 to 0.55). The fact that this is perfectly reproduced by the Euler throughflow code may be taken as evidence for the adequacy of the captured shocks (which are strongest at nozzle roots and rotor tips). On the other hand, captured axisymmetric shocks (visible in Figure 8.6 downstream of the last stage) will exacerbate any initial deviation of the flow direction from axial, which explains the poor agreement of $\alpha$ at the casing ($-25\,\text{deg}$).

### 8.2.5 Nucleating Nozzle Flow

The switched condensation model is demonstrated on a long, straight nozzle used in experiments by Barschdorff et al. (1976), Figure 8.8. Incidentally, this test case also provides a good quantitative validation of the basic tabular steam properties implementation. A circular arc nozzle profile yields a nearly constant expansion rate, Gyarmathy (1976) lauds the high quality of the measured data. Figure 8.9a shows a sketch of the experimental setup. The constant width of this nozzle is 50 mm, the full

<table>
<thead>
<tr>
<th>Table 8.3: Global performance data of the Ansaldo 4-stage LP turbine.</th>
</tr>
</thead>
<tbody>
<tr>
<td>$m_\text{out}$ [kg/s]</td>
</tr>
<tr>
<td>----------------------</td>
</tr>
<tr>
<td>Experiment</td>
</tr>
<tr>
<td>91.7</td>
</tr>
<tr>
<td>47.3</td>
</tr>
<tr>
<td>89.0</td>
</tr>
<tr>
<td>–</td>
</tr>
<tr>
<td>–</td>
</tr>
<tr>
<td>–</td>
</tr>
</tbody>
</table>
height at the throat 60 mm, and the radius 1.027 mm.

The very thin boundary layers are assumed to be laminar and the walls to be adiabatic. Although the flow is clearly 3D, the difference between a 3D and a 2D calculation was found to be negligible. 2D calculations have therefore been performed. The mesh, shown in Figure 8.8, contains 128 cells axially and 32 cells in the normal direction (4,096 cells in total).

The inlet total pressure and temperature are respectively 1.45 bar and 131°C (total enthalpy 2734.6 kJ/kg, entropy 7343.7 J/kg/K), which corresponds to a superheat of 20.7 K. This case has been selected because the condensation zone is located sufficiently far from the nozzle throat to allow variations in its location and width to be studied. With a lower inlet superheat, spontaneous condensation occurs closer to the throat and becomes very sensitive to minute variations in geometry and boundary layer thickness. In some cases, unsteady oscillation of the condensation shock wave is observed experimentally.

Figure 8.8 Barschdorff et al. (1976) nozzle 2: computational grid (128 x 32 = 4,096 cells).

Figure 8.9 Barschdorff et al. (1976) nozzle 2: (a) sketch of the experimental set-up (from Barschdorff), (b) calculated pressure distribution along the axis for different locations and spreads of the switch between subcooled dry and equilibrium wet steam ($p_0 = 1.45$ bar, $T_0 = 131$°C).
Figure 8.9b compares the static pressure distribution along the nozzle axis computed in different modes with the experimental data. The x-coordinate measures the distance from the nozzle throat, the hump in the experimental curve (filled circles) marks the condensation zone. A non-condensing computation (light dashed line, switch $\alpha = 0$ in the whole field) yields a smooth, near-isentropic expansion, in perfect agreement with the data upstream of the condensation zone. A smooth, near-isentropic expansion is also obtained for equilibrium wet steam (light dotted line, switch $\alpha = 1$ in the whole field), but at a lower mass flow. This is because the equilibrium, wet sound speed and isentropic exponent are lower than the respective frozen, dry values. When the flow crosses the saturation line, in this case in the subsonic part of the nozzle, there is thus a discontinuity in those properties for the equilibrium wet expansion. The velocity at the throat is then limited by the lower equilibrium sound speed.

Five cases of switched condensation have been calculated to examine the sensitivity with respect to two parameters: the abruptness and the location of the transition. The switch $\alpha$ equals 0 upstream of the condensation zone and 1 downstream, with a linear transition in between to represent the condensation process. The applied variations have been chosen much larger than any potential uncertainty on the limiting subcooling. (At these conditions, an 0.05 change in pressure ratio roughly corresponds to a 10 K temperature change.) There are 13 intermediate grid points for the cases labelled default, early and late. Immediate means that $\alpha$ jumps from 0 to 1, with no intermediate points. It is seen that (i) the switched condensation model gives nearly the correct pressure jump, (ii) the downstream flow state is practically independent of the width of the condensation zone, and (iii) early condensation produces a slightly lower pressure, late condensation a minimally higher one.

In the above calculations, the problem of determining the correct location at which spontaneous condensation occurs has been circumvented by setting a fixed spatial distribution of the condensation switch. Results of a predictive calculation of the same operating point are shown in Figure 8.10. In the convergence curves, Figure 8.10a, one clearly distinguishes the initial dry calculation and the following switched wet steam calculation. Even though the convergence rate deteriorates slightly when condensation is activated, single precision machine accuracy is reached after only 130 iterations, of which about 50 are in the frozen dry phase. In particular, convergence of the switch equation (circles) parallels that of the mass, momentum, and energy equations. The apparently higher residual level is due to a smaller first residual, arbitrarily generated by initialising the switch field with a value

Figure 8.10 Barschdorff et al. (1976) nozzle 2 ($p_0 = 1.45$ bar, $T_0 = 131^\circ$C), switched condensation model: convergence history, (b) calculated pressure distribution with two source terms.
that is slightly different from the imposed inlet value. The nozzle calculations use 4 multigrid levels in a V-cycle, the CFL number is 3.

The pressure distribution calculated with the two-source term method is in excellent agreement with the data, Figure 8.10b. In this calculation admittedly the four model parameters have been adjusted by trial and error (\(K_N = 5\) K, \(K_W = 1\) K, \(q_1 = 5\) 000/m\(^3\), \(q_2 = 40\) 000/m\(^3\)). We are, however, confident that simple algebraic correlations can be developed which will provide a similarly good match over a broad range of conditions. It appears from these parameters that, in agreement with the actual physical processes, nucleation must be triggered at less than critical subcooling and that the vapour-to-liquid conversion rate is about an order of magnitude larger in the condensation dominated phase than in the nucleation dominated phase. The small deviation of 1 K between the Wilson temperature required in the calculation and that predicted by the analysis of Huang and Young (1996) is attributed to details of the numerical implementation and must be viewed in relation to the absolute amount of subcooling which for expansion rates of the order of 3 000/s and pressures of the order of 1 bar is around 30 K. It is significantly smaller (and of opposite sign) than the 10% deviation reported by Hill et al. (1999, who do not discuss implementation aspects), and consistent with a systematic error on \(\Delta T_W\) inherent in the Huang and Young analytical solution (e.g., 0.8 K at 0.1 bar and \(p = 1\) 000/s).

8.2.6 Conclusions

Recognizing the shortcomings of traditional methods to account for real gas and wet steam properties in CFD codes, a purely tabular formulation has been developed and implemented in a state-of-the-art Navier-Stokes solver for turbomachinery applications. All required thermodynamic variables are directly interpolated from dedicated, structured tables representing the scientific reference formulation IAPWS-95 to any desired level of accuracy. The presented sample calculations used full-range 30 x 30 bicubic tables with a less than 0.1% RMS error. Careful optimization of the interpolation technique and the strategic choice of interpolated and dependent thermodynamic variables have lead to an efficient method. Depending on the test cases and selected options, overheads between 5% and 30% have been measured, with respect to the perfect gas (11% in the shown throughflow example).

The interpolation tables allow to address two issues simultaneously. First, they might provide a final answer to the question of how to best account for the real gas properties of steam in CFD codes. Second, wet steam can be modelled as a single fluid with the properties of the equilibrium two-phase mixture of saturated water and steam, directly accessible in the same way as dry steam properties. This basic formulation has been demonstrated with a throughflow calculation of a four-stage low-pressure condensing steam turbine. Convergence rates with steam and with the perfect gas were shown to be strictly identical, requiring only about 100 iterations for a 5-order residual drop. Within the limitations of the equilibrium wet steam model, the results are plausible in all respects. Agreement with available data is fair, a realistic wetness level is predicted, and comparison with an equivalent perfect gas calculation showed the expected effect of wet steam on efficiency and power output.

First tests after an original idea of Hill et al. (1999) confirmed that a gradual switch between the meta-stable single-phase dry and equilibrium two-phase wet steam properties at an appropriate level of subcooling constitutes a viable algebraic model for spontaneous condensation, offering a favourable accuracy-to-cost quotient. The sensitivity of the pressure jump observed in supersonic condensation to details of the transition process was examined for nozzle flow and found to be small, especially when compared with equilibrium or non-condensing flow. A refined transition in two stages, inspired by the dominating physical processes (nucleation and condensation), enables a surprisingly faithful reproduction of the measured pressure distribution. To ensure independence from mesh topology, the condensation switch is set by a simple, well-behaved convection equation.

It is concluded that interpolation tables are an excellent means of including real gas and wet steam properties in CFD codes, in terms of all relevant criteria (accuracy, robustness, speed, physical models, software engineering), giving access to exact thermodynamic data at essentially perfect gas cost.
8.3 Application to LH2

8.3.1 Motivation

Cryogenic liquids and supercritical fluids such as LH2 have peculiar thermodynamic properties. Contrary to what the name liquid suggests, these fluids are not incompressible or nearly incompressible, as for example water at ambient conditions. Rather, they are characterized by a strong and highly variable compressibility, with a particularly strong dependence of density on temperature, as pointed out by Merkle et al. (1998, the interest of these authors being in the regenerative cooling of rocket engine nozzles and combustion chambers).

Mathematically, fluid flow is described by the 3D Navier–Stokes equations, a system of 5 second order partial differential equations which represent the conservation laws of mass, momentum, and energy for a viscous, heat conducting fluid. No general analytical solution being known, these equations must be discretized and solved numerically for each concrete case. Typical calculations involve meshes of the order of $10^5$ to $10^6$ cells and require of the order of $10^2$ to $10^4$ iterations, with computing times of the order of hours up to several days. Each iteration requires one or more evaluations of fluid thermodynamic properties in each cell, resulting in at least $10^7$ to $10^{10}$ evaluations for one simulation.

It is therefore essential that the computational overhead from the state equation be kept to a minimum. Simple approximations to the real fluid properties are normally used: the fluid is either considered to behave as a perfect gas, or to be incompressible. Neither of these models provides satisfactory accuracy for flows of cryogenic and supercritical fluids with strong pressure or temperature variations. On an intermediate stage, the barotropic fluid model considers the density to be a function of pressure only, $\rho = \rho(p)$. The temperature dependence of density cannot be captured with this model.

To eliminate from the numerical flow simulation all uncertainty arising from the fluid properties, a recognized reference state equation must be used. The usual reference for hydrogen is a modified Benedict–Webb–Rubin (MBWR) equation devised by McCarty et al. (1981), see also Younglove (1982). The complexity of this EOS makes its use in a flow solver prohibitively expensive. Quantitative data on this situation are due to Gerolymos and Geai (1994) who implemented the above mentioned MBWR state equation for LH2 in a 1D code (with area variation), solving the Euler equations by an explicit Runge–Kutta method. The required EOS inversion was done through Newton iterations. The authors concluded that the LH2 state equation causes an overhead of 8 times the cost of a perfect gas calculation. Extrapolating from 1D Euler to 3D Navier–Stokes, they estimated an overhead of 3 times: LH2 calculations would take 4 times as long as perfect gas calculations. Another drawback of directly using state equations which cannot be inverted explicitly is the added complexity in the code caused by the necessary iterative inversion. For complex 3D test cases with strong variations in fluid properties, especially during the first several iterations, such a procedure may also diminish the robustness of the numerical method.

Within the present work, an efficient technique to account for real fluid properties by means of table interpolation and implemented it in the 3D Navier–Stokes solver Euranus (Hirsch, 1991). Euranus has been used extensively both in industrial and academic environments to simulate the flow in various components of the turbo pumps of cryogenic engines, so far, however, with simplified fluid models only. The interpolation of thermodynamic properties from tables now allows the use of the exact physical properties of LH2 with only a minimal computational overhead.

8.3.2 Boundary Conditions for LH2

Depending on the availability of thermodynamic data and hence interpolation tables, it may not always be possible to apply the usual inlet and exit boundary conditions, imposing, for example, total pressure and total temperature at inlets for compressible flows. Alternatives have therefore been investigated, and an overview of some inlet boundary conditions programmed and tested for condensible fluids is given in Table 7.1. Proposed exit BC are listed in Table 7.2 (dir(\hat{v}) = flow direction, $w_5$...
At inlets imposing the total enthalpy and the entropy, is the tabular property equivalent of imposing total temperature and total pressure for the perfect gas. For situations in which tables in function of enthalpy, h, and entropy, s, are not available, an alternative that performs equally well has however been found with ID # 8, which imposes the total energy and the density. Inlet boundary conditions imposing the velocity (ID # 3, 4, 5, and 6) are applicable for very low normal Mach numbers only, independently of the imposed and extrapolated thermodynamic variable. Tests with the perfect gas indicate that preconditioning can somewhat increase the allowable Mach number, for example by a factor of 2. Convergence rates with these BC are poor. Imposing two static thermodynamic variables, ID # 7, is not recommended, since it may lead to an ill-posed problem, for instance if the exit BC imposes the pressure in a section with the same cross sectional area as the inlet (Reference?). A symmetric converging-diverging nozzle would be a typical example. The indeterminacy of the problem described by this combination of BC manifests itself in a linear drift of the solution; calculations do not diverge exponentially as they would in case of a numerical instability. It is, however, enough for convergence to break down that the condition be fulfilled approximately, as on the flat plate. The only universal, robust inlet BC for LH2 with a basic set of tables involving only pressure, density, and internal energy therefore is ID # 8.

At subsonic exits, the static pressure is usually imposed. ID # 3 and 4 both extrapolate the total enthalpy (rothalpy in the relative system) and velocity vector, but use different tables calculate the density. In the absence of interpolation errors, both are identical. ID # 5, which extrapolates the total energy (rotary total energy in the relative system), uses an inverted p(e,ρ) table. When the latter is transformed into a ρ(e,ρ) table, the mixture region cannot be adequately represented at a normal resolution in the original tables because it (almost) collapses into a line. The outflow therefore must be

Table 8.4: Overview of inlet boundary conditions for condensable fluids.

<table>
<thead>
<tr>
<th>ID #</th>
<th>Imposed</th>
<th>Extrapolated</th>
<th>Tables</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>H, s, dir(̇v)</td>
<td>̇v</td>
<td>ρ(h,s), p(h,s)</td>
<td>for all subsonic speeds</td>
</tr>
<tr>
<td>3</td>
<td>H, ̇v</td>
<td>p</td>
<td>ρ(h,p)</td>
<td>low speed only (M &lt; 0.03)</td>
</tr>
<tr>
<td>4</td>
<td>T, ̇v</td>
<td>r</td>
<td>e(T,ρ), p(e,ρ)</td>
<td>low speed only (M &lt; 0.03)</td>
</tr>
<tr>
<td>5</td>
<td>E, ̇v</td>
<td>p</td>
<td>ρ(e,ρ)</td>
<td>low speed only (M &lt; 0.03)</td>
</tr>
<tr>
<td>6</td>
<td>E, ̇v</td>
<td>w_5</td>
<td>ρ(e,ρ)</td>
<td>low speed only (M &lt; 0.03)</td>
</tr>
<tr>
<td>7</td>
<td>e, ρ, dir(̇v)</td>
<td>̇v</td>
<td>ρ(e,ρ)</td>
<td>ill-posed problem possible</td>
</tr>
<tr>
<td>8</td>
<td>E, ρ, dir(̇v)</td>
<td>̇v</td>
<td>ρ(e,ρ)</td>
<td>all subsonic speeds</td>
</tr>
</tbody>
</table>

Table 8.5: Overview of exit boundary conditions for condensable fluids.

<table>
<thead>
<tr>
<th>ID #</th>
<th>Imposed</th>
<th>Extrapolated</th>
<th>Tables</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>p</td>
<td>H, ̇v</td>
<td>s(h,p), ρ(h,s)</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>p</td>
<td>H, ̇v</td>
<td>ρ(h,p)</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>p</td>
<td>E, ̇v</td>
<td>ρ(e,ρ)</td>
<td>not for mixture region</td>
</tr>
</tbody>
</table>
in the single phase region, if this ID # is chosen. In practice, no difference was noted between ID # 4 and 5.

8.3.3 Validation of the Condensable Fluid Model with LH2

The condensable fluid implementation has been validated for LH2 with three test cases: subsonic inviscid flow through a symmetric Laval nozzle, the laminar flat plate, and the turbulent flat plate. Each will be discussed in this section, giving first a description of the test case, then a summary of the principal numerical settings and parameters, and finally an assessment of the results with emphasis on the solution accuracy and convergence rate. Comparisons are made either with analytical solutions, or, for the turbulent flat plate, with the perfect gas at equivalent flow conditions which serves as a reference.

8.3.3.1 Inviscid nozzle flow

Subsonic inviscid nozzle flow has been selected as the first application of the condensable fluid implementation with hydrogen. This test case serves to verify the modifications that have been made to the code and to gain experience with calculations on the liquid side. Previous tests with steam had been limited to the vapor side, with markedly subcritical states and low wetness fractions, resulting in perfect gas–like fluid properties.

Figure 8.11a shows a schematic representation of the selected nozzle geometry. The symmetric ($A_{\text{in}} = A_{\text{out}}$) converging–diverging circular arc nozzle has a half angle $\alpha$ of 20 deg. The area ratio $A_{\text{in}}/A^* = 2$, $A^*$ denoting the section area at the throat. At the inlet, $E = -200\,000$ J/kg and $\rho = 70$ kg/m$^3$ are imposed. At the exit, the total energy and the velocity vector are extrapolated, the density is consistently interpolated from the $\rho(e,p)$ table. The imposed exit pressure of 5.5 MPa yields a peak Mach number of about 0.28.

With pure liquid hydrogen, both the convergence rate and the solution accuracy are excellent. Figure 8.11b shows convergence history. The central scheme with the default values for the artificial dissipation coefficients is used (1.0 and 0.1 for VIS2 and VIS4, respectively). The convergence acceleration parameters are: 4-level multigrid, residual smoothing with time step dependent coefficients, CFL number 4. Only around 100 multigrid cycles are required for convergence.

Calculated inlet values are: $M_{\text{in}} = 0.12$, $\rho_{\text{in}} = 70$ kg/m$^3$, $p_{\text{in}} = 5.5$ MPa, $T_{\text{in}} = 27.5$ K, and calculated throat values: $M_{\text{thr}} = 0.27$, $\rho_{\text{thr}} = 68.25$ kg/m$^3$, $p_{\text{thr}} = 3.25$ MPa, $T_{\text{thr}} = 26.2$ K. The density variation is approximately 2.5%, which is of the same order as for a perfect gas with isentropic exponent 1.4.

To have a quantitative validation of the solution, Fig. 8.12 compares the evolution of velocity (Fig. 8.12a) and pressure (Fig. 8.12b) along the nozzle axis with the analytical incompressible solution for quasi-one-dimensional flow at the mean density of the LH2 flow. Since at this low Mach number the density variation is small, there should be good agreement between the two solutions. One indeed finds that the compressible LH2 and analytical incompressible curves nearly coincide. The results presented here were obtained on a fairly coarse mesh of $64 \times 16 = 1024$ cells.

By contrast with the promising results in the liquid region, converged solutions could not be obtained in the two-phase region, neither for flows that locally expand from the liquid into two phase region (cavitating liquid flows), nor for pure mixture flows.

8.3.3.2 Laminar flat plate

A number of calculations have been performed on the laminar flat plate, both adiabatic and isothermal. As an example, results are here presented for an adiabatic plate with the following boundary conditions. At the inlet, the total energy $E = -94\,487.5$ J/kg, density $\rho = 60$ kg/m$^3$ and the flow direction are imposed, the modulus $|\mathbf{v}|$ is taken from the extrapolated velocity vector. The pressure $p(e,\rho)$ is calculated from the table. At the exit, a pressure of $p = 7.123$ MPa is imposed, $E$ and $|\mathbf{v}|$ are extrapolated and $\rho(e,\rho)$ is interpolated from the table. These conditions produce a flow at Mach 0.1 that is
entirely in the liquid region. A Reynolds number of 10,000 was selected. Anecdotally, we note that, due to the very low viscosity of liquid hydrogen, the required plate length is about 12.6 µm.

Constant kinematic viscosity and thermal conductivity were chosen for comparison with the Blasius solution. (Variable viscosity and conductivity interpolated from tables have also been tested successfully.) For the chosen boundary conditions, the values obtained through interactive evaluation of the respective interpolation tables are \( \nu = 1.3227 \times 10^{-7} \text{ m}^2/\text{s} \), \( k = 0.1101 \text{ W/m/K} \).

The Blasius solution of the thermal boundary layer depends on the Prandtl number, which is a property of the fluid and cannot be chosen. For liquid hydrogen, it is of the order of 1. A value of 1.0589 was calculated for the selected liquid state, with the above values of \( \nu \) and \( k \). An interactive table evaluation tool served to numerically estimate the specific heat at constant pressure, \( c_p \), by taking differences of enthalpy and temperature at constant pressure. The value obtained through this manual procedure is 14,690 J/kg/K at \( e = -100,000 \text{ J/kg} \) and \( \rho = 60 \text{ kg/m}^3 \). (Neither \( c_p \) nor the Prandtl number enter explicitly into the numerical calculation.)

The obtained solution is shown in Fig. 8.13. As evidenced by Figure 8.13d, convergence is again excellent. The relevant parameters are: 6-level multigrid in a V-cycle with 1, 2, 4, 8, 16, 32 sweeps on grid levels 1, 2, 3, 4, 5, 6 respectively, time step dependent residual smoothing, CFL number 4. Machine accuracy is reached in 150 multigrid cycles. Figures 8.13a, b, and c show respectively the calculated velocity and temperature profiles, and the skin friction coefficient along the plate, each compared with the analytical Blasius solution. The accuracy is adequate for the central scheme on a mesh of this resolution (64 x 64 = 4096 cells).

In calculations with an isothermal wall, converged solutions could not be obtained with wall temperatures that would result in local two-phase flow near the plate. The reasons for the encountered difficulties appear to be the same as for the nozzle flow.

**8.3.3.3 Turbulent flat plate**

To ensure the compatibility of the LH2 implementation with the turbulence models, in particular the \( k-\varepsilon \) models, the turbulent flat plate was picked as a representative test case that permits a quantitative validation. Figures 8.14 and 8.15 show results for one of the cases that have been calculated. The thermodynamic conditions are essentially the same as for the laminar flat plate discussed in the previous subsection.

At the inlet, \( E = -95,000 \text{ J/kg} \) and \( \rho = 60 \text{ kg/m}^3 \); the imposed values of \( k \) and \( \varepsilon \) correspond to a free stream turbulence intensity of 1% and a turbulent eddy viscosity equal to the molecular viscosity. The Mach number is 0.1. The Reynolds number at the plate trailing edge is \( 2E+6 \), velocity profiles are extracted at \( Re_\theta = 1.6E+6 \), which corresponds to a momentum thickness Reynolds number \( Re_\theta \) of approximately 3600. An exit pressure of 7.123 MPa is imposed. The resulting temperatures range from 38.38 K in the free stream to 38.71 K at the wall.

The mesh has 96 cells in the streamwise direction, of which 32 are located upstream of the leading edge, and 64 cells in the wall normal direction (total 6144 cells). The \( y^+ \) coordinate of the first inner cell center is 1, the highest cell aspect ratio is approximately 1200.

The default settings have been retained for the numerous switches related to the \( k-\varepsilon \) model. Convergence relevant parameters are: MG-4-V[1248] or [1234] (4-level multigrid in a V-cycle with 1, 2, 4, 8 (Baldwin-Lomax) or 1, 2, 3, 4 Runge-Kutta sweeps (\( k-\varepsilon \)) on grid levels 1, 2, 3, 4, respectively), CFL 4 (Baldwin-Lomax) or 2 (\( k-\varepsilon \)), time step dependent residual smoothing. Second order artificial dissipation was disactivated for the flat plate calculations in the mass, momentum and energy equations.

The essential comparison as far as the condensable fluid module is concerned is between LH2 and air, or more generally, the perfect gas model. To avoid the generation of a separate mesh, a perfect gas with the properties of air is used, except for the kinematic viscosity, which has been decreased from the usual room temperature value of about \( 1.36E-5 \text{ m}^2/\text{s} \) down to \( 4.38E-8 \text{ m}^2/\text{s} \), keeping the same Prandtl number; the same mesh can then be used for hydrogen and for the perfect gas.

Calculations were performed with the Baldwin-Lomax model and with \( k-\varepsilon \) models. Figure 8.14 presents the results for the Baldwin-Lomax model. Looking first at the velocity profile, normal-
ized with the friction velocity and plotted in Fig. 8.14a against the law-of-the-wall coordinate $y^+$, the LH2 solution is found to coincide perfectly with the solution for air. Similarly, the convergence histories in Fig. 8.14b are practically identical, reaching machine accuracy in approximately 500 multigrid cycles.

The same comparison is made in Fig. 8.15 for the Yang-Shih low-Reynolds $k$-$\varepsilon$ model. For information on the turbulence models we refer to the Euranus/FINE user’s guide. Focusing first on convergence, Fig. 8.15b shows the convergence histories of the streamwise momentum equation (labelled $u$) and of the $k$ equation ($k$). The difference between these two variables is normal and should not be mistaken for the difference between the two fluid models. These are distinguished by color (red for air, blue for LH2) and nearly coincide. The same is true of the normalized velocity profiles in Fig. 8.15a, where LH2 and air would be indistinguishable had different symbols not been used. An analogous test has been performed with the high-Re $k$-$\varepsilon$ model with wall functions, again yielding identical solutions and convergence histories for liquid hydrogen and for the perfect gas.

In conclusion, the full compatibility of the condensable fluid implementation with the different turbulence models can be considered as demonstrated.

Figure 8.11  Subsonic nozzle flow of subcooled LH2, (a) nozzle geometry, (b) convergence history.
Figure 8.12  Inviscid nozzle flow of LH2 (area ratio 2, approximate throat conditions: $p \approx 3.25$ MPa, $T \approx 26.2$ K, $\rho \approx 68.3$ kg/m$^3$, $M \approx 0.27$), comparison with incompressible flow (exact quasi-1D solution with LH2 mean density): (a) velocity, (b) pressure.
Figure 8.13  Laminar adiabatic flat plate with LH2 (Pr ≈ 1.06, Re = 10 000, M = 0.1): (a) streamwise velocity, (b) temperature, (c) skin friction, (d) convergence history (MG-6-V, CFL 4); mesh size 64 x 64 = 4096 cells.
Figure 8.14 Turbulent flat plate ($Re = 2E+6$, $M = 0.1$, Baldwin-Lomax model), comparison of air and LH2: (a) velocity profiles, (b) convergence history (MG-4-V[1248], CFL 4); mesh size 96 x 64 = 6144 cells.

Figure 8.15 Turbulent flat plate ($Re = 2E+6$, $M = 0.1$, Yang-Shih low-Re $k$-$\varepsilon$ model), comparison of air and LH2: (a) velocity profiles, (b) convergence history (MG-4-V[1234], CFL 2); mesh size 96 x 64 = 6144 cells.
Figure 8.16 Flow of superheated (metastable) LH2 in a Laval nozzle: (a) convergence histories for three different multigrid synchronization strategies, (b) computational grid (64 x 16 = 1024 cells) and color contours of static pressure (range –18.9 to 11.2 bar).
8.3.4 LH2 Turbo Pump

In what are believed to be the first ever 3D Navier-Stokes calculations of a complete pump configuration with the exact thermodynamic and transport properties of LH2, the effect of using exact properties rather than the customary simplified fluid models has been examined by calculating the flow through the first stage of the Vulcain LH2 turbo pump (also abbreviated TPH = turbopompe hydrogène, consisting of an impeller, a vaned diffuser, and a deswirl cascade) on a mesh of about 350 000 cells for one operating point (33 000 rpm, 44.62 kg/s, inlet static temperature 24.4 K, exit static pressure 80.7 bar). LH2 and incompressible flow at two Reynolds numbers, 150 000 (air, \( y_1^+ \approx 1 \)) and 40 000 000 (LH2, \( y_1^+ \approx 50 \)), are compared both graphically and in terms of section-averaged variables at five data stations.

8.3.4.1 LH2 interpolation tables

The required thermophysical properties of liquid hydrogen were provided by an industrial partner in the form of raw data tables for pressure, temperature, speed of sound, dynamic viscosity, thermal conductivity, and void fraction in function of internal energy (e) and density (\( \rho \)). The raw data tables cover a density range from 50 to 75 kg/m\(^3\), which corresponds to around 1.6 to 2.4 critical densities, covered by 251 uniformly distributed data points. The internal energies span a range of 800 kJ/kg, located between –300 and 500 kJ/kg and resolved by 161 uniformly distributed data points. The total size of the tables is 251 \( \times \) 161 = 40 411 data points. Extremal values of thermodynamic variables encountered on this rectangular (e, \( \rho \))-region are pressures of 0.129 and 834.8 bar (0.01 and 64.2 p\(_\text{crit}\)), temperatures of 14.9 and 113.2 K (0.45 and 3.43 T\(_\text{crit}\)), sound speeds of 35.4 and 1 975 m/s, and void fractions of 0 and 0.34. In the absence of chemical reactions and thus when dealing with a single pure fluid such as LH2, the zero point of energy can be chosen arbitrarily. Where it has been placed in these particular data tables is therefore irrelevant and has not been communicated. The raw data tables cover an appropriate range of thermodynamic states.

During the convergence transient, computations may temporarily exceed the table range. Initially, this posed a severe obstacle, because the numerical solution procedure would systematically diverge. The calculations proved particularly sensitive to the quality of the p(e,\( \rho \)) table, which implicitly determines the speed of sound as a combination of the two partial derivatives. Special attention was therefore given to the extrapolation behavior of the tables. It is not essential that the extended liquid properties be highly accurate, but they must smoothly continue the stable, subcooled liquid to avoid numerical problems due to a sudden change in fluid properties. This is of particular importance for the extrapolation to metastable, superheated liquid states.

To demonstrate the achieved benign extrapolation behavior, Figure 8.21a does not show the actual p(e,\( \rho \)) interpolation table, but a surface interpolated from this table on a uniform mesh of 75 x 75 points which covers the range of the table plus a 20 % margin around it. The extrapolated margin clearly appears as a flat square in each of the four corners. The margin width of 20 % has been chosen for the purpose of visualization only. The illustrated extrapolation behavior extends arbitrarily far in all four directions.

The presented calculations use tables for pressure in function of internal energy and density (PER), a corresponding inverse table for density in function of internal energy and pressure (REP), temperature in function of internal energy and pressure (TER), a corresponding inverse table for internal energy in function of temperature and density (ETR), as well as tables in function of internal energy and density for dynamic viscosity (MER), thermal conductivity (KER), and optionally the speed of sound (CER) and void fraction (AER, used for postprocessing only).

The interpolation tables are bilinear single tables using every other data point. A test against a set of 1000 random data points (independent of the data points from which the tables were generated) showed the table accuracy (and hence their mutual thermodynamic consistency) to be better than 1 % everywhere, with RMS errors below 0.1 % for all tables. Figure 8.21b shows the error map for the PER table.
8.3.4.2 Geometry and mesh

The TPH (turbo-pompe hydrogène) test case represents the first stage of the Vulcain 1 LH2 turbo pump with the A4 return channel. It is composed of a shrouded impeller with six main and six splitter blades, followed by a vaned radial diffuser and a vaned return channel with 11 blades each, Figure 8.17. Numeca has already performed calculations on this TPH configuration in the past with air modelled as a perfect gas for comparison with air experimental data (Lorrain, 1997). The mesh generated for those calculations has been re-used in the present LH2 demonstration calculations. It is composed of six blocks, four in the impeller (one H-block in each of the two blade passages and a small block downstream of each trailing edge) and one each in the diffuser and the return channel (as a H-block). The impeller mesh has 179 124 points, the diffuser and return channel meshes together 176 418 points, giving a total mesh size of 355 542 points, Table 8.6. The existing mesh, of which Figure 8.18 shows two views, allows four levels of multigrid.

<table>
<thead>
<tr>
<th>Block</th>
<th>Azimuthal, I</th>
<th>Radial, J</th>
<th>Streamwise, K</th>
<th>Points</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 and 2</td>
<td>25</td>
<td>33</td>
<td>97</td>
<td>80 025</td>
</tr>
<tr>
<td>3 and 4</td>
<td>17</td>
<td>33</td>
<td>17</td>
<td>9 573</td>
</tr>
<tr>
<td>Impeller</td>
<td></td>
<td></td>
<td></td>
<td>179 124</td>
</tr>
<tr>
<td>5</td>
<td>33</td>
<td>33</td>
<td>73</td>
<td>79 497</td>
</tr>
<tr>
<td>6</td>
<td>33</td>
<td>33</td>
<td>89</td>
<td>96 921</td>
</tr>
<tr>
<td>Diffuser and return channel</td>
<td></td>
<td></td>
<td></td>
<td>176 418</td>
</tr>
<tr>
<td>Total</td>
<td></td>
<td></td>
<td></td>
<td>355 542</td>
</tr>
</tbody>
</table>

8.3.4.3 Boundary conditions and calculated operating point

The following data for operation with LH2 were available for this test case: rotational speed (33 000 rpm), mass flow (44.62 kg/s), hub and shroud velocity triangles at inlet and exit of the impeller, as well as static pressure, static temperature, and density at five stations (station 1 at stage inlet, slightly upstream of the impeller leading edge, station 2 at the diffuser inlet, station 3 at the diffuser trailing edge, station 4 at the leading edge of the return vanes, and station 5 at stage exit, all shown in Figure 8.17a). The thermodynamic variables are assumed to represent section-averaged values (as opposed to local values at mid-span).

Angular velocity, the uniform inlet temperature, the two inlet velocity profiles, and the exit pressure were chosen to define the calculated operating point.

All walls are adiabatic. In the impeller, the periodic connection upstream of the blade passage, between the suction side of block 1 and the pressure side of block 2, is non-matching for improved mesh orthogonality. The wall velocity condition therefore is area-defined: both the hub and the shroud walls are rotating between z = 0 (shortly upstream of the impeller leading edge) and r = 122.5 mm (the impeller trailing edge radius), and non-rotating upstream and downstream of this region.

At the inlet, the velocity vector and static temperature are imposed. The density is extrapolated. The imposed temperature is that at station 1, $T_{in} = 24.4$ K. The inlet of the computational domain consists of a straight cylindrical duct. The inlet radial velocity is therefore set to zero. The inlet absolute tangential velocities at hub and tip are taken from specified velocity triangles, with a linear radial distribution.

At the exit, the static pressure at station 5, 80.7 bar, is imposed at mid-height ($r = 65$ mm), in conjunction with radial equilibrium. Exit pressures as low as 70 bar and as high as 200 bar have also been tested successfully.

At the inlet, the three cylindrical absolute velocity components and static temperature are im-
posed, the density is extrapolated. The boundary internal energy and the boundary pressure are then determined from the ETR and PER tables, in this order.

At the outlet, the static pressure is imposed and the three cylindrical absolute velocity components and absolute total energy, $E$, are extrapolated. The boundary density and boundary static temperature are then interpolated from the REP and TER tables, respectively.

The four impeller blocks and the diffuser block are connected through a quasi-steady rotor-stator interface which conserves the pitch-averaged inviscid fluxes for each tangential row of cells. Under-relaxation of the pitch-averaged interface flow state is not required for this test case.

8.3.4.4 Numerical parameters

Initial solution
The LH2 calculations can be started from a uniform initial solution, if full multigrid (FMG) is activated. (Starting without FMG has not been tested.) An initial relative velocity of 100 m/s fitted to the streamwise ($K$) mesh lines is specified. The combination of specified thermodynamic variables is selected through the expert parameter INIVAP. Internal energy and density are chosen (INIVAP = 4) and given as input through the expert parameter VAPFRE. The specified values of 0 (zero) J/kg and 50 kg/m³ produce uniform pressure and temperature initial fields of 74.3 bar and 46.5 K.

Spatial discretization
The standard second order central scheme is used for spatial discretization, with the default values of the second ($VIS2 = 1$) and fourth order artificial dissipation coefficients ($VIS4 = 0.1$). The switch function is based on the pressure only, which ensures high accuracy in the boundary layers. In the TPH application, frictional heating in the boundary layers is a dominant effect. Because of the resulting strong wall-normal temperature and density gradients, a temperature- or density-based switch might lead to activation of the second order artificial dissipation there, which clearly is not advisable as it may compromise the prediction of skin friction, heat release, and wall temperature, and of heat transfer rates if isothermal walls are present. Fortunately therefore, it was found that additional dissipation switches are not required for LH2 calculations. (A switch based on pressure and density has been experimented with in an attempt to overcome initial convergence problems.)

As most CFD techniques, the standard pressure switch of the central scheme was originally developed for the perfect gas (by Jameson et al., 1981, for aerofoil applications), for which the pressure is always positive, and pressure gradients in a given solution scale with the global pressure level. For liquids and thus for LH2, by contrast, pressure gradients in a first approximation are independent of the pressure level. Worse yet, if, as in the present non-cavitating fluid model, metastable states and negative pressures are admitted, the local average pressure in the denominator of the switch function may vanish completely, leading to uncontrolled, arbitrarily large pressure switches. The following modified formulation is therefore used for LH2,

$$v_i = \frac{\left| p_{i-1} - 2p_i + p_{i+1} \right|}{\max(4p_{\text{ref}}, \left| p_{i-1} \right| + 2\left| p_i \right| + \left| p_{i+1} \right|)}$$  \hspace{1cm} (8.15)

For the presented LH2 TPH calculations, a value of 10 bar was used for the constant reference pressure in the denominator, $p_{\text{ref}}$.

Time integration and convergence acceleration
Time integration is accomplished by the explicit four-stage Runge-Kutta scheme, with the default coefficients for the central scheme (0.125, 0.306, 0.587, and 1). Convergence to the steady state solution is accelerated through local time stepping, implicit residual smoothing with a time step dependent coefficient, and multigrid on four grid levels in a V-cycle with 1-4-16-96 sweeps. The CFL number is 3.

The speed of sound is required to calculate the spectral radius of the inviscid flux jacobian, which enters in several places of the numerical algorithm. In the presented LH2 calculations, the speed of sound table provided by Snecma DMF has not been used. Rather, the speed of sound has
been calculated from the PER table. This yields an exact, thermodynamically consistent speed of sound for LH2, also in the extrapolated low pressure region and beyond the limits of the core table.

**Expert parameters**

A minimum velocity of 1 m/s is specified for mass weighting in the calculation of the pitch-averaged solution.

### 8.3.4.5 Performed calculations

To demonstrate the effect of using the true thermodynamic properties of LH2, four calculations are presented: two calculations with LH2 at different Reynolds numbers and two incompressible reference calculations at these same Reynolds numbers.

Following Cumpsty (1989) and Strub et al. (1987), the Reynolds number is defined as $\text{Re} = \frac{U_2 b_2}{\nu}$, where $U_2$ is the impeller tip speed, $b_2$ is the impeller tip width (the span at impeller exit, as determined from the mesh), and $\nu$ the kinematic viscosity. Table 8.7 summarizes the relevant geometric and operational parameters.

The existing mesh, through the clustering in the boundary layers, is adapted to the Reynolds number of the air perfect gas calculations for which it was originally created, which is approximately 150 000.

It was the initial idea therefore to perform the LH2 calculations at this same Reynolds number. These calculations will be referred to as “low Reynolds number” (not to be confused with the low Reynolds $k-\varepsilon$ turbulence model — all calculations use the algebraic Baldwin-Lomax model). The respective columns in Table 8.8 list the chosen fluid properties for both the LH2 and the incompressible reference calculations. Values in plain font are imposed as constants, while values in italics depend on the variable thermodynamic and transport properties of LH2. A constant kinematic viscosity of 5.0E–5 m²/s has been chosen. With a mean density of 70.5 kg/m³ (read from the solution at the end of the calculation), this yields a dynamic viscosity of about 3.525E–3 Pa s and with Table 8.7 a Reynolds number of 144 271. To preserve also thermal similarity with the original air perfect gas calculations, the same constant Prandtl number of 0.72 was chosen. Combined with an also constant specific heat of 10 000 J/kg/K (a representative value for LH2 at the thermodynamic states occurring in the TPH) and the dynamic viscosity, this yields a thermal conductivity of 48.96 W/m/K.

In the corresponding incompressible reference calculation, all these parameters are essentially the same and strictly constant. The only difference is that the density remains constant. In order to be able to trace differences between the incompressible and LH2 calculations as the flow evolves through the TPH stage, the LH2 inlet density of 68.1 kg/m³ was chosen.

This low Reynolds number LH2 calculation overestimates the mass flow by 1.8 % (45.42 kg/s instead of 44.62 kg/s), which means that there must be a corresponding error in density, since the inlet velocity is imposed. The density, in turn, is fixed by the imposed inlet temperature and the pressure rise across the stage, the exit pressure being again imposed. The pressure rise is indeed found to be underpredicted by 12.3 % (56.39 bar instead of 64.3 bar), resulting in an inlet static pressure which is wrong by 48.5 % (24.35 bar instead of 16.4 bar).

<table>
<thead>
<tr>
<th>Rotational speed</th>
<th>$\Omega$ [rpm]</th>
<th>33 000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller trailing edge radius</td>
<td>$r_2$ [mm]</td>
<td>122.50</td>
</tr>
<tr>
<td>Impeller tip (outlet) width</td>
<td>$b_2$ [mm]</td>
<td>17.04</td>
</tr>
<tr>
<td>Impeller tip (outlet) speed</td>
<td>$U_2$ [m/s]</td>
<td>423.33</td>
</tr>
</tbody>
</table>

Table 8.7: Characteristic geometric and operational parameters of the TPH impeller.
Two hypotheses were examined in an attempt to improve the LH2 solution. The first concentrated on work input and the inlet velocity profiles. The incongruity observed in the inlet velocity triangles provided by Snecma DMF (tip triangle outside flow path) suggested that the inlet tangential velocity might not be correct, leading to a wrong amount of work input and thus pressure rise. Assuming incompressible flow and unchanged impeller exit flow angle, conservation of rothalpy indicates a reduction of $V_{\theta}^{in}$ by 17.5 m/s.

The second hypothesis incriminated the low Reynolds number. A calculation was therefore performed with the Snecma DMF viscosity and conductivity tables and thus the true temperature- and pressure-dependent transport properties of LH2, knowing however that on the existing mesh the boundary layers would be poorly resolved. These calculations (LH2 and corresponding incompressible reference) are referred to as “high Reynolds number”. Representative values of the dynamic viscosity and thermal conductivity, taken at 45 bar and 26 K, are 1.2E–5 Pa s and 0.12 W/m/K, respectively. This particular data point was chosen, after exploration of the region of interest with the interactive table evaluation tool, because it yields round, representative viscosity and conductivity values. With LH2, they only serve to estimate the Reynolds number for which, in conjunction with a mean density of 70 kg/m$^3$ (again read from the solution at the end of the calculation) and the length and velocity scales of Table 8.7, a value of 4.208E+7 is obtained. This is about 290 times the low Reynolds number. On the existing mesh, $y^+$ values of the first inner cell center are of the order of 1 at the low Reynolds number and of the order of 50 at the high Reynolds number.

The high Reynolds number incompressible reference calculation again uses constant properties, listed in Table 8.8. Dynamic viscosity and thermal conductivity are set to the respective LH2 reference values, $c_p$ is left unchanged at 10 000 J/kg/K from the low Reynolds number case, which implies a Prandtl number of 1. In the absence of a $c_p$ table, this constant $c_p$ is also used with LH2 to calculate the turbulence contribution to the apparent thermal conductivity.

The effect of the two modifications clearly validates the Reynolds number hypothesis. Reducing the inlet tangential velocity by the determined amount does reduce the mass flow, but not nearly enough (from 45.42 kg/s to 45.28 kg/s). This option was therefore not considered further. With the correct viscosity and thermal conductivity, on the other hand, the target mass flow is predicted almost exactly. Results at both the low and the high Reynolds number have been analyzed in detail and are presented in the following.

### 8.3.4.6 Convergence histories

At the low Reynolds number, the LH2 calculation converges to machine accuracy in 600 fine grid iterations. With the new multigrid strategy (new for this test case and for LH2), the mass flow is fully stabilized after only 300 fine grid iterations. The inlet-exit mass flow discrepancy is below 0.1
% in all four cases.

At the high Reynolds number, too, convergence rates with LH2 and with incompressible flow are similar. Figure 8.19 shows that the non-preconditioned compressible LH2 calculations converge at nearly the same rate as the preconditioned incompressible calculations. The latter employ the default numerical settings (preconditioning parameter 3, Rademark-Rossow residual smoothing with coefficient 2) and use four multigrid levels with 1-2-4-8 sweeps in a V-cycle. All other numerical parameters and boundary conditions are the same as for LH2.

The LH2 calculations do, however, require a more expensive multigrid strategy (1-4-16-96 sweeps instead of 1-2-4-8 sweeps incompressible). To assess the cost impact of the LH2 tables, one LH2 and one incompressible reference calculation have therefore been carefully timed. A number of iterations were performed on a dedicated machine (a 500 MHz DEC Alpha) and wall clock time read off periodically. Since between 95 and 99% of CPU time was consumed by Euranus, wall clock time may be substituted for CPU time. Because of the different multigrid strategies, one LH2 iteration costs 74% more than one incompressible iteration (1573 versus 905 CPU seconds for 60 iterations).

With the 1-4-16-96 strategy of the LH2 calculations, the cost of one multigrid cycle is 2.69 times that of a single grid iteration (estimated algebraically from the number of cells and the number of sweeps on the respective grid levels), while the 1-2-4-8 strategy of the incompressible calculations costs 1.64 times more. The two multigrid strategies therefore differ by a factor of 1.64 (= 2.69/1.64). To determine the penalty for interpolating the fluid properties from tables, the incompressible CPU times have been scaled up by this factor to the LH2 multigrid strategy. The cost increase due to the tables is found to be only 6%, compared with an incompressible fluid (1573 versus 1484 CPU seconds for 60 iterations).

On the 350,000-point mesh of the TPH test case, 1000 iterations with LH2 take approximately 8 hours, including time for input and output (on a dedicated 500 MHz DEC Alpha).

8.3.4.7 Table usage and void fraction

Table usage

As explained in Section 8.3.4.1, the LH2 thermodynamic interpolation tables deliberately only cover a limited range in their respective independent variables. In such a situation, it is recommended to verify that calculated solutions do not extend too far beyond the table limits, where extrapolation is used and where the accuracy will therefore decrease.

A straightforward way of getting a first, visual impression of table usage is to flag all table cells from which values have actually been interpolated, which produces plots such as that shown in Figure 8.21c for the PER table. Although the covered regions usually are quite small for the final converged solution, they may be larger while converging, as indeed suggested by the discussed requirement for smooth extrapolation behavior.

Table 8.9 lists the minimum and maximum values of pressure, temperature, and density occurring in the two LH2 solutions. The extremal Mach numbers and sound speeds are also shown. The variable ranges spanned at the two tested Reynolds numbers do not differ dramatically. Both calculations combined, pressures range from 1.4 bar to 109.3 bar, temperatures from 23.8 K to 39.4 K, and densities from 55.9 kg/m³ to 75.1 kg/m³. Pressure, temperature, and internal energy (not shown in Table 8.9) are comfortably contained within their respective table limits. The only variable which (in the steady-state solution) actually uses a significant part of its table range is the density. The LH2 thermodynamic data provided by Snecma DMF covers densities between 50 kg/m³ and 75 kg/m³. The lowest densities found in the solutions are 55.9 kg/m³ and 60.7 kg/m³ at the low and high Reynolds numbers, respectively, which is clearly above the lower table limit. The highest solution densities, however, which occur around the diffuser leading edge, do actually reach the upper table limit, remaining just below (74.9 kg/m³ at the high Reynolds number) or even exceeding it slightly (75.1 kg/m³ at the low Reynolds number).

It must be emphasized here that mildly out-of-range values are not by themselves a problem, neither for convergence of the numerical solution algorithm, nor for global solution accuracy. In the interpolation tables that have been generated from the Snecma LH2 thermodynamic data, out-of-
range points are extrapolated, approximately (but not exactly) linearly. Even at Re 150 000, only a few tiny cells (< 10) do actually have out-of-range densities.

**Table 8.9: Minimum and maximum values of thermodynamic variables and Mach numbers occurring in the steady state LH2 solutions at the two tested Reynolds numbers.**

<table>
<thead>
<tr>
<th></th>
<th>Low Reynolds number</th>
<th></th>
<th>High Reynolds number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Min</td>
<td>Max</td>
<td>Min</td>
</tr>
<tr>
<td>Pressure p [bar]</td>
<td>8.9</td>
<td>109.3</td>
<td>1.4</td>
</tr>
<tr>
<td>Temperature T [K]</td>
<td>23.8</td>
<td>39.4</td>
<td>24.1</td>
</tr>
<tr>
<td>Density ρ [kg/m³]</td>
<td>55.9</td>
<td>75.1</td>
<td>60.7</td>
</tr>
<tr>
<td>Absolute Mach number</td>
<td>M_{abs}</td>
<td>0</td>
<td>0.46</td>
</tr>
<tr>
<td>Relative Mach number</td>
<td>M_{rel}</td>
<td>0</td>
<td>0.50</td>
</tr>
<tr>
<td>Speed of sound c [m/s]</td>
<td>839</td>
<td>1403</td>
<td>595</td>
</tr>
</tbody>
</table>

An analogous observation holds for low and possibly negative pressures. Theoretically, there is no intrinsic lower limit on pressure other than the condition that the speed of sound remain positive. The possibility to perform calculations at negative pressures has been convincingly demonstrated with a symmetric converging-diverging nozzle, using LH2. The chosen boundary conditions yielded pressures between ~20 bar (at the throat) and +10 bar (at inlet and exit). Temperatures were of the order of 20 to 25 K and densities ranged between 65 and 70 kg/m³. Convergence was identical to an analogous computation at all-positive pressure.

**Void fraction**

Although these first LH2 calculations do not consider two-phase flow and cavitation, they nonetheless provide some information on the cavitation risk through the void fraction of the equilibrium mixture defined by the two independent thermodynamic variables of the solver, e and ρ, for which an interpolation table can be generated as for any other thermodynamic variable. To allow a quick, convenient assessment of the cavitation risk in the pitch-averaged solution, the void fraction is first calculated in the 3D field and then averaged circumferentially (as opposed to being calculated consistently from the 2D pitch-averaged fields of density and internal energy).

In the TPH test case, the lowest pressures occur at the impeller leading edge. At the operating point specified by Snecma DMF, they do, however, remain positive everywhere, staying even above the vapor pressure: in the low pressure region at the impeller inlet, the void fraction is strictly zero.

**8.3.4.8 Global performance parameters**

Table 8.10 collects global performance parameters for the four cases. At Re 150 000, the LH2 calculation underestimates the mass flow by 1.8 %, which had prompted the additional calculations with the true viscosity of LH2 at Re 40 000 000. In the incompressible reference calculations, the inlet density is fixed at the correct value and the correct mass flow obtained always, independently of the Reynolds number. The mass flow deviation ∆m/˙m is the difference between the calculated and exact mass flows, not between the inlet and exit mass flows. There also is a tiny difference in the low Reynolds incompressible case, due to artificial viscosity and the presence of the non-matching periodic connection and the quasi-steady rotor-stator interface. At Re 40 000 000, the LH2 calculation predicts the mass flow of the operating point almost exactly (44.58 kg/s instead of 44.62 kg/s, which is a deviation of only ~0.09 %).

The impeller shaft power has been estimated in two ways, from the total enthalpy rise and from
the torque. Table 8.10 lists both the torque and the total enthalpy rise between stations 1 and 5, as well as the power calculated according to the two definitions. Unfortunately, calculation of the torque cannot yet take into account area-defined rotating wall boundary conditions. The hub and shroud end walls of the impeller blocks have each been declared as one boundary condition patch and the rotating part which physically belongs to the impeller delineated by an area-defined rotation speed. To avoid any influence from the wall shear stresses upstream and downstream of the impeller, the end walls have been omitted altogether; the torque contains the viscous and pressure contributions from the impeller blades only.

The predicted powers according to the enthalpy definition span a range of 2.6 %, those according to the torque definition a range of 3.4 %. Neglecting the end wall contribution to the torque results in a slightly lower power than with the enthalpy definition, at the low Reynolds number by 3.0 % (LH2) and 2.8 % (incompressible), at the high Reynolds number by 1.5 % (LH2) and 1.2 % (incompressible).

Irrespective of the definition and of the Reynolds number, LH2 consistently predicts a higher power than the incompressible fluid, at low Reynolds number by 2.6 % (enthalpy definition) and 2.4 % (torque definition), at high Reynolds number by 1.0 % (enthalpy definition) and 0.7 % (torque definition). The reason is to be sought mainly in the fact that in the incompressible calculations the density is kept constant at the inlet value, while in the LH2 calculations it increases as the hydrogen is compressed. At Re 150 000, the different mass flow with LH2 also has an impact (ΔH is higher by only 0.9 %).

The most accurate estimate for shaft power is 4.814 MW (LH2, high Reynolds number, enthalpy definition, judged from the fluid properties and power definition).

### Table 8.10: Global results for the TPH demonstration LH2 and incompressible reference calculations at the two tested Reynolds numbers.

<table>
<thead>
<tr>
<th></th>
<th>Low Reynolds number</th>
<th>High Reynolds number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>LH2</td>
<td>Incompr.</td>
</tr>
<tr>
<td>Mass flow m [kg/s]</td>
<td>45.42</td>
<td>44.64</td>
</tr>
<tr>
<td>Mass flow deviation Δm</td>
<td>m [kJ/kg]</td>
<td>+1.8</td>
</tr>
<tr>
<td>Torque M [Nm]</td>
<td>1 358</td>
<td>1 325</td>
</tr>
<tr>
<td>Power, definition 1 P = m ΔH [MW]</td>
<td>4.840</td>
<td>4.714</td>
</tr>
</tbody>
</table>

#### 8.3.4.9 Section-averaged variables at five data stations

Tables 8.11 through 8.14 compare section-averaged thermodynamic variables at the five stations defined in Figure 8.17a. In the following, pressure, temperature, and density will each be discussed in turn, discrepancies with the Snecma DMF data pointed out, and both the numerical solutions and the Snecma DMF data checked for thermodynamic consistency.

**Pressure**

Table 8.11 lists the static pressure at stations 1 through 5, as well as the static pressure rise between stations 1 and 5, comparing the four calculations with data communicated by Snecma DMF. To facilitate analysis of the results, for each calculated value the percentage error compared with the respective Snecma value is shown as a subscript.

Station 5 is located very close to the exit boundary and the section-averaged pressure therefore equal to the imposed exit pressure (to within 0.05 % in all four cases).
Regarding the static pressure rise across the stage, it has already been seen that at Re 150 000 this is substantially underestimated, by 12.3% with LH2 (56.39 bar instead of 64.3 bar) and even by 16% incompressible (54.03 bar). In terms of predicted inlet static pressure, the relative error is even larger, 48.5% with LH2 (24.35 bar instead of 16.4 bar) and 62.8% incompressible (26.70 bar). At Re 40 000 000, by contrast, the LH2 calculation predicts the stage pressure rise nearly exactly (64.32 bar instead of 64.3 bar, which corresponds to a relative error of 0.03%), while the incompressible reference calculation falls short by 5.5% (60.78 bar). In terms of inlet static pressure, the respective errors are only 0.06% for LH2 (16.41 bar instead of 16.4 bar), but 21.4% incompressible (19.91 bar).

We wish to emphasize that the excellent agreement with LH2 is incidental and not the result of any kind of tuning of parameters. On a finer mesh and with another turbulence model, a slightly different solution would probably be obtained which might agree less well with the data. Moreover, focusing on the LH2 calculation at the correct (high) Reynolds number, it is interesting to observe that while the prediction of overall pressure rise is indeed perfect, such good agreement is not found for all intermediate stations. At station 2 (in the gap between the impeller and the diffuser, near the diffuser leading edge, Figure 27c), pressure is underpredicted by 5.1% (64.64 bar instead 68.1 bar), at station 3 (the diffuser trailing edge) overpredicted by 1.3% (80.52 bar instead of 79.5 bar).

Since 85x799

Table 8.11: Section-averaged static pressure at five stations for LH2 and incompressible TPH calculations at two Reynolds numbers (subscripts = percentage error with respect to Snecma data).

<table>
<thead>
<tr>
<th>Station</th>
<th>Pressure, p [bar]</th>
<th>Snecma</th>
<th>Low Reynolds number</th>
<th>High Reynolds number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>LH2 Inc.</td>
<td>LH2 Inc.</td>
</tr>
<tr>
<td>Station 1</td>
<td>16.4</td>
<td>24.35 +48.5</td>
<td>26.70 +62.8</td>
<td>16.41 +0.06</td>
</tr>
<tr>
<td>Station 2</td>
<td>68.1</td>
<td>70.86 +4.0</td>
<td>71.69 +5.3</td>
<td>64.64 -5.1</td>
</tr>
<tr>
<td>Station 3</td>
<td>79.5</td>
<td>80.65 +1.4</td>
<td>80.21 +0.89</td>
<td>80.52 +1.3</td>
</tr>
<tr>
<td>Station 4</td>
<td>79.7</td>
<td>80.26 +0.70</td>
<td>79.80 +0.13</td>
<td>79.78 +0.10</td>
</tr>
<tr>
<td>Station 5</td>
<td>80.7</td>
<td>80.74 +0.05</td>
<td>80.73 +0.04</td>
<td>80.73 +0.04</td>
</tr>
<tr>
<td>Static pressure rise</td>
<td>64.3</td>
<td>56.39 -12.3</td>
<td>54.03 -16.0</td>
<td>64.32 +0.03</td>
</tr>
</tbody>
</table>

The excellent agreement with LH2 is incidental and not the result of any kind of tuning of parameters. On a finer mesh and with another turbulence model, a slightly different solution would probably be obtained which might agree less well with the data. Moreover, focusing on the LH2 calculation at the correct (high) Reynolds number, it is interesting to observe that while the prediction of overall pressure rise is indeed perfect, such good agreement is not found for all intermediate stations. At station 2 (in the gap between the impeller and the diffuser, near the diffuser leading edge, Figure 27c), pressure is underpredicted by 5.1% (64.64 bar instead of 68.1 bar), at station 3 (the diffuser trailing edge) overpredicted by 1.3% (80.52 bar instead of 79.5 bar).

Table 8.12 lists the static temperature at the five test stations as provided by Snecma DMF and as obtained in the four calculations. The most instructive variable for a direct comparison is the temperature rise which has therefore been added as a subscript to each column. For each case, the zero point is set independently at station 1 (where the temperature is still nearly equal to the imposed inlet value of 24.4 K, the maximum calculated value being 24.45 K, an increase of 0.2%).

Between stations 1 and 5, the temperature increases by 4.9 K, or 20.1%, according to the Snecma data. Two distinct mechanisms are responsible for this increase. First, the LH2 is heated by friction. With an incompressible fluid and adiabatic flow, this is the only possible mechanism. For a substance with non-negligible compressibility, such as LH2, the adiabatic compression of the fluid as it passes through the machine constitutes a second mechanism.

At Re 150 000, the LH2 calculation predicts a global temperature rise of 5.11 K, which comes to within 4.3% of the Snecma data. The incompressible calculation, on the other hand, foresees an increase by only 2.39 K, which falls short of the Snecma data by 51.2%. Even at this low Reynolds number, therefore, the effect of compressibility on temperature is significant, and must not be neglected if thermal processes are to be predicted with any degree of accuracy. In fact, it may be concluded from the two calculated exit temperatures (LH2 and incompressible) that the two effects are of equal strength, each viscous friction and adiabatic compression contributing about 50% to the static temperature rise.

At Re 40 000 000, friction work is reduced, so that the gap between LH2 and the incompressible fluid becomes narrower.
Condensable Fluids

A condensable fluid can be expected to widen and this is indeed what one finds. In the LH2 calculation, the temperature increases by 4.59 K (minus 6.3 % compared with the Snecma value of 4.9 K), while it rises by only 1.32 K if the fluid is assumed to be incompressible (minus 73.1 % compared with Snecma). With the true viscosity of LH2, therefore, friction work accounts for only about 30 % of the global static temperature rise. The remaining 70 % are caused by adiabatic (reversible) compression, an effect that cannot be captured with an incompressible fluid model.

Regarding the small deviation of our high Reynolds number LH2 calculation from the Snecma data, we observe that the latter presumably have not been determined experimentally, but are themselves the result of some form of computation with a consequent uncertainty. Furthermore, the mesh admittedly is not optimal and the poor resolution of the boundary layers might in part explain the small underprediction with LH2. In any case, this difference pales when compared with the gross error that would be committed if an analysis of the temperature field were attempted on the basis of an incompressible simulation; it is not possible to predict the temperature field unless the thermodynamic properties of LH2 are correctly taken into account.

**Density**

Table 8.13 lists the density calculated at stations 1 through 5 with LH2 at the two Reynolds numbers. Subscripts denote the percentage error with respect to the Snecma data. In the two incompressible reference calculations the density itself is of course constant, but not the errors, which increase as the LH2 is gradually compressed. There is, however, strictly no difference between the two Reynolds numbers, because the error is defined on the Snecma data, not the respective LH2 values. The two incompressible columns thus are strictly identical and repeated only for visual consistency with the other tables.

According to the Snecma data, the density increases by 4.6 % from 68.1 kg/m$^3$ at the inlet to 71.2 kg/m$^3$ at the exit. The maximum section-averaged density does not, however, occur at the exit, but rather at station 2, the diffuser inlet, where 72.5 kg/m$^3$ are attained, an increase of 6.5 % over the inlet value.

At Re 150 000, the LH2 calculation faithfully respects this trend, predicting a maximum density of 71.66 kg/m$^3$ at station 2 (minus 1.2 % compared with the respective Snecma value), with a subsequent monotonous decline to 70.95 kg/m$^3$ at the exit, station 5 (minus 0.35 % error). Due to excessive friction losses, the global pressure rise at the low Reynolds number is severely underpredicted, as was seen in Table 8.11, meaning that the density must come out too high, since the inlet temperature is imposed. With 69.24 kg/m$^3$, the inlet density is indeed overestimated by 1.7 %.

At Re 40 000 000, meanwhile, the maximum density with LH2 occurs at station 3, where a value of 71.78 kg/m$^3$ is reached (plus 0.25 % compared with the respective Snecma value of 71.6 kg/m$^3$). The density of 71.19 kg/m$^3$ predicted at station 2 falls short of the respective Snecma value of

<table>
<thead>
<tr>
<th>Temperature, T [K]</th>
<th>Snecma</th>
<th>Low Reynolds number</th>
<th>High Reynolds number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>LH2 Inc.</td>
<td></td>
</tr>
<tr>
<td>Station 1</td>
<td>24.4 0.0</td>
<td>24.43 0.0</td>
<td>24.45 0.0</td>
</tr>
<tr>
<td>Station 2</td>
<td>26.4 2.0</td>
<td>27.62 3.19</td>
<td>25.35 0.90</td>
</tr>
<tr>
<td>Station 3</td>
<td>28.6 4.2</td>
<td>28.89 4.46</td>
<td>26.16 1.71</td>
</tr>
<tr>
<td>Station 4</td>
<td>28.7 4.3</td>
<td>29.20 4.77</td>
<td>26.51 2.06</td>
</tr>
<tr>
<td>Station 5</td>
<td>29.3 4.9</td>
<td>29.54 5.11</td>
<td>26.84 2.39</td>
</tr>
</tbody>
</table>

Error at station 5 [%] +4.3 -51.2 -6.3 -73.1
72.5 kg/m³ by 1.8 %. The corresponding errors with an incompressible fluid are minus 6.1 % at station 2 and minus 4.4 % at station 5.

Recognizing that the five test stations actually are located at only three different pressure levels, namely the inlet (station 1), the gap between the impeller and the diffuser (station 2), and the approximately isobaric return system (stations 3, 4, and 5), to ensure a fair comparison the mean density error given in Table 8.13 is calculated from the signed station errors $\varepsilon_i, i = 1, \ldots, 5$, as

$$
\bar{\varepsilon} = \frac{|\varepsilon_1| + |\varepsilon_2| + |\varepsilon_3| + |\varepsilon_4| + |\varepsilon_5|}{3} \quad (8.16)
$$

With this definition, the mean errors are 1.06 % for LH2 low Re, 0.71 % for LH2 high Re, but 3.6 % incompressible. On an element (blade row) basis, giving equal weight to each station and twice counting station 2 (impeller TE and diffuser LE), the mean errors would be 0.83 % for LH2 low Re, 0.74 % for LH2 high Re, and 4.38 % incompressible. Thus, from a purely mechanical point of view, too, an incompressible fluid is a bad approximation for liquid hydrogen submitted to substantial pressure variations.

**Disagreement at station 2**

The LH2 density evolution depends on the balance between the imposed pressure variation and the temperature increase subsequent to frictional heating. It is therefore most peculiar that the qualitative (and quantitative) behavior downstream of station 2 should better agree with the Snecma data at $\text{Re} = 150,000$ than at $\text{Re} = 40,000,000$.

At this station, the agreement with the data is significantly worse than at all other stations, for all variables. Focusing on the most plausible calculation only, LH2 at the high Reynolds number, and synoptically viewing pressure, temperature, and density in Tables 8.11, 8.12, and 8.13, respectively, at station 2 the pressure is found to be off by 5.1 % (6.7 % in terms of the pressure rise, $\Delta p$), the temperature by 3.8 % (50.5 % in terms of $\Delta T$), and the density by 1.8 % (28.4 % in terms of $\Delta \rho$). At the other four stations, the relative errors, both in absolute and in increase terms, generally are smaller by about one order of magnitude.

It should be mentioned here that in the actual combination of the TPH impeller with the return channel the increase in flow path width of 7.9 % between the impeller and the diffuser abruptly occurs at the impeller trailing edge, and that in view of keeping the mesh simple this sudden increase was replaced by a linear transition over the first third of the vaneless space between the two components, cf. Figures 8.17a and 8.18b.

**Thermodynamic consistency**

To put the preceding analysis in perspective, the thermodynamic consistency of both the Snecma data
and the section-averaged solution data for LH2 at the two Reynolds numbers has been checked with
an interactive table interrogation tool. Table 8.14 shows the result of this test. The consistency of
pressure, temperature, and density was verified at the five data stations. Since only two thermody-
namic variables are independent, any additional variable known from an independent source can be
used as a consistency criterion. A table that directly links $p$, $T$, and $\rho$ was not available, however, so
that the check had to pass through one intermediate table. With the available tables, the first step is
to calculate the internal energy from temperature and density using the ETR table. One may then
compare either pressure interpolated from the PER table, or density interpolated from the REP table.
Pressure was chosen because it is the more sensitive variable (varies more strongly) and because PER
is the fundamental table in the numerical solution method. For the LH2 section-averaged solution da-
ta, the described test would reproduce all four recorded digits exactly, if the REP table were used and
the density compared.

In Table 8.14, the column titled $p$ contains the original pressure (from Snecma data or from
the numerical solution), the column titled $p(T, \rho)$ the pressure interpolated from the LH2 tables as
$p(e(T, \rho), \rho)$. The largest consistency error in both the Snecma data and the LH2 solution occurs at
the inlet, station 1. However, where it is only 0.21 % and 0.12 % in the LH2 solutions, it reaches 2.2
% in the Snecma data. Further significant consistency errors in the Snecma data occur at stations 3
(diffuser exit, 0.99 %) and 4 (return vane inlet, 1.32 %). On average, the pressure thermodynamic
consistency error in the Snecma data is about 1 % (arithmetic mean of the absolute percentage errors
at the five stations), while it is around 0.1 % for the LH2 section-averaged data.

Additional cross-checks between the LH2 interpolation tables involving pressure, temperature,
density, and internal energy showed that, for these four variables and at the five data stations, they
reproduce four digits exactly. The (small) consistency errors in the LH2 averaged values therefore
are caused by solution non-uniformity over the cross-sections. (Velocity, density, pressure, and total
energy are mass averaged circumferentially and temperature and internal energy calculated consist-
ently for this 2D field. Each variable of interest is then mass averaged individually in the spanwise
direction.) These non-uniformities are stronger at $Re = 150,000$ than at $Re = 40,000,000$, which explains
why the thermodynamic inconsistency is slightly larger at the low Reynolds number.

<table>
<thead>
<tr>
<th>Station</th>
<th>$p$ [bar]</th>
<th>$p(T, \rho)$</th>
<th>$\Delta$ [%]</th>
<th>$p$ [bar]</th>
<th>$p(T, \rho)$</th>
<th>$\Delta$ [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Station 1</td>
<td>16.4</td>
<td>16.76</td>
<td>+2.20</td>
<td>24.35</td>
<td>24.30</td>
<td>–0.21</td>
</tr>
<tr>
<td>Station 2</td>
<td>68.1</td>
<td>68.04</td>
<td>–0.09</td>
<td>70.86</td>
<td>70.73</td>
<td>–0.18</td>
</tr>
<tr>
<td>Station 3</td>
<td>79.5</td>
<td>78.71</td>
<td>–0.99</td>
<td>80.65</td>
<td>80.57</td>
<td>–0.10</td>
</tr>
<tr>
<td>Station 4</td>
<td>79.7</td>
<td>78.64</td>
<td>–1.33</td>
<td>80.26</td>
<td>80.20</td>
<td>–0.07</td>
</tr>
<tr>
<td>Station 5</td>
<td>80.7</td>
<td>80.96</td>
<td>+0.32</td>
<td>80.74</td>
<td>80.71</td>
<td>–0.04</td>
</tr>
<tr>
<td>Arithmetic average</td>
<td></td>
<td></td>
<td>0.99</td>
<td></td>
<td></td>
<td>0.12</td>
</tr>
</tbody>
</table>

Table 8.14: Consistency check of $p$, $T$, and $\rho$ thermodynamic data for Snecma operational data and LH2
low and high Reynolds number section-averaged data at the five TPH data stations.
8.3.4.10 Global velocity field

At the operating point specified by Snecma DMF, which we presume to be the design point or close to it, all blade rows operate at low incidence. With one exception (see below), there appear to be no extended zones of separated flow. To illustrate this fact as well as to provide a global impression of the 3D velocity field, Figure 8.22 shows vectors of relative velocity on a blade-to-blade mesh surface at mid-span for the more realistic high Reynolds number case (LH2 calculation). The linear velocity profiles imposed at the inlet result in approximately zero incidence angle at the leading edge of both the impeller main and splitter blades, Figure 8.22a. The same holds for the diffuser blade, Figure 8.22b. In this same figure, the one recirculation zone is clearly visible on the extended pressure (convex) surface.

The only blade operating at a sizable non-zero incidence angle is the return vane, Figure 8.22c. As far as can be judged from this figure, the stagnation point seems to be located approximately at the junction of the leading edge radius and the pressure surface. According to our simulation, with its known deficiencies in mesh resolution, the flow nevertheless remains well attached to the suction surface. Outside of the end wall boundary layers, the velocity fields near the hub and near the shroud are similar those at mid-span.

Figure 8.22 also gives some indication on the degree of inadequacy of the low Reynolds number mesh for the high Reynolds number calculations, confirming that the situation is not everywhere as bad as might be feared, given the factor of about 290 between the two tested Reynolds numbers. In the leading edge region of the impeller main and splitter blades, a rudimentary boundary layer is resolved on both suction and pressure side, Figure 8.22a. The same is true of the leading edge region of the diffuser and return vane blades, Figures 8.22b and 8.22c.

The one exception to the absence of sizable regions of separated flow is the corner between the extended diffuser blade pressure (convex) surface and the hub wall. In Figure 8.22d, this recirculation has been made visible by means of streamlines and velocity vectors. Vectors are plotted on a blade-to-blade mesh surface at 20 % span. As determined on this (sub-optimal) mesh, the separation region would seem to cover the inboard half of the span and be about as high as it is wide.

8.3.4.11 Speed of sound and Mach number

The Mach number involves both a thermodynamic component through the speed of sound and a kinematic component through the velocity. Using the real thermodynamic properties of liquid hydrogen offers the possibility to obtain information on the actual sound speeds and resulting Mach numbers. Both are shown in Figure 8.22 for the high Re LH2 solution.

Figure 8.22a shows the speed of sound calculated from the pitch-averaged LH2 solution (colormap range from 1 045 m/s to 1 300 m/s). Comparison with the corresponding views of pressure (Figure 8.24b), temperature (Figure 8.25b), and density (Figure 8.25b) indicates that the speed of sound increases approximately linearly with the pressure (judged from the even spacing of the contour levels), with only a small influence from temperature variations.

In the presented LH2 solutions, the speed of sound varies between about 1 050 m/s at the inlet and about 1 300 m/s at the exit. It is calculated consistently from the PER table and therefore exactly represents the speed of sound seen by the flow. The jagged appearance of the contour lines results from piecewise constant partial derivatives on the PER table involved in this calculation. It has no adverse effect on convergence or solution accuracy. The same convergence rate and final residual drop were observed in calculations with a constant, uniform speed of sound. A constant, uniform speed of sound leads to less efficient dosage of artificial dissipation and precludes accurate information on Mach numbers.

Since the variation of the speed of sound is relatively small (±11 % or ±125 m/s about a mean value of 1 150 m/s), the relative Mach number essentially reflects the relative velocity (defined as relative to the respective blade row, thus absolute in stators). It is shown in Figure 8.22b for the pitch-averaged solution (colormap range between 0.06 and 0.235). The largest Mach numbers of about 0.23 occur in the vaneless space between the impeller and diffuser, due to the switch between the relative and absolute systems (absolute Mach number shown in the diffuser and return channel). In the im-
peller, a local maximum of about 0.2 occurs at the tip of the leading edge. The lowest Mach numbers are found inside the return vane, where free stream values drop to about 0.08.

The above observations without exception also apply to the actual 3D flow. Figure 8.22c shows the relative Mach number on a blade-to-blade mesh surface at mid-span. Maximum Mach numbers occur on the suction (concave) side of the diffuser blade, where at Re $40\,000\,000$ values of about 0.26 are reached. Comparing Figure 8.22c with its analogue at Re $150\,000$ (not shown) also allows the interesting observation that the low speed (separation) region at the back of the diffuser blade is larger at Re $40\,000\,000$ than at Re $150\,000$.

Average relative Mach numbers range between 0.1 and 0.2, with a global mean value of about 0.15. Such a Mach number level usually is adequate for a compressible calculation without preconditioning.

### 8.3.4.12 Pressure, temperature, and density evolution

Qualitatively, all four calculations predict essentially the same pressure distribution. The observed field is typical of any centrifugal pump and characterized by an almost exclusively streamwise gradient. In the spanwise direction, the pressure therefore appears uniform.

Quantitatively, on the other hand, the four cases differ substantially, and this has already been discussed in detail in Section 8.3.4.9, with the main conclusion that only the LH2 calculation at the correct, high Reynolds number predicts the same pressure rise as the Snecma validation data, the other three cases falling short by between 5.5 % and 16 %.

### 8.3.4.13 Influence of the Reynolds number

### 8.3.4.14 Kinematic effects

Figure 8.30a shows spanwise profiles of the meridional velocity at the diffuser vane inlet. Corresponding flow angle profiles at this as well as three other locations are given in Figure 8.31. An intuitive explanation for the smoother velocity (and flow angle) profiles observed with LH2 would be that the compressible fluid to some extent yields to a given pressure gradient which in an incompressible flow would cause more violent acceleration. On the other hand, when comparing flow response to a given smooth change in geometry on the basis of a stationary quasi-1D model, one finds stronger velocity (and pressure) gradients in a subsonic compressible flow, by a factor $1/(1-M^2)$, Table 8.15, $S$ denoting cross sectional area. Given the contradictory results, either of these interpretations probably is simplistic. Since differences in spanwise profiles are to be explained by a difference in the fluid thermodynamic model, the most obvious thing to look for are differences in the spanwise gradients of thermodynamic variables. The one variable which has a direct impact on the velocity field and which differs appreciably between the LH2 and incompressible solutions at the respective Reynolds numbers is the density, Figure 8.30b. A straightforward explanation based on continuity considerations that would link a density deficit to excess velocity fails, however, because the opposite is observed.

A plausible explanation for the observed spanwise velocity gradients in function of the spanwise density gradients is provided by the inviscid radial momentum equation. For steady axisymmetric flow with no axial component,
where \( V_r \) and \( V_\theta \) are the absolute cylindrical velocity components and where \( \beta \) is the relative flow angle. The second term on the right hand side is the radial component of a blade force which introduces the inviscid work input by the blades into the axisymmetric flow. It has been included for the sake of completeness so as to have a consistent throughflow model in the impeller but does not enter into the argument. \( V_\theta \) depends on \( V_r \) and \( \beta \) only.

Supposing that Equation (8.17) is satisfied by some solution with a spanwise uniform density distribution, we ask how a small local deviation of density from this flat baseline solution affects the radial velocity gradient. Noting that both \( \partial_x u \) and \( \partial_x p \) are positive (both variables are monotonously increasing through the impeller) and all other things being equal (in particular, the radial pressure gradient is determined by the mean density and therefore unchanged, and the pressure is spanwise constant), we see that a locally reduced density (as in the hot near-wall fluid) requires a reduced radial velocity gradient for Equation (8.17) to remain balanced, and vice versa. The spanwise profiles at impeller exit of pitch-averaged meridional velocity shown in Figure 8.30a fully agree with this analysis: at both end walls, the LH2 solutions (circles) have a velocity deficit, compared with the incompressible reference solutions (triangles), and the described effect conforms to the boundary layer thickness at the respective Reynolds number (larger at \( Re = 150 \ 000 \), open symbols, smaller at \( Re = 40 \ 000 \ 000 \), closed symbols). The expansion of the LH2 due to frictional heating therefore attenuates the boundary layer centrifugal effect in the impeller, through what appears to be an essentially inviscid mechanism.

In the LH2 solutions (circles), the density profiles are approximately mirror images of the temperature profiles, compare Figures 8.30b and 8.30c, a necessary consequence of the spanwise constant pressure, Figure 8.30d. In the high Re LH2 solution (closed circles), the spanwise density gradient is nearly zero in the vaneless space between the impeller and the diffuser where the cartesian plots of Figure 8.30 are drawn. This further corroborates our hypothesis on the origin of the different profiles shapes with LH2 and incompressible flow. Clearly, they cannot arise locally but must depend on the history of the flow in the impeller, where spanwise density gradients do occur at both Reynolds numbers, Figures 8.26b and 8.29c. A history mechanism also makes plausible how modest density variations can give rise to much larger velocity fluctuations.

Since spanwise pressure gradients are negligible everywhere in the impeller, Figure 8.24b, a barotropic fluid model would most likely produce the same spanwise velocity and flow angle profile shapes as an incompressible model and thus not offer any improvement in this respect. It follows therefrom that only a full LH2 thermodynamic model with two independent thermodynamic variables can predict correct spanwise flow angle and velocity profiles. Any simplified fluid model will make an error on the profile level or on the profile shape or on both.

---

**Table 8.15: Velocity and pressure gradients in steady quasi-1D incompressible and compressible flow.**

<table>
<thead>
<tr>
<th></th>
<th>Incompressible</th>
<th>Compressible</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \frac{\partial_x u}{\partial x} = )</td>
<td>( -u \frac{\partial_x S}{S} )</td>
<td>( \frac{u}{1 - M^2} \frac{\partial_x S}{S} )</td>
</tr>
<tr>
<td>( \frac{\partial_x p}{\partial x} = )</td>
<td>( \rho u^2 \frac{\partial_x S}{S} )</td>
<td>( \rho u^2 \frac{\partial_x S}{1 - M^2} )</td>
</tr>
</tbody>
</table>

where \( V_r \) and \( V_\theta \) are the absolute cylindrical velocity components and where \( \beta \) is the relative flow angle. The second term on the right hand side is the radial component of a blade force which introduces the inviscid work input by the blades into the axisymmetric flow. It has been included for the sake of completeness so as to have a consistent throughflow model in the impeller but does not enter into the argument. \( V_\theta \) depends on \( V_r \) and \( \beta \) only.

---

The equations governing the flow are:

\[
V_r \frac{\partial_r V_r}{\rho} + \frac{\partial_p}{\rho} = \frac{V_\theta^2}{r} = \tan \beta \frac{V_r}{r} \partial_r (r V_\theta)
\]  

(8.17)
8.3.4.15  3D blade-to-blade flow field
Figure 8.17  LH2 turbo pump: (a) components and data stations, (b) 3D view of the geometry (yellow = impeller, blue = vaned diffuser, red = deswirl cascade).

Figure 8.18  LH2 turbo pump, computational grid (355 542 points): (a) axial view of the surface mesh, (d) meridional projection of the mesh.
Figure 8.19  LH2 turbo pump (Re = 40 000 000), convergence histories: (a) density residual, (b) mass flow (red = LH2, blue = incompressible).

Figure 8.20  LH2 turbo pump: p(e,ρ) table for hydrogen with approximate range of thermodynamic states occurring in the steady state solution.
Figure 8.21  LH2 turbo pump: $p(e, \rho)$ table used for the presented calculations, (a) table range and interpolation behavior, (b) relative error according to random test, (c) thermodynamic states occurring in the steady state solution.

Figure 8.22  LH2 turbo pump (Re 40 000 000, exact properties of LH2): details of the mid-span blade-to-blade velocity field, (a) at the impeller leading edge, (b) at the impeller trailing edge and in the diffuser blade passage, (c) at the return vane leading edge; (d) at 20% span at the diffuser exit.
Figure 8.23 LH2 turbo pump (Re 40 000 000, exact properties of LH2): (a) speed of sound and (b) relative Mach number in the pitch-averaged solution, (c) relative Mach number on a blade-to-blade mesh surface at mid-span.

Figure 8.24 LH2 turbo pump, predicted static pressure field, azimuthal average: (a) evolution between inlet and exit along a mesh line at mid-span; color contours (b) with LH2 and (c) with an incompressible fluid.
Figure 8.25 LH2 turbo pump, predicted static temperature field, azimuthal average: (a) evolution between inlet and exit along a mesh line at mid-span, color contours (b) with LH2 and (c) with an incompressible fluid.

Figure 8.26 LH2 turbo pump, predicted density field, azimuthal average: (a) evolution between inlet and exit along a mesh line at mid-span, color contours (b) with LH2 and (c) with an incompressible fluid.
Figure 8.27  LH2 turbo pump, predicted static temperature field at Re 150 000, azimuthal average: (a) with LH2, (b) with an incompressible fluid.

Figure 8.28  LH2 turbo pump, predicted density field at Re 150 000, azimuthal average: (a) with LH2, (b) with an incompressible fluid.

Figure 8.29  LH2 turbo pump, pitch-averaged velocity field (a) at Re 150 000 and (b) at Re 40 000 000.
Figure 8.30  LH2 turbo pump, spanwise profiles at the diffuser vane inlet of (a) meridional velocity, (b) density, (c) temperature, (d) pressure (circles = LH2, triangles = incompressible, closed symbols - Re 40 000 000, open symbols = Re 150 000).
Figure 8.31  LH2 turbo pump, spanwise profiles of tangential flow angle: (a) at impeller exit, (b) at diffuser vane inlet, (c) at return vane inlet, (d) at stage exit (circles = LH2, triangles = incompressible, closed symbols - Re 40 000 000, open symbols = Re 150 000).
Figure 8.32  LH2 turbo pump, LH2 calculation at Re 40 000 000: color contours of static pressure on a blade-to-blade mesh surface at mid-span.

Figure 8.33  LH2 turbo pump (Re 40 000 000): color contours of static temperature on a blade-to-blade mesh surface at mid-span (a) for LH2 and (b) incompressible.

Figure 8.34  LH2 turbo pump (Re 40 000 000): color contours of density on a blade-to-blade mesh surface at mid-span (a) for LH2 and (b) incompressible.
Figure 8.35  LH2 turbo pump, LH2 calculation at Re 40 000 000: color contours of absolute total enthalpy on a blade-to-blade mesh surface at mid-span.
Chapter 9 Conclusions

General Summary
A Navier-Stokes solver representative of current CFD technology has been used as the basis for a number of new developments aimed at turbomachinery applications. Suitable boundary conditions have been developed, implemented, and tested, including notably a non-matching connecting boundary condition for structured-mesh solvers and a quasi-steady rotor-stator interface. A throughflow module based on the Euler equations has been developed from scratch, with special emphasis on numerical robustness and efficiency, and on the shock capturing properties. Finally, a long-standing problem regarding the treatment of condensable fluids in CFD codes has been solved through an efficient technology based on interpolation tables.

Basic Turbomachinery Capability
A 3D multiblock/multigrid Navier-Stokes solver originally developed for chemically reacting hypersonic external flows has been used as the basis for developments aimed at turbomachinery applications, preserving already implemented models, such as turbulence models, as well as the open software architecture which allows the code to easily accept new physical or numerical models, and to share these models between different classes of applications.

The advantages and drawbacks of fundamental choices concerning the steadily rotating relative frame of reference and the coordinate system have been weighed, leading to the decision to retain cartesian coordinates also for turbomachinery applications and to solve for relative velocities, except in propeller (external) flow applications.

From the literature as well as from our own systematic tests, skew meshes were recognized as a major source of loss of accuracy and one reason for poor convergence rates. In turbomachinery applications and with standard structured H-type meshes, skewness occurs primarily in the blade-to-blade surface due to the requirement that grid lines be continuous across the periodic boundaries upstream and downstream of the blade passage. The situation can be alleviated through the use of special mesh topologies, such as C- and O-meshes (and which the solver all of course supports), but these do not always constitute an optimal solution, because, unless fully automated, their generation may be more time consuming (which is an important consideration in an industrial environment), because multigrid conditions on the numbers of grid nodes may be more difficult to satisfy, and because it may not be possible to cluster the mesh along blade wakes. A more flexible solution is given by the admission of discontinuous grid lines. The novelty of the non-matching connecting boundary condition developed in this work consists in the semi-algebraic approach in which the boundaries of the connected faces must be continuous, but complete freedom is given with regard to the two connected surface meshes. Obtaining the requisite interpolation information from mapping of the surface meshes into a parametric space allows accurate connection of the stretched cells near curved solid walls, which, although logically connected, in the employed linear geometry model may not be congruent in physical space. For continuous grids, the usual matching connecting boundary condition is recovered. Inlet, exit, and solid wall boundary conditions were equally adapted to the special demands of turbomachinery applications, paying particular attention to the pressure gradient normal to solid surfaces. The basic turbomachinery capability was validated with a battery of simple test cases.

When the present work commenced, computing power had reached a level which allowed to contemplate the simultaneous calculation (in a quasi-steady manner) of two or more blade rows in the design cycle. The advantage over the calculation of isolated blade rows is that boundary conditions need to be provided only at inlet and exit of the computational domain, with individual blade rows being coupled through a mixing plane. After a comprehensive survey, of the various levels of sophistication that can be built into the quasi-steady model, a straight-forward, spanwise non-matching mixing plane has been implemented. Borrowing ideas from AUSM schemes, the flux is split into convective and pressure terms which are averaged and transferred separately. Working directly and exclusively with fluxes ensures conservation and avoids the decoding of primitive variables from fluxes or flux-like terms, for which there may be no solution (not all combinations of mass, momentum, and energy fluxes are physically realizable locally) and which might be expensive for fluids with complex state equations. Coupling the blade rows on all grid levels results in a transparent connection
which in the vast majority of cases does not require underrelaxation nor induces any perceptible slowdown in the convergence, compared with an equivalent single blade row computation.

A turbomachinery oriented post-processing module was initiated and has been grown into a comprehensive code-internal data preparation system for external visualization systems, including as an important component a computationally efficient tool to collapse 3D solutions obtained on arbitrary multiblock configurations into a meridional average. The design of this software around quantity codes has proven most satisfactory in that it accommodated all later additions to the solver (in its diverse domains of application) without revision or modification, including the request of selected output through a graphical user interface added at a later stage.

Mesh Dependence
To be reliable, numerical flow solutions must be grid independent, in the sense that changes in monitored variables remain below a tolerated level of error when the mesh is refined. For 3D Navier-Stokes calculations on highly-skewed periodic H-meshes of an LH2 pump inducer which guided much of the basic turbomachinery-specific developments, this situation was far from achieved. In a systematic study progressing from a comparison of various skew and quasi-orthogonal topologies on one 2D blade-to-blade surface back to the full 3D configuration using meshes between ~25 000 and ~400 000 cells, low grid shear (enabled by the non-matching periodic boundary condition), strong clustering in turbulent boundary layers even on coarse meshes ($y^+ \approx 1$), and sufficient resolution in the inviscid core flow, in particular in the spanwise direction with at least ~65 blade-to-blade sections, were identified as the decisive factors in ensuring grid converged solutions on meshes from about 200 000 cells onwards.

Euler throughflow model
The spectacular advances in full 3D Euler and Navier-Stokes capability notwithstanding, the throughflow representation remains an essential part of the turbomachinery design process, be it as a design tool, as a common background tying together the individual engine components, or increasingly also as a complete-engine model. However, with the increased levels of Mach number in contemporary designs and the need for better spatial resolution, the limitations of the classical models are more and more keenly felt. Furthermore, the need has arisen in industrial organizations to maintain multiple codes which moreover cannot directly interact, numerical methodology, required boundary conditions, level of spatial resolution, and unsteady capability being all fundamentally different.

The idea therefore suggests itself to endow a 3D Navier-Stokes solver intended for turbomachinery applications with a fully embedded throughflow model based on the solution of the Euler equations, thereby eliminating in one stroke all the aforementioned limitations and inconveniences, and it is the formulation and validation of such a model which constituted one central part of the present work.

Owing in part to a relative paucity of precedence, this task turned out to require a far greater development effort than had been initially estimated. Arriving at a satisfactory tool necessitated an unusual amount of attention to fine details of the implementation, and we mention that, with the notable exception of a method who’s implementation choices are very similar to those made here, not one single Euler throughflow paper presents convergence histories, as one should expect for a new program which in an iterative design cycle will be executed many thousand times and whose computational efficiency therefore is crucial for its acceptance.

It has been shown that the straight-forward, rigid coupling of the blade force to the steady-state tangential momentum equation, a preferred choice in the (sparse) literature presumably conditionend by the respective authors’ background in the classical steady-state throughflow models, modifies the properties of the time-dependent system formed by the remaining four Euler equations, leading to complex eigenvalues and increased wave speeds with the consequence of a more restrictive time step limit and increased numerical dissipation, e.g., by a factor of 3 and 5 in respectively the design and analysis mode for axially sonic flow at a relative flow angle of 60 deg. It is also shown how this undesirable modification of properties can be avoided by a correct elimination of the tangential momentum equation from the system at the time-dependent level. The pure steady-state approach amounts to unwanted (and in all published instances also unnoticed) preconditioning.
An alternative choice emerging naturally in the time-marching Euler framework is to treat the blade force as a time-dependent variable for which an additional equation is solved. This approach offers the conceptual beauty of mimicking the actual physical mechanism in which the turning results from the action of an external force. Both are therefore exactly consistent without any discretization error, which leads to superior behavior in the delicate trans- and supersonic flow regimes, in particular if captured shocks are located near blade edges. The integration of the time-dependent blade force with the basic multigrid-accelerated time marching algorithm was perfected until convergence rates and robustness comparable to non-throughflow 2D Euler calculations were achieved.

A similar level of precision was enabled for the friction force by directly linking it to the imposed loss coefficient, instead of the usual approach of passing through an entropy field.

Another aspect which sets the here developed Euler throughflow model apart from the few that have been reported in the literature is a rigorous insistence on design capability by offering a genuine design mode with prescribed swirl distribution and through a purely parametric representation of the effect of the blade combining spanwise profiles and streamwise distributions for turning, losses, and blade blockage. Although the program does offer the possibility to process existing blade geometries into the parameters required as input by the throughflow model, no advantages were found to be associated with such an approach. On the contrary, it was found more beneficial to focus instead on the actual flow, a strategy which is indeed more coherent with the concept of the axisymmetric model in which by definition the blade and with it details of the geometry disappear and only its effect on the mean flow remains.

Two parameters have been identified as decisive for the accurate throughflow prediction of trans- and supersonic flows, namely the throat width, which determines the choking massflow, and the so-called effective blockage, which, to ensure a smooth, monotonous evolution of the flow without erratic captured shocks, must vary smoothly. Input modes are provided which offer direct control of each.

In the throughflow developments, special emphasis was put on axial transonic compressors because of their increasingly important role in today’s design practice and because their mode of operation critically hinges on shocks. A three-pronged approach was thereby adopted consisting of (i) an analysis of the shock capturing properties of the Euler throughflow model in the different modes, (ii) a survey of the shock structure in transonic compressors at the various possible operating points considering both experimental and computational data, and (iii) a systematic search for ways of representing the actual shock pattern with the captured shocks admitted by the Euler throughflow model.

For one well-documented test case (NASA Rotor 67), 3D Navier-Stokes solutions covering the complete design speed performance curve were analyzed with regard to throughflow modelling parameters. It was established that nearly all distributions are well described by the simple power laws used in the present parametric blade model. Using the blade modelling parameters suggested by this analysis, all calculated operating points between near-stall and deeply choked could be accurately reproduced in both throughflow modes. The effect of the different shock capturing properties in design mode (axisymmetric shock, only if $M_{\text{axial}} > 1$) an in analysis mode (normal shock, if $M_{\text{rel}} > 1$) on agreement with the pitch-averaged solution led to the definition of a new, hybrid mode which combines the imposed exit flow angle of the analysis mode with the imposed streamwise swirl distribution of the design mode. In lieu of general recommendations regarding the suitability of the different throughflow modes for certain applications, which we feel cannot be given because each test case and each user’s priorities are different, we summarize here the two principal points of divergence, namely (i) that the analysis mode allows to control throat width and effective blockage (which the design and hybrid modes do not), but that (ii) captured (quasi-)normal shocks over which one has only limited control may interfere with imposed losses and distort the flowfield inside the blade passage (which cannot occur in the design and hybrid modes as long as the axial Mach number remains subsonic).

The Euler throughflow implementation supports the classical Q3D approach to blade design by providing output of streamsurface data taking account of the need of blade-to-blade programs for a smooth surface of revolution with a continuously varying streamtube thickness. In Euler throughflow solutions with captured shocks and high streamwise mesh density around the blade edges, neither is guaranteed so that suitable distributions are reconstructed by fitting cubic splines through a set of skeleton points obtained from spanwise integration of the massflow.
The versatility of the Euler throughflow model has been further demonstrated with a selection of test cases, comprising compressors and turbines ranging from isolated blade rows to multistage bypass configurations and covering Mach numbers from low subsonic to fully supersonic, which illustrate both its numerical quality and the new possibilities it opens up compared with the classical methods.

Condensable Fluids

Condensable fluids and more generally fluids with non-trivial state equations have been posing a severe problem to CFD codes for a long time. The recent embrace by CFD of all sorts of industrial applications has exacerbated the need for a better description of the fluid thermodynamic properties beyond the perfect gas and incompressible fluid models. With the present work, we have given a definitive answer to this dilemma.

Taking what can be deemed the most radical approach possible, we eliminated the notion of a state equation, in the sense of an equation that explicitly appears in the solver. It has been replaced by the direct interpolation of thermodynamic variables and transport coefficients from dedicated tables. The burden of processing thermodynamic data and inverting state equations is thereby shifted away from the solver, which sees only the final polished, efficient, accurate interpolation tables. Simplified state equations are henceforth eliminated as a possible source of error.

Two key observations leading to this choice are that searching on even a large structured table can be made extremely efficient, and that the memory required for storing such large (and thus accurate) tables has become small compared with that required for typical 3D Navier-Stokes calculations. Optimized algorithms have been developed for both the searching phase and the interpolation phase proper. All these factors combine to produce an efficient, robust, and versatile method.

Depending on the test case and type of interpolation tables, computational overheads between 5% and 30% have been registered. This must be compared with the practical impossibility of directly using the reference state equation as in the case of the Helmholtz equation for steam, or unacceptable overheads, e.g., between 300% and 800% for the MBWR equation for liquid hydrogen (Gerolymos and Geai, 1994).

From an industrial perspective, the table implementation facilitates software engineering, maintenance, and quality control, since one simple implementation (consisting of a single, black-box subroutine call wherever a thermodynamic or related variable is needed) can now handle all state equations, present and future. The tabular properties module is complemented by an external table generation tool.

While the tabular properties module as it stands has already proven its qualities and usefulness, it also constitutes a unique, stable platform from which a variety of application can now be explored. Coming to mind are the construction of a differential Eulerian condensation model by adding two or four transport equations describing the droplet population, the pursual of the modelling of two-phase flow on the liquid side (cavitation), and the exploration of flows in the critical region and of fluids with special properties, for instance so-called heavy gases as suggested by Thompson and Lambrakis (1973), and this list will certainly grow.


Bibliography


Arnone, A. (). *On the Use of Multigrid in Turbomachinery Calculations*.


Boadway, J. D. (1976). Transformation of Elliptic Partial Differential Equations for Solving Two-


Vuillez, C., and Petot, B. (1994). New Methods, New Methodology—Advanced CFD in the SNEC-


**Missing**


Appendix A  The Mathematical Formulation of the System of Euler Throughflow Equations

A.1  2D Non-linear Convection

We first consider the 2D non-linear convection problem, Figure A.1,

\[
\partial_t \begin{bmatrix} u \\ v \end{bmatrix} + \partial_x \begin{bmatrix} u^2 \\ uv \end{bmatrix} + \partial_y \begin{bmatrix} vu \\ v^2 \end{bmatrix} = 0
\]  \hspace{1cm} (A.1)

Grouping the conserved variables and the fluxes,

\[
U = \begin{bmatrix} \mu \\ v \end{bmatrix} \hspace{1cm} F = \begin{bmatrix} u^2 \\ uv \end{bmatrix} \hspace{1cm} G = \begin{bmatrix} vu \\ v^2 \end{bmatrix}
\]  \hspace{1cm} (A.2)

one can define the Jacobi matrices

\[
A = \frac{\partial F}{\partial U} = \begin{bmatrix} 2u & 0 \\ v & u \end{bmatrix} \hspace{1cm} B = \frac{\partial G}{\partial U} = \begin{bmatrix} v & u \\ 0 & 2v \end{bmatrix}
\]  \hspace{1cm} (A.3)

with eigenvalues

A: \[
\lambda_1 = u \\
\lambda_2 = 2u
\]  \hspace{1cm} (A.4)

B: \[
\lambda_1 = v \\
\lambda_2 = 2v
\]

Disturbances propagate in a given direction \( \hat{k} \), \( |\hat{k}| = 1 \), at speeds given by the eigenvalues of the matrix

\[
K = \hat{A} \cdot \hat{k}
\]  \hspace{1cm} (A.5)

These are

K: \[
\lambda_1 = \hat{u} \cdot \hat{k} \\
\lambda_2 = 2\hat{u} \cdot \hat{k}
\]  \hspace{1cm} (A.6)

The Rankine-Hugoniot relations for discontinuities admitted by (A.1) are

\[
\begin{align*}
\text{moving:} & & \left\{ \begin{array}{l} [u^2] = [u]c \\ [uv] = [v]c \end{array} \right. \\
\text{stationary:} & & \left\{ \begin{array}{l} [u^2] = 0 \\ [uv] = 0 \end{array} \right.
\]  \hspace{1cm} (A.7)

Without loss of generality, (A.7) assumes that the discontinuity is aligned with the y-direction; \( c \) denotes the x-component of its velocity. A stationary discontinuity only admits the trivial solutions

\[
u_2 = \pm v_1 \hspace{1cm} (A.8)
\]

Introduction of the throughflow hypotheses will reduce (A.1) to a 1D non-linear convection problem. It will prove useful to compare this with the simplest such problem, namely Burger’s equation without viscous terms, which bears the same relation to (A.1) as the 1D Euler equations to the 2D Euler equations. The conservative form
The Mathematical Formulation of the System of Euler Throughflow Equations

shows the flux, \( u^2 \), while the non-conservative form

shows the convection speed, \( 2u \).

The throughflow equations associated with (A.1) are obtained by assuming the flow to be uniform in the \( y \)-direction, \( \partial_y = 0 \), and by including the effect of \( y \)-gradients in a source term,

where formally \( P = -B\partial_y U \). Explicitly,

By analogy with the physical throughflow model, the vector

represents a force acting perpendicular to the velocity vector,

The above conclusion holds for \( u \neq 0 \). Equation (A.14) determines the direction of \( \hat{\mathbf{p}} \); its magnitude is determined by one of two conditions:

(i) \( v \) is imposed (the so-called design problem).

(ii) \( a = v/u \) is imposed (the so-called analysis problem).

The imposed distribution of either \( v \) or \( a \) is time-independent and smooth. In the design problem, \( v = v(x) \) is not an unknown anymore, while in the analysis problem \( v = v(u(x,t)) = a(x) u(x,t) \) is a known function of \( u \).

In either case, elimination of \( P \) from system (A.12) yields one equation for the remaining unknown \( u \),

(A.15)
For the design problem, its conservative form is
\[ \partial_t u + \partial_x (u^2 + v^2 \ln u) = (2 \ln u - 1) v \partial_x v \]
(A.16)

and the non-conservative form
\[ \partial_t u + 2u \left( 1 + \frac{v^2}{2u^2} \right) \partial_x u = -v \partial_x v \]
(A.17)

For the analysis problem, its conservative form is
\[ \partial_t u + \partial_x u^2 = \frac{-au^2 \partial_x a}{1 + a^2} \]
(A.18)

and the non-conservative form
\[ \partial_t u + 2u \partial_x u = \frac{-au^2 \partial_x a}{1 + a^2} \]
(A.19)

Compared with the simple 1D convection problem, the flux of the design problem includes the additional contribution \( v^2 \ln u \), equations (A.9) and (A.16). The convection speed is multiplied by a factor \( 1 + v^2/2u^2 \), equations (A.10) and (A.17). The analysis problem differs from the simple 1D convection problem only through a source term, compare equations (A.9), (A.10) with (A.18), (A.19). This source term and those arising in the design problem are well-behaved, smooth functions of \( u \). The design problem requires \( u > 0 \).

An alternative formulation of the analysis problem is obtained by dropping the term \( \partial_t v \) in equations (A.12) and (A.15), leading to the conservative form
\[ \partial_t u + \partial_x ((1 + a^2) u^2) = au^2 \partial_x a \]
(A.20)

and the non-conservative form
\[ \partial_t u + 2u(1 + a^2) \partial_x u = -au^2 \partial_x a \]
(A.21)

These equations have the same steady state solution as (A.18) or (A.19), but both the flux and the convective speed are multiplied by \( 1 + a^2 \). The residual \( \partial_t u \) is increased by the same factor.

The steady Rankine-Hugoniot relation for the simple 1D convection problem and the analysis problem has a trivial solution only,
\[ [u^2] = 0 \quad \Rightarrow \quad u_2 = \pm u_1 \]
(A.22)

The same holds for the design problem,
\[ [u^2] = -v^2 [\ln u] \quad \Rightarrow \quad u_2 = u_1 \]
(A.23)

**A.2 The 2D Euler Equations**

We now consider the 2D Euler equations,

\[
\begin{align*}
\partial_t & \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{pmatrix} \\
+ & \partial_x \begin{pmatrix} \rho u^2 + p \\ \rho u^2 + p \\ \rho v^2 + p \\ \rho uH + \rho vH \end{pmatrix} \\
+ & \partial_y \begin{pmatrix} \rho v \\ \rho uv \\ \rho H \end{pmatrix} = 0 \\
\end{align*}
\]

(A.24)

where the total energy \( E \) and total enthalpy \( H \) are defined by
The mathematical formulation of the system of Euler throughflow equations has been analyzed by Hirsch (1990). For later reference, we repeat some results for the 1D Euler equations:

\[
E = \frac{1}{\gamma - 1} \rho + \frac{1}{2} (u^2 + v^2) \quad E = \frac{\gamma}{\gamma - 1} \rho + \frac{1}{2} (u^2 + v^2) \quad (A.25)
\]

The mathematical properties of system (A.24) have been analyzed by Hirsch (1990). For later reference, we repeat some results for the 1D Euler equations:

\[
\begin{bmatrix}
\partial_t \\
\rho \\
\rho u + \partial_x \\
\rho E
\end{bmatrix} = 0 \quad (A.26)
\]

where the total energy \( E \) and total enthalpy \( H \) are defined by

\[
E = \frac{1}{\gamma - 1} \rho + \frac{u^2}{2} \quad E = \frac{\gamma}{\gamma - 1} \rho + \frac{u^2}{2} \quad (A.27)
\]

The Jacobian matrix for the primitive variables and its eigenvalues are

\[
\tilde{A} = \begin{bmatrix}
u & \rho & \cdot \\
\cdot & u & 1/\rho \\
\cdot & \rho c^2 & u
\end{bmatrix} \quad \lambda_1 = u \\
\lambda_2 = u \pm c^2 \quad (A.28)
\]

The throughflow equations associated with (A.24) are interpreted as a description of the flow averaged in the \( y \)-direction. They are formally obtained by setting \( \partial_y = 0 \) and adding an external force on the RHS of the momentum equation. Explicitly,

\[
\begin{align*}
\partial_t (\rho u) + \partial_x (\rho u^2 + p) &= \rho r \quad (a) \\
\partial_t (\rho u) + \partial_x (\rho u^2 + p) &= \rho r \quad (b)
\end{align*} \quad (A.29)
\]

The force

\[
\tilde{f} = \begin{bmatrix}
r \\
s
\end{bmatrix} \quad (A.30)
\]

acts normal to the velocity vector,

\[
\tilde{f} \cdot \hat{u} = 0 \quad \Rightarrow \quad r = \frac{v}{u} s \quad (A.31)
\]

Design and analysis problems have been defined in the preceding section. Elimination of \( \tilde{f} \) from system (A.29) yields the scalar momentum equation

\[
\partial_t (\rho u) + \frac{v}{u} \partial_t (\rho v) + \partial_x (\rho u^2 + p) + \frac{v}{u} \partial_x (\rho uv) = 0 \quad (A.32)
\]

For the design problem, its conservative form is

\[
\partial_t (\rho u) + \partial_x (\rho u^2 + p) = -\rho v \partial_x v \quad (A.33)
\]

An alternative formulation of the design problem, obtained by dropping the term \( \partial_t (\rho v) \) in equations (A.29) and (A.32), cannot be brought in conservative form,

\[
\partial_t (\rho u) + \partial_x (\rho u^2 + p) + \rho v^2 \partial_x \ln (\rho u) = -\rho v \partial_x v \quad (A.34)
\]

For the analysis problem, the conservative form of (A.32) is
The Mathematical Formulation of the System of Euler Throughflow Equations

\[ \partial_t (\rho u) + \partial_x \left( \rho u^2 + \frac{p}{1 + \alpha^2} \right) = \frac{-a}{1 + \alpha^2} \left( \rho u^2 + \frac{2p}{1 + \alpha^2} \right) \partial_x a \]  

(A.35)

which is equivalent with

\[ (1 + \alpha^2) \partial_t (\rho u) + \partial_x ((1 + \alpha^2) \rho u^2 + p) = a \rho u^2 \partial_x a \]  

(A.36)

An alternative formulation of the analysis problem is obtained by dropping the term \( \partial_t (\rho v) \) in equations (A.29) and (A.32), leading to the conservative form

\[ \partial_t (\rho u) + \partial_x ((1 + \alpha^2) \rho u^2 + p) = a \rho u^2 \partial_x a \]  

(A.37)

The systems of Euler throughflow equations for the design and analysis problem have the same mass equation,

\[ \partial_t \rho + \partial_x (\rho u) = 0 \]  

(A.38)

and the same energy equation,

\[ \partial_t (\rho E) + \partial_x (\rho H u) = 0 \]  

(A.39)

but different momentum equations, (A.33) or (A.34) for the design problem and (A.35) or (A.37) for the analysis problem. In system form, the design problem is described by (using equation (A.33))

\[ \begin{bmatrix} \partial_t & \rho \\ \rho u & + \partial_x \end{bmatrix} \begin{bmatrix} \rho u \\ \rho u^2 + p \end{bmatrix} = \begin{bmatrix} 0 \\ -\rho v \partial_x v \end{bmatrix} \]  

(A.40)

or by (using equation (A.34))

\[ \begin{bmatrix} \partial_t & \rho u \\ \rho E & + \partial_x \end{bmatrix} \begin{bmatrix} \rho u \\ \rho u^2 + p + \rho v^2 \partial_x \end{bmatrix} = \begin{bmatrix} 0 \\ ln(\rho u) \end{bmatrix} = \begin{bmatrix} 0 \\ -\rho v \partial_x v \end{bmatrix} \]  

(A.41)

and the analysis problem by (using equation (A.35))

\[ \begin{bmatrix} \partial_t & \rho u \\ \rho E & + \partial_x \end{bmatrix} \begin{bmatrix} \rho u \\ \rho u^2 + \frac{p}{1 + \alpha^2} \end{bmatrix} = \begin{bmatrix} 0 \\ -\frac{a}{1 + \alpha^2} \left( \rho u^2 + \frac{2p}{1 + \alpha^2} \right) \partial_x a \end{bmatrix} \]  

(A.42)

or by (using equation (A.37))

\[ \begin{bmatrix} \partial_t & \rho u \\ \rho E & + \partial_x \end{bmatrix} \begin{bmatrix} \rho u \\ (1 + \alpha^2) \rho u^2 + p \end{bmatrix} = \begin{bmatrix} 0 \\ a \rho u^2 \partial_x a \end{bmatrix} \]  

(A.43)

Compared with 1D Euler, equation (A.26), both the design and analysis problem contain a momentum source term and include the contribution from the transverse velocity component in E and H, equation (A.25) instead of (A.27). For the analysis problem, the momentum flux contains the additional term \(-a^2p/(1 + a^2)\), system (A.42), or \(a^2\rho u^2\), system (A.43). Note the non-conservative form of the design problem (A.41) and the fact that the two systems which include the term \(\partial_t (\rho v)\)
in the momentum equation, (A.40) and (A.42), can be obtained from the respective other system in which the term \( \partial_t (\rho v) \) has been dropped, (A.41) or (A.43), through preconditioning.

The Rankine-Hugoniot relations for stationary discontinuity surfaces admitted by the 2D Euler equations, system (A.24), are

\[
[p\hat{\nu}] \cdot \vec{l}_n = 0
\]
\[
[\hat{\nu}] \rho \hat{\nu} \cdot \vec{l}_n + [p] \vec{l}_n = 0
\]
\[
[H] = 0
\]

(A.44)

with the velocity vector \( \hat{\nu} = (u, v)^T \) and the normal vector of the discontinuity surface \( \vec{l}_n = (n_x, n_y)^T \). For \( \vec{l}_n = (1, 0)^T \), they are identical with the Rankine-Hugoniot relations for stationary discontinuities admitted by the design problem, equations (A.40) or (A.41),

\[
\text{Design Problem} \quad \rho u\vec{l}_n + 0 = 0 \quad \rho u\vec{l}_n + [p] \vec{l}_n = 0 \quad [v] = 0 \quad [H] = 0
\]

(A.45)

The design problem thus admits shocks normal to the x-direction, so-called axisymmetric shocks. Observe that the term \( p v^2 \partial_x \ln(\rho u) \) in equation (A.41) will not appear in the jump relations, because \( \rho u \) is continuous at the shock. For \( \vec{l}_n = (u_1, v_1)^T / |u_1| \), where the index 1 denotes the upstream flow state, the 2D Euler equations admit a normal shock, which leaves the flow direction unchanged. One may therefore set \( \vec{l}_n = (1, a)^T / \sqrt{1 + a^2} \). If this is inserted in (A.44), the resulting Rankine-Hugoniot relations are identical with those of the analysis problem, equations (A.42) or (A.43),

\[
\text{Analysis Problem} \quad \rho u\vec{l}_n + 0 = 0 \quad \rho u\vec{l}_n + [p]/(1 + a^2) = 0 \quad [v] - a[u] = 0 \quad [H] = 0
\]

(A.46)

The analysis problem thus admits normal shocks. Whether or not the term \( \partial_t (\rho v) \) is dropped has no influence on the Rankine-Hugoniot relations.

The Jacobian matrix for primitive variables for the design problem (A.40) is the same as that for the 1D Euler equations,

\[
\tilde{A}^{TF}_{\text{design 1}} = \tilde{A} = \begin{vmatrix} u & \rho & \cdot \\ \cdot & u / \rho & 1/\rho \\ \cdot & \rho c^2 & u \end{vmatrix}
\]
\[
\begin{align*}
\lambda_1 &= u \\
\lambda_{2,3} &= u \pm c
\end{align*}
\]

(A.47)

while that for the analysis problem (A.42) is

\[
\tilde{A}^{TF}_{\text{analysis 1}} = \begin{vmatrix} u & \rho & \cdot \\ \cdot & u / (\rho (1 + a^2)) & 1/\rho (1 + a^2) \\ \cdot & \rho c^2 & u \end{vmatrix}
\]
\[
\begin{align*}
\lambda_1 &= u \\
\lambda_{2,3} &= u \pm c / (\sqrt{1 + a^2})
\end{align*}
\]

(A.48)

The eigenvalues give the speed at which the entropy wave and the upstream and downstream running acoustic waves propagate along the coordinate direction \( x \). For the design problem, the
acoustic waves propagate axially (i.e., in the x-direction) at a speed relative to the flow velocity, see \( \lambda_{2,3} \) in equation (A.47). For the analysis problem, they are constrained to the imposed flow direction along which they propagate at a speed \( \pm c \) relative to the flow velocity, which corresponds to an axial component \( c_x = \pm c/\sqrt{1 + a^2} \), see \( \lambda_{2,3} \) in equation (A.48).

The Jacobian matrix for primitive variables for the design problem (A.41) is

\[
\tilde{A}_{\text{design} \ 2} = \begin{bmatrix}
  u & \rho \\
  v^2/\rho & u + v^2/u & 1/\rho \\
  -(\gamma - 1)uv^2 & \rho(c^2 - (\gamma - 1)v^2) & u
\end{bmatrix} \quad (A.49)
\]

With the aid of suitable auxiliary variables, its eigenvalues can be expressed in compact form, \( i = \sqrt{-1} \),

\[
A_1 = 3u^2(c^2 - (\gamma - 2)v^2) + v^4 \\
A_2 = (3u^2 + v^2)(2(3u^2 + v^2)^2 + 9u^2(c^2 - 3u^2 - \gamma v^2)) - 27u^4(c^2 - u^2) \\
A_3 = v^2(9u^2(c^2 - 3(\gamma - 1)u^2 - (\gamma - 2)v^2) + 2v^4) \\
B_1 = \left( A_2 + \sqrt{A_3^2 - 4A_1^3} \right)^{1/3} \\
C_1 = A_1/(2^{2/3}B_1) \\
C_2 = B_1/2^{4/3}
\]

\[
D_1 = C_1 + C_2 \\
D_2 = C_2 - C_1 \\
\lambda_1 = u + (v^2 + 2D_1)/(3u) \\
\lambda_{2,3} = u + (v^2 - D_1 \pm i\sqrt{3}D_2)/(3u)
\]

The Jacobian matrix for primitive variables for the analysis problem (A.43) is

\[
\tilde{A}_{\text{analysis} \ 2} = \begin{bmatrix}
  u & \rho \\
  (a^2u^2)/\rho & (1 + 2a^2)u & 1/\rho \\
  -(\gamma - 1)(1 + a^2)a^2u^3 & \rho(c^2 - 2(\gamma - 1)(1 + a^2)a^2u^2) & (1 - (\gamma - 1)a^2)u
\end{bmatrix} \quad (A.51)
\]

With the aid of suitable auxiliary variables, its eigenvalues can be expressed in compact form, \( i = \sqrt{-1} \),
\[ A_1 = (1 + (1 - \gamma/3)a^2)u \]
\[ A_2 = c^2 - 3(1 + a^2)u^2 \]
\[ A_3 = (c^2 - (1 + a^2)u^2)u \]
\[ B_1 = 3(3A_1^2 + A_2) \]
\[ B_2 = 27(2A_1^3 + A_1A_2 - A_3) \]
\[ C_1 = \left( B_2 + \frac{B_2^2}{4B_1^3} \right)^{1/3} \]
\[ D_1 = B_1/(3 \cdot 2^{2/3}C_1) \]
\[ D_2 = C_1/(6 \cdot 2^{1/3}) \]
\[ E_1 = D_1 + D_2 \]
\[ E_2 = D_2 + D_1 \]
\[ \lambda_1 = A_1 + 2E_1 \]
\[ \lambda_{2,3} = A_1 - E_1 \pm i\sqrt{3}E_2 \]

Evaluating these expressions for selected values of the two parameters \(u\) and \(v\) (design problem) or \(u\) and \(a\) (analysis problem), keeping \(c\) and \(\gamma\) fixed, shows that (i) the \(\lambda\) are not always real and (ii) the spectral radius is generally larger than for the simpler formulation.
Figure A.2 Alternative formulation of the design problem (equations (A.41) and (A.50), $\gamma = 1.4$, $c = 300$)—maximum imaginary part of the three eigenvalues divided by the corresponding real part.

Figure A.3 Design problem ($\gamma = 1.4$, $c = 300$)—spectral radius of the alternative formulation (equations (A.41) and (A.50)) divided by the spectral radius of the simpler formulation (equations (A.40) and
Figure A.4  Alternative formulation of the analysis problem (equations (A.43) and (A.52), $\gamma = 1.4$, $c = 300$)—maximum imaginary part of the three eigenvalues divided by the corresponding real part.

Figure A.5  Analysis problem ($\gamma = 1.4$, $c = 300$)—spectral radius of the alternative formulation (equations (A.43) and (A.52)) divided by the spectral radius of the simpler formulation (equations (A.42) and
Figures A.2 to A.5 compare the properties of the alternative formulations (equations (A.41) and (A.50) for the design problem and equations (A.43) and (A.52) for the analysis problem) to the simpler formulations (equations (A.40) and (A.47) for the design problem and equations (A.42) and (A.48) for the analysis problem). Two quantities relevant to the properties of the system of Euler throughflow equations and its numerical solution are studied. The first quantity is the ratio between the maximum imaginary part of all three eigenvalues and the corresponding real part, both in absolute value,

\[ \max_{i=1,2,3} \left| \frac{\text{Im}(\lambda_i)}{\text{Re}(\lambda_i)} \right| \]  \hspace{1cm} (A.53)

The second quantity is the ratio between the spectral radius of the alternative formulation and that of the simpler formulation,

\[ \frac{\sigma_{\text{alternative}}}{\sigma_{\text{simpler}}} \quad \text{with} \quad \sigma = \max_{i=1,2,3} |\text{Re}(\lambda_i)| \]  \hspace{1cm} (A.54)

The contour plots show the range

\[ 0 < u < 1000 \quad 0 < a < 2 \]  \hspace{1cm} (A.55)

for fixed values \( \gamma = 1.4 \) and \( c = 300 \). The range of \( a \) corresponds to flow angles between 0 and 63.4 deg. Note that although the independent parameters of the design problem are \( u \) and \( v \), it is more instructive to examine it in function of \( u \) and \( a \); \( a \) is converted to \( v \) through

\[ v = au \]  \hspace{1cm} (A.56)

The vertical line in the figures indicates axially sonic flow.

Figures A.2 and A.4 show that for the alternative formulations of both the design and analysis problems two eigenvalues become complex in a region characterized by moderate flow angle and high supersonic Mach number. Systems (A.41) and (A.43) are then not purely hyperbolic anymore but hyperbolic-elliptic, see for instance Hirsch (1989), section 3.4.

Figures A.3 and A.5 show that the ratio between the spectral radius of the alternative formulation and that of the simpler formulation increases rapidly with increasing flow angle. This implies a more stringent time step limitation for the alternative formulation, if the system of equations is solved by explicit time integration.
Appendix B  Equations Used for Shock Comparisons

For an oblique shock with upstream Mach number $M_1$ and shock angle $\eta = 90 \text{ deg} - \theta$, Figure 2b, in a perfect gas with ratio of specific heats $\gamma$ the pressure and density ratios are given by

\begin{align*}
\frac{p_2}{p_1} &= \xi = \frac{2 \gamma M_{1n}^2 - (\gamma - 1)}{\gamma + 1} \quad \text{(B.1)} \\
\frac{\rho_2}{\rho_1} &= \mu = \frac{(\gamma + 1) M_{1n}^2}{(\gamma - 1) M_{1n}^2 + 2} \quad \text{(B.2)}
\end{align*}

The indices 1 and 2 denote respectively the upstream and downstream flow states. $M_1$ is the relative Mach number and $M_{1n}$ the normal Mach number,

\begin{equation}
M_{1n} = M_1 \cos \eta \quad \text{(B.3)}
\end{equation}

The downstream Mach number can be calculated as

\begin{equation}
M_2^2 = \frac{\mu}{\xi} \left( M_1^2 - \left( \frac{2}{\gamma + 1} \right)^2 (M_{1n}^2 - 1)(\gamma M_{1n}^2 + 1) \right) \quad \text{(B.4)}
\end{equation}

The flow deflection angle $\delta$, Figures 2b and B.1, is

\begin{equation}
\delta = \arctan \frac{2 \tan \eta (M_{1n}^2 - 1)}{2 + M_{1n}^2 (\gamma + 1 - 2 \cos^2 \eta)} \quad \text{(B.5)}
\end{equation}

The relative total pressure ratio is obtained from the isentropic relations,

\begin{equation}
\frac{p_{t2}}{p_{t1}} = \Pi_t = \xi \left( \frac{1}{\tau} \right)^{\frac{\gamma}{\gamma - 1}} \quad \text{(B.6)}
\end{equation}

where

\begin{equation}
\frac{T_2}{T_1} = \tau = \frac{\xi}{\mu} = \frac{2 + (\gamma - 1) M_1^2}{2 + (\gamma - 1) M_2^2} \quad \text{(B.7)}
\end{equation}

is the temperature ratio. Variables without the subscript ‘abs’ are understood to be relative. The loss coefficient is calculated as

\begin{equation}
\psi = \frac{p_{t1} - p_{t2}}{p_{t1} - p_1} = \frac{1 - \Pi_t}{1 - p_1 / p_{t1}} \quad \text{(B.8)}
\end{equation}

where

\begin{equation}
\frac{p_1}{p_{t1}} = \left( 1 + \frac{\gamma - 1}{2} M_1^2 \right)^{-\frac{\gamma}{\gamma - 1}} \quad \text{(B.9)}
\end{equation}

The absolute total pressure ratio is obtained from the isentropic relations with absolute Mach numbers,

\begin{equation}
\frac{p_{t2 \text{ abs}}}{p_{t1 \text{ abs}}} = \Pi_{t \text{ abs}} = \xi A^{\gamma - 1} \quad \text{(B.10)}
\end{equation}

where
The auxiliary quantity $A$ is also used to calculate isentropic efficiency,

$$
\eta_{is} = \frac{\frac{\gamma - 1}{\gamma} \frac{\gamma}{\gamma - 1} - 1}{\frac{\xi}{\gamma - 1} \frac{A - 1}{\tau A - 1}}
$$

If $M_2 > 1$, the oblique shock (A in figure 2c) may be followed by a normal shock (B). If this has been requested, $\xi, \mu$ and $M_2$ calculated according to Equations (B.1), (B.2), and (B.4) are correct-
Equations Used for Shock Comparisons

ed to reflect the additional normal shock. In algorithmic notation, respecting the order of the operations,

\[
\xi \leftarrow \frac{2\gamma M_2^2 - (\gamma - 1)}{\gamma + 1} \tag{B.16}
\]

\[
\mu \leftarrow \frac{(\gamma + 1) M_2^2}{(\gamma - 1) M_2^2 + 2} \tag{B.17}
\]

\[
M_2^2 \leftarrow \frac{(\gamma - 1) M_2^2 + 2}{2\gamma M_2^2 - (\gamma - 1)} \tag{B.18}
\]

All subsequent equations remain unchanged.

For the oblique shock with realignment to the original flow direction, the loss, expressed in terms of the relative total pressure ratio, is first calculated exactly as above, Equations (B.6) and (B.8). The oblique shock with realignment is then defined through the conservation of mass and energy, combined with the loss incurred by the oblique shock, which replaces conservation of momentum:

\[
\text{mass: } \rho_1 M_1 \sqrt{T_1} = \rho_2 M_2 \sqrt{T_2} \tag{B.19}
\]

\[
\text{energy: } T_1 \left(1 + \frac{\gamma - 1}{2} M_1^2\right) = T_2 \left(1 + \frac{\gamma - 1}{2} M_2^2\right) \tag{B.20}
\]

\[
\text{loss / momentum: } \frac{p_2}{p_1} \left(\frac{2 + (\gamma - 1) M_2^2}{2 + (\gamma - 1) M_1^2}\right)^{\frac{\gamma}{\gamma - 1}} = \Pi_{\text{oblique shock}} \tag{B.21}
\]

The implicit equation for \( M_2 \) derived from this system,

\[
f(M_2) = \left(\frac{2 + (\gamma - 1) M_2^2}{2 + (\gamma - 1) M_1^2}\right)^{\frac{\gamma + 1}{\gamma - 1}} \left(\Pi_{\text{oblique shock}} \frac{M_2^2}{M_1^2}\right)^2 = 0 \tag{B.22}
\]

is solved by a bisection algorithm. (The Newton method fails for the subsonic root because it nearly coincides with the only inflection point of the curve \( f(M_2) \).) For the considered Mach numbers \( M_1 \) and shock angles \( \eta \), there are always two distinct real solutions. Once \( M_2 \) has been determined, the density ratio, \( \mu \), pressure ratio, \( \xi \), and temperature ratio, \( \tau \), are easily found from Equations (B.19) and (B.20). The absolute total pressure ratio and efficiency are obtained as for an oblique shock, Equations (B.10)–(B.15), except that the flow deflection angle \( \delta = 0 \), so that \( b_2 = b_1 \). The case of a second, normal shock in 2D if the downstream Mach number of the first, oblique shock is \( > 1 \) (after realignment) is handled in the same way as for an oblique shock without realignment: \( \xi, \mu, \) and \( M_2 \) are corrected according to Equations (B.16)–(B.18) to reflect the additional normal shock.
Appendix C  Technical Details of Table Interpolation

The thermodynamic interpolation tables use bicubic interpolation. Denoting the dependent variable by \( z \) and the independent variables by \( x \) and \( y \),

\[
z(x, y) = c_{00} + c_{01}y + c_{02}y^2 + c_{03}y^3 + c_{10}x + c_{11}xy + c_{12}xy^2 + c_{13}xy^3 + c_{20}x^2 + c_{21}x^2y + c_{22}x^2y^2 + c_{23}x^2y^3 + c_{30}x^3 + c_{31}x^3y + c_{32}x^3y^2 + c_{33}x^3y^3
\]  

Suitably programmed, the evaluation of (C.1) involves 15 multiplications and 15 additions, for a total of 30 operations. The bicubic spline surface is \( C^2 \) at patch boundaries. (The second partial derivatives are continuous.) Partial derivatives are calculated from the same set of coefficients, for example in \( x \),

\[
\left( \frac{\partial z}{\partial x} \right) = c_{10} + c_{11}y + c_{12}y^2 + c_{13}y^3 + 2(c_{20}x + c_{21}xy + c_{22}x^2y^2 + c_{23}x^2y^3) + 3(c_{30}x^2 + c_{31}x^2y + c_{32}x^2y^2 + c_{33}x^2y^3)
\]  

The cost of evaluating one first partial derivative is 13 multiplications and 11 additions, for a total of 24 operations.

The single and two-phase surfaces are slope discontinuous at the saturation line. Separate spline surfaces are therefore created in the two regions. An implicit intersection technique ensures a seamless connection between the two surfaces, as illustrated in Fig. C.1. The figure shows on the right a top view of the \( x,y \)-plane, subdivided into table patches, and on the left a cut through this plane at \( x = \text{cst} \). Because the two surfaces must overlap, the tables contain two sets of coefficients, each covering the entire table.

To find the region for a given \((x, y)\) pair, one could compare one of the two independent variables, for example \( y \), against the interpolated location of the saturation line \( y_{\text{sat}}(x) \), obtained for example from a cubic spline representation. In the general case, this explicit representation of the saturation line, point B in Fig. C.1a, does not, however, coincide with the projection onto the \( x,y \)-plane of the intersection curve between the two bicubic spline surfaces \( z_1(x, y) \) and \( z_2(x, y) \), point A. The resulting compound surface, represented by the bold lines in Fig. C.1a, would have a discontinuity because the switch between \( z_1 \) and \( z_2 \) occurs at point B instead of point A. Implicit intersection avoids this.

For implicit intersection, the type of each patch is stored when the table is generated. Three patch types are distinguished: type 1 patches which lie entirely in the single phase region, type 2 patches which lie entirely in the two-phase region, and mixed patches which cut by the saturation line. For type 1 and type 2 patches, \( z(x, y) \) is calculated directly as \( z_1 \) or as \( z_2 \). For mixed patches (type 3), both \( z_1 \) and \( z_2 \) are calculated. Depending on the type of intersection, either the minimum or the maximum is then taken. The intersection type is encoded in the patch type \((1 = z_1, 2 = z_2, 3 = \max(z_1, z_2), -3 = \min(z_1, z_2))\).

Errors may still occur if the saturation line passes close to patch edges or corners. To rule these out, the original mixed patch region is enlarged by a halo, obtained by declaring the immediate neighbors of original mixed patches also mixed patches, as shown in Fig. C.1b. No distinction is made between original and halo mixed patches.

Three kinds of problems may arise with the implicit intersection technique. First, the two surfaces may be slope continuous at the saturation line. This is the case for the ph,s-surface, represented by the p(h, s) and s(h, p) tables. The intersection technique fails, but a single spline surface can cover both regions. A slightly higher error is to be expected near the saturation line, because each patch of the spline surface is \( C^\infty \) while the p,h,s-surface is only \( C^1 \) across the saturation line.
Second, the intersection may not be clean, in the sense that the surfaces are nearly tangent to each other, resulting in multiple intersections of the spline approximations. This situation occurs for the \( \rho, h, s \)-surface, affecting the \( \rho(h, s) \) table. The problem was solved by making a table for \( p/\rho \) instead, which has a clean intersection.

Third, the intersection type may change from max to min along the saturation line. This situation occurs for the thermal conductivity. A special technique has been developed to handle this case, introducing a fourth patch type, called transition patch. For such patches, the particulars of the intersection between the two surfaces are encoded in the patch type and piecewise bilinear interpolation is performed.

![Diagram showing implicit surface intersection at the saturation line](image)

**Figure C.1** Implicit surface intersection at the saturation line: (a) cut through the \( z(x, y) \) property surface at \( x = \text{cst} \), (b) top view of the \( x,y \)-plane