SPATIAL DIRECT NUMERICAL SIMULATION
OF COMPRESSIBLE
BOUNDARY LAYER FLOW

PROEFSCHRIFT

ter verkrijging van
de graad van doctor aan de Universiteit Twente,
op gezag van de rector magnificus,
prof. dr. F.A. van Vught,
volgens besluit van het College voor Promoties
in het openbaar te verdedigen
op donderdag 18 december 1997 te 13.15 uur

door

Bono Wasistho

geboren op 5 februari 1966
te Semarang - Indonesia
Dit proefschrift is goedgekeurd door de promotor en de assistent-promotor:

Prof. dr. ir. P.J. Zandbergen
Dr. J.G.M. Kuerten
to my mother
and the memory of my father
Wasistho, Bono

Spatial direct numerical simulation of compressible boundary layer flow
Thesis Universiteit Twente, Enschede.
- With ref. - With summary in English, Dutch and Indonesian.
ISBN 9036510619

The work described in this paper is part of the research program of the J.M. Burgers Centre.
## Contents

1 Introduction 1
   1.1 Practical and model problems 1
   1.2 Approach followed 4
   1.3 Overview of relevant literature 6
   1.4 Purpose, research questions and outline 8

2 Governing equations, initial and boundary conditions 11
   2.1 The Navier-Stokes equations 11
   2.2 Solution to the compressible boundary layer equations 12
   2.3 Artificial boundary conditions 17
      2.3.1 Extrapolation method 18
      2.3.2 Characteristic method 19
      2.3.3 Numerical comparison 27

3 Numerical methods 31
   3.1 Spatial discretization and temporal integration 31
   3.2 Buffer domain technique 35

4 Perturbed laminar flow under zero pressure gradient 39
   4.1 Small perturbation stability theories 40
      4.1.1 Linear stability theory 40
      4.1.2 Linear parabolized stability equations 44
   4.2 Parallel base flow 46
      4.2.1 Supersonic flow with small perturbations 46
      4.2.2 Subsonic flow with small perturbations 50
   4.3 Non-parallel base flow 54
      4.3.1 Subsonic flow with small perturbations 54
      4.3.2 Subsonic flow with large perturbations 59
      4.3.3 Conclusion 61

5 Laminar separation and transition 63
   5.1 Pressure prescription boundary condition 65
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.2</td>
<td>Blowing and suction boundary condition</td>
<td>69</td>
</tr>
<tr>
<td>5.3</td>
<td>Effect of pressure gradient and upstream disturbances</td>
<td>72</td>
</tr>
<tr>
<td>5.4</td>
<td>Transition induced by separation bubble; 3D simulation</td>
<td>76</td>
</tr>
<tr>
<td>5.4.1</td>
<td>Configuration</td>
<td>77</td>
</tr>
<tr>
<td>5.4.2</td>
<td>Flow visualization</td>
<td>80</td>
</tr>
<tr>
<td>5.4.3</td>
<td>Mean quantities</td>
<td>83</td>
</tr>
<tr>
<td>5.4.4</td>
<td>Fluctuations</td>
<td>88</td>
</tr>
<tr>
<td>5.5</td>
<td>Conclusion</td>
<td>92</td>
</tr>
<tr>
<td>6</td>
<td>Shock boundary-layer interaction under blowing and suction</td>
<td>95</td>
</tr>
<tr>
<td>6.1</td>
<td>Numerical method and boundary conditions</td>
<td>96</td>
</tr>
<tr>
<td>6.2</td>
<td>Validation: steady SBLI</td>
<td>98</td>
</tr>
<tr>
<td>6.2.1</td>
<td>Grid arrangement and location of boundaries</td>
<td>99</td>
</tr>
<tr>
<td>6.2.2</td>
<td>Spatial discretization, upper boundary conditions and comparison with reference results</td>
<td>101</td>
</tr>
<tr>
<td>6.3</td>
<td>Unsteady SBLI under suction and blowing</td>
<td>105</td>
</tr>
<tr>
<td>6.3.1</td>
<td>Reference configuration</td>
<td>106</td>
</tr>
<tr>
<td>6.3.2</td>
<td>Influence of physical parameters</td>
<td>108</td>
</tr>
<tr>
<td>6.3.3</td>
<td>Onset criterion for vortex shedding</td>
<td>115</td>
</tr>
<tr>
<td>6.4</td>
<td>Conclusion</td>
<td>120</td>
</tr>
<tr>
<td>7</td>
<td>Conclusions and recommendations</td>
<td>121</td>
</tr>
<tr>
<td>7.1</td>
<td>Numerical aspects</td>
<td>121</td>
</tr>
<tr>
<td>7.2</td>
<td>Physical aspects</td>
<td>124</td>
</tr>
<tr>
<td>A</td>
<td>Fourth order spatial discretization</td>
<td>127</td>
</tr>
<tr>
<td></td>
<td>References</td>
<td>130</td>
</tr>
</tbody>
</table>
1.1 Practical and model problems

Flows around a moving vehicle typically exhibit a thin layer along the solid body in which the relative flow velocity with respect to the body drops rapidly close to the solid walls. The flow behavior in this so called boundary layer is dominated by viscous effects, in contrast with the inviscidly dominated flow outside it. Depending on the body surface and the inviscid flow condition, which in turn depends on the body shape, the boundary layer flow can be proceeding downstream in a regular way, forming a laminar flow, or behave in a chaotic way, forming a turbulent flow. The transition from the laminar to the turbulent state is an interesting phenomenon in itself. The boundary layer can also separate from the body surface, forming a closed (the flow reattaches) or an open separation region, depending on the inviscid flow condition. We are especially interested in the boundary layer flow and the influence of the enveloping inviscid flow on it when the vehicle moves with high speed. In the case that the (local) speed is higher than the speed of sound, shock waves may occur causing a sudden pressure rise near the body surface. The occurrence of a shock affects the boundary layer below and vice versa in that the shock may cause a flow separation and the resulting boundary layer thickening influences the structure of the shock feet. Due to this mutual influences, the shock boundary layer interaction complicates the flow considerably, especially if the interaction is unsteady as in the case of shock buffeting. All the phenomena concerning the boundary layer flow, such as transition, separation, and shock boundary layer interaction, play a very important role in the aerodynamic performance of the body, the efficiency of propulsion systems and the aerodynamic load on the body structure. This is the reason for the long time investigations on boundary layer phenomena, as primarily required for the development of aircraft design technology.

As an illustration, Fig. 1-1 - 1-3 sketch phenomena that typically may occur on an aircraft wing in subsonic and transonic flow regimes. During the cruise flight (Fig. 1-1) the airfoil angle of attack is low and the laminar boundary layer becomes turbulent typically through a natural transition, due to small perturbations in the laminar flow. These perturbations grow as they proceed downstream and form the
origin of laminar flow instabilities. The interaction between the growing instabilities triggers the breakdown of the laminar flow and results in turbulence. Although generally, early transitions are avoided, in some cases, a turbulent boundary layer is preferred as it better prevents the flow from separation than a laminar boundary layer. On the other hand, if the angle of attack is high (Fig. 1-2), which is the case during the take off and landing of an airplane, a laminar separation region may occur not far downstream of the leading edge, and downstream of the separation point the flow reattaches in combination with a shedding of vortices or as a turbulent flow. The flow can separate over the rear of the airfoil if the angle of attack is extremely high, corresponding to a stall situation, which causes a considerable loss of lift. This situation is responsible for a number of aircraft accidents during a high angle of attack flight at low altitude. In a transonic flight, a local supersonic region forms along the upper surface of the airfoil and is closed by a shock wave to return to free-stream conditions (Fig. 1-3). The boundary layer encountering the shock wave is in general turbulent, but it can be laminar in the case of a laminar controlled airfoil. The occurrence of a shock wave on the wing surface causes an unpleasant fly experience, as voiced by Chuck Yeager (1985), the first to break the sound barrier: ”On October 5 (1947), I made my sixth powered flight and experienced shock wave buffeting for the first time when I reached 0.86 Mach. It felt like I was driving on bad shock absorbers over uneven paving stones. The right wing suddenly got heavy and began to drop, and when I tried to correct it my controls were sluggish.”

Figure 1-1: Boundary layer flow around wing airfoil with zero or small angle of attack.

Figure 1-2: Boundary layer flow around wing airfoil with high angle of attack.

Shock buffeting can also occur in propulsion systems such as a supersonic combustion ramjet (Fig. 1-4). Fundamentally, the inlet leads to a large drag due to the formation of steady plane shocks and the exhaust nozzle leads to a large thrust. The difference between these two defines the net thrust, which should be sufficiently high to overcome the total drag of the vehicle. In the combustor gaseous fuel is injected and reacts with the turbulent air. When entering the combustor, the incoming flow is still supersonic and this in combination with the blowing due to the fuel injection and the shape of the nozzle can generate shock buffeting and in turn flow separation. The buffeting and flow separation result in a momentum loss which implies a reduction in the net thrust and should therefore be avoided.
Shock boundary layer interactions encountered on transonic wing airfoils.

Figure 1-3: Shock boundary layer interactions encountered on transonic wing airfoils.

The above examples illustrate the important role of boundary layer flows in the performance of high speed vehicles. Therefore the understanding of phenomena which can appear in the compressible boundary layer flows is of importance, not only to have a clear insight in the physics of the flows but also to be able to influence them in a desired direction. Pertinent phenomena of the boundary layer flow studied in the present work are the development of instabilities in the laminar boundary layer, transition to turbulent flow, laminar separation bubbles and shock laminar boundary-layer interactions. It should be noted that the term high speed flows here is not always associated with high Mach number (ratio between the flow velocity and the speed of sound) flows. In some cases low Mach numbers are considered for the sake of comparison with results in incompressible flows.

As a model problem, we consider undisturbed or disturbed laminar boundary layer flows over a flat plate subjected to an arbitrary pressure distribution or blowing and suction prescribed at the upper boundary (Fig. 1-5). In the case of a perturbed flow under zero pressure gradient, the instability of the laminar flow due
to perturbations is studied. In the case of flows subjected to a pressure gradient or suction and blowing, laminar breakdown, separation bubbles and shock boundary layer interaction can be reproduced. In this way, all the interesting phenomena can be captured, while a simple framework is maintained. This model is advantageous in terms of removing many complications, such as grid non-orthogonality, which can reduce the accuracy and increase the amount of work in the approach followed to study the flow. The disadvantage is that the influence of surface curvature is not taken into account. However, in cases where the radius of curvature is large compared to the boundary layer thickness, such as on the surface of an airfoil, the role of surface curvature is small compared to the role of the streamwise pressure gradient.

![Diagram](image)

**Figure 1-5:** Boundary layer flow over a flat plate as model problem.

We should alert that the emphasis of the present work is not fully put on the physical aspects of the flow. Rather, a balance is made between the physical aspects and the adequacy of the approach followed to study these physical aspects. The latter is discussed in the following section.

### 1.2 Approach followed

Several approaches can be followed to study boundary layer phenomena described above: experimental, numerical and, up to some extent, theoretical approaches. We follow the numerical path, which has the advantage of providing detailed results over the experimental approach. The experimental and theoretical results serve as a reference. Although the flow phenomena to be studied vary in physical character, they obey the same physical laws: the laws of conservation of mass, momentum and energy. These laws form together the Navier-Stokes equations, representing an initial boundary value problem. We solve these equations directly in a numerical way. We use results from boundary layer theory and inviscid flow theory to construct initial
and boundary conditions. Direct numerical simulation (DNS) of the Navier-Stokes equations is selected above other advanced numerical methods, such as viscous-inviscid interaction method for the sake of accuracy and is made possible by the growth in computer performance.

Most direct numerical simulations are performed using periodic boundary conditions in the streamwise direction, and the computational frame is thought to move with a 'characteristic' velocity, the phase velocity, along the plate [40]. In this case, the states at the inflow and outflow boundary are exactly the same. The advantage of this so called 'temporal setting' is that only a relatively small computational domain is needed in the streamwise direction and very accurate numerical methods such as spectral or pseudo-spectral spatial discretization (e.g. Murdock (1977), Gottlieb et al. (1983), Voigt et al. (1984), Canuto et al. (1988) and Boyd (1989)) can be used in the streamwise direction. The periodicity assumption, however, limits the applicability of these simulations to parallel flows which renders a direct comparison with physical experiment difficult. A direct application of the standard parallel flow formulation to supersonic boundary layers is found to be unsatisfactory. Considerable discrepancies were found in the growth rates of the instability waves compared to other approaches which do not use the parallel flow assumption (Guo et al. (1994)).

In a more general setting there is no periodicity in the streamwise direction and artificial inflow and outflow boundaries are needed. In this configuration, the computational frame is fixed in space and the flow is entering the domain through the inflow boundary and leaving it through the outflow boundary, i.e. the simulations are performed in a so called 'spatial setting'. The major drawback of this type of simulations is that a much larger streamwise extent of the computational domain needs to be incorporated and hence the computational effort is considerably increased. The requirement of the large extent of the computational domain is caused by the difficulty of providing consistent and accurate inflow and outflow boundary conditions. The boundary conditions should be consistent in order to ensure well-posedness of the Navier-Stokes equations. Moreover, a proper inflow boundary condition should let the flow enter the computational domain such that a minimum downstream region is needed for the flow to adjust itself to the solution within the domain. On the other hand, an outflow boundary condition should prevent as much as possible spurious numerical reflections which could disturb the solution within the domain. Stable outflow boundary conditions for compressible viscous flow are difficult to specify, especially when large disturbances are considered. In this case, the use of an additional buffer domain in the vicinity of the outflow boundary is indispensable, in particular for subsonic flow, in order to efficiently damp wave reflections from the outflow boundary.

In order to study the flow phenomena as described above, which are in general strongly non-parallel, we adopt the spatial setting and employ high order finite difference methods for the spatial discretization. It is noted that a direct simulation of non-parallel flows using periodic streamwise boundary conditions is possible in an approximate way, as demonstrated by Spalart et al. (1991) by adopting a fringe zone in the vicinity of the outflow boundary in which the flow is brought back to the
inflow condition. However, this approach does not enable a substantial decrease in the extent of the computational domain as it should capture the spatial development of the flow. The advantage lies in the possibility to implement spectral discretization in the streamwise direction. In the present study, the finite difference approach is preferred in view of the extension to applications in general geometries where this approach is much more flexible than the spectral approach. Moreover, finite difference methods have better shock capturing capabilities than spectral methods.

1.3 Overview of relevant literature

Some aspects of the problem area we focus upon in this thesis are already known from studies by other researchers. The foundation of the linear stability of laminar, parallel flows is formed by the work of Mack (1984). In contrast to the incompressible flat plate boundary layer, where no inflection points exist, the compressible flat plate boundary layer has generalized inflection points and hence is inviscidly unstable. The generalized inflection point moves away from the wall with increasing Mach number causing the inviscid instability to increase and the viscous instability to weaken. Whereas there is at most a single unstable wave number (frequency) for a certain frequency (wave number) and Reynolds number, Mack found multiple unstable modes in the flow region where the velocity is supersonic relative to the phase velocity of the disturbance. The first unstable mode is similar to the one in the incompressible flow and the other modes, which have no incompressible counterparts, are called Mack modes. The effect of viscosity on these modes is stabilizing and hence the maximum growth rate occurs at infinite Reynolds number. Further, the most unstable first mode in supersonic flow is three-dimensional in contrast to the incompressible flow where the most unstable mode is two-dimensional. However, the most unstable Mack modes are two-dimensional. The effect of a favorable or adverse pressure gradient is such that it weakens or enhances inflections, and hence stabilizes or destabilizes the flow, respectively. Bertolotti (1990) in cooperation with Herbert developed the parabolic stability equations which govern the stability of laminar, non-parallel flows. Their results show that non-parallelism in the flat plate boundary layer flow leads to a higher growth rate of the instability waves. Van Ingen (1956) and Smith (1956) separately showed that there is a critical amplification factor of instability waves in laminar flows at which transition occurs. This semi-empirical transition prediction method is commonly called the $e^n$ method.

A sufficiently strong adverse pressure gradient can cause a laminar flat plate boundary layer to separate. The structure of a steady two-dimensional laminar separation bubble in incompressible flow was described by Horton (1968). The flow is typically characterized by a laminar separation and turbulent reattachment. The kernel of the recirculation region lies near the reattachment point, where a strong wall adverse pressure gradient was observed. Thwaites (1949), Curle & Skan (1957) and Stratford (1954) proposed approximate methods to predict laminar separation in an incompressible boundary layer, while Dobbinga, Van Ingen & Kooi (1972)
and Oswatitsch (1957) proposed relations predicting the angle of separation. Many researchers, a.o. Bestek et al. (1989) and Pauley et al. (1990), observed in their numerical calculations of incompressible flow that a steady separation structure was not possible if a strong external adverse pressure gradient is applied. Pauley et al. reported that a strong adverse pressure gradient, which is generated by means of a suction port at the wall opposite to the test wall, created periodic vortex shedding from the separation and proposed a criterion for the onset of this vortex shedding. Spalart & Coleman (1997) in [73] performed a direct simulation of rapid separation and reattachment of the turbulent boundary layer on an isothermal flat wall. The characteristic of the wall heat transfer was studied. A transitional separation bubble simulation (the separated flow is laminar) was reported by Spalart & Strelets (1997) in [74]. In this work, a comparison between DNS and a Reynolds averaged (RANS) approach with a one-equation turbulence model was carried out.

In supersonic flows, strong adverse pressure gradients and even shocks do not always cause unsteady separation. An experimental study of shock laminar boundary-layer interaction at high Reynolds numbers by Hakkinen et al. (1959) showed that the separation region due to the shock impingement is steady. This observation was confirmed by Katzer (1989) based on Navier-Stokes calculations. However, unsteady separation is reported by Loth & Matthys (1995) in their numerical simulations of shock boundary layer interaction similar to that by Hakkinen, but at low Reynolds numbers. This indicates the existence of viscous instabilities in a low Reynolds number shock boundary layer interaction.

Regarding the numerical aspects, direct numerical simulations are playing an increasingly important role in the investigation of transition thanks to the development of highly accurate discretization methods and the advance in high performance computing, in particular during the last decade. However, only recently direct simulations in the spatial setting are conducted, mostly for incompressible flows (Fasel et al. (1990), Konzelman et al. (1989), Rist (1991)). The main stumbling block for the spatial simulations is, besides their need for enormous computer power, also the difficulties encountered in the specification of appropriate boundary conditions at the artificial boundaries. In particular the reflection of upstream disturbances at the boundaries creates a serious problem. Much effort has been given by many researchers in solving the problem of boundary conditions. The results are of a great importance not only for transition simulations but also for other complex flow simulations with non-periodic boundary conditions. In contrast to the inviscid fluid dynamics (Euler) equations for which exact boundary conditions ensuring well-posedness can be derived (Kreiss (1970), Engquist & Majda (1977)), the problem is more sophisticated for the Navier-Stokes equations. The number of boundary conditions needed for the Navier-Stokes equations is obtained by theoretical analysis of well-posedness (Strikwerda (1977), Gustafsson & Sundström (1978), Dutt (1988)). However, determining whether a given set of boundary conditions for the Navier-Stokes equations will lead to a well-posed problem, can only be assessed in specific simple cases (Gustafsson & Sundström (1978), Dutt (1988)). The reflection of acoustic waves crossing the boundaries is for instance not addressed. Rudy and
Strikwerda (1981) proposed boundary conditions for subsonic Navier-Stokes calculations based on extrapolation of variables and provided a one-dimensional non-reflecting condition. Thompson (1987) presented multi-dimensional non-reflecting boundary conditions based on (inviscid) characteristic theory. Poinsot & Lele (1992) proposed an extension of this method to the Navier-Stokes equations, but the implementation was restricted to simple test cases where large disturbances do not occur. To prevent reflections of large upstream disturbances, Streett & Macaraeg (1989) developed a buffer domain technique for incompressible flows, based on suppression of velocity fluctuations. Liu & Liu (1994) applied a buffer domain based on increasing viscosity and parabolization of the equations for their incompressible transition simulation using an implicit time marching method.

In generating an external adverse pressure gradient, a steady suction (Pauley et al. (1990), Alam & Sandham (1996)) or a decreasing streamwise velocity (Bestek et al. (1994), Rist & Maucher (1994)) is prescribed at the upper boundary. The other variables are obtained by requiring a zero stress (Pauley et al.), assuming that the upper boundary acts as an outflow boundary.

1.4 Purpose, research questions and outline

The research laid down in this thesis has two objectives. The first objective is the development of a set of artificial boundary conditions and numerical methods suitable for compressible DNS in the spatial setting and the investigation of their performance in spatially developing flat plate boundary layers with increasing complexity. The second objective is the study of the physical phenomena in the flows. For this objective we start with a reproduction of known results from literature to get confidence in the approach followed and shed additional light on the findings. In achieving this goal, we use and combine suitable results from the literature and perform further investigations to obtain the information which is still lacking. To make the investigations goal-directed, general research questions are formulated corresponding to the two main objectives.

Relating to the numerical methods and artificial boundary conditions, the following research questions are to be answered.

1. How to construct a high order spatial discretization scheme which is suitable for the present flow applications yet easily adjustable for an extension to a general geometry and how does it perform?

2. How to specify a good initial flow?

3. What is an appropriate way to treat the artificial boundaries?

4. How accurate is the DNS in predicting the stability of small perturbations and in reproducing known results, for example the second Mack mode at high Mach numbers, a laminar separation as predicted by semi-empirical theories
and steady shock boundary layer interactions as conducted experimentally by Hakkinen et al.?

5. What roles can small perturbation theories play in spatial DNS?

6. What is the minimum requirement in terms of CPU time and memory in order to perform a simulation in the turbulent regime using the developed numerical methods?

The general research questions corresponding to the study of physical flow phenomena are formulated in the following.

1. Can we obtain a desired pressure distribution using blowing and suction boundary conditions?

2. What is the influence of two and three-dimensional upstream perturbations and the pressure gradient on a laminar separation bubble flow?

3. How do the fluctuation statistics develop in the streamwise direction in the case of a laminar separation bubble with a turbulent reattachment?

4. What phenomena can occur if a supersonic flow is subjected to an arbitrary suction and blowing, as commonly happens in propulsion systems? Specifically, what physical parameters have an important influence?

5. Is there any criterion, for example a vortex shedding criterion, which is specific to compressible flows under suction and blowing?

Finally, we close this introduction by outlining the contents of this thesis. The next chapters are presented in such a sequence that the emphasis is being shifted from the numerical aspect at the beginning to the physical aspect at the end of the thesis. Moreover, it represents test cases with increasing complexity and difficulties for the numerical approach. Specifically, in Chapter 2, we present the governing equations, and the initial and the boundary conditions. The initial conditions, which are derived from the analytical solution of the zero pressure gradient compressible boundary layer equations are compared to the steady state solution of the Navier-Stokes equations to illustrate their quality. Different techniques to model artificial boundary conditions are presented. In Chapter 3, we provide numerical methods which are appropriate for a spatial DNS. We focus on spatial discretization methods and the numerical treatment at the boundaries. Various techniques for artificial boundary conditions are compared to yield an optimal combination. Further, a buffer domain technique is developed in this chapter. In Chapter 4, the performance of the spatial discretization methods is scrutinized in simulations predicting the stability of small disturbance waves. The results are compared to linear perturbation theories. The buffer function parameters are optimized and the performance of the boundary conditions is further validated. The validated numerical method is applied in Chapter 5 to separated laminar flows followed by vortex shedding or turbulent
reattachment. The results are compared to existing semi-empirical theories and findings by other researchers. Different adverse pressure gradient generators are compared. Moreover, a three dimensional simulation of transition induced by a separation bubble is presented. In Chapter 6, the numerical approach is applied to flows involving shocks and separation. A well known steady shock boundary layer interaction flow is used to validate the numerical methods, especially to test the performance of a high order upwind scheme and the upper boundary conditions which act as a shock generator. The validated method is then applied to shock boundary layer interactions under suction and blowing. In this last application the spatial DNS plays its role to predict flow phenomena which are difficult to describe theoretically and even to generate in a laboratory experiment. We summarize our findings by explicitly answering the research questions and propose recommendations for a future study in Chapter 7.
Chapter 2

Governing equations, initial and boundary conditions

The equations governing compressible viscous flows are the Navier-Stokes equations. Solving these equations, we encounter an initial boundary value problem. In this chapter we focus on the governing equations, the boundary conditions and the initial condition, specific for the compressible flat plate boundary layer flow.

The general form of the Navier-Stokes equations and the wall condition are outlined in Section 1. The choice of the initial condition is presented in Section 2 and the discussion of the inflow, outflow and upper boundary conditions can be found in Section 3.

2.1 The Navier-Stokes equations

The Navier-Stokes equations, excluding external force terms, can conveniently be expressed in conservation form ([25]),

conservation of mass:
\[
\partial_t \rho + \partial_j (\rho u_j) = 0,
\]

conservation of momentum:
\[
\partial_t (\rho u_i) + \partial_j (\rho u_i u_j) + \partial_i p - \partial_j \tau_{ij} = 0,
\]

conservation of energy:
\[
\partial_t e + \partial_j ((e + p) u_j) - \partial_j (\tau_{ij} u_i - q_j) = 0.
\]

The summation convention is used in the system of equations. The symbols \(\partial_t\) and \(\partial_j\) denote the partial differential operators \(\partial/\partial t\) and \(\partial/\partial x_j\) with respect to time \((t)\) and spatial coordinate \((x_j)\) respectively; \(\rho\) is the density, \(p\) the pressure, \(u_i\) the \(i^{th}\) component of the velocity vector, and \(e\) the total energy density which is given for a perfect gas by:
\[
e = \frac{p}{\gamma - 1} + \frac{1}{2} \rho u_i u_i,
\]
where $\gamma$ denotes the adiabatic gas constant. Moreover, $\tau_{ij}$ is the stress tensor which is a function of the dynamic viscosity $\mu$ and velocity vector $\mathbf{u}=(u_1, u_2, u_3)^T$:

$$
\tau_{ij} = \frac{\mu(T)}{Re} \left( \partial_j u_i + \partial_i u_j - \frac{2}{3} \delta_{ij} \partial_k u_k \right), \quad (2-5)
$$

where $Re = (\rho_\infty u_\infty \delta_1^*)/\mu(T_\infty)$ is the reference Reynolds number. The dynamic viscosity $\mu$ is related to the temperature $T$ by Sutherland’s law,

$$
\mu(T) = \frac{1 + C}{T + C} T^{\frac{7}{2}}, \quad (2-6)
$$

where $C = 120 K/T_\infty$. Finally, $q_j$ is the $j^{th}$ component of the heat flux vector, defined as

$$
q_j = -\frac{\mu}{(\gamma - 1)RePrM_\infty^2} \partial_j T, \quad (2-7)
$$

where $Pr$ is the Prandtl number and $M_\infty$ is the reference Mach number. The temperature $T$ is related to the density $\rho$ and the pressure $p$ by the perfect gas law

$$
T = \gamma M_\infty^2 \frac{p}{\rho}. \quad (2-8)
$$

Throughout we use $\gamma = 1.4$ and $Pr = 0.72$. The values of the reference Mach number $M_\infty$ and the Reynolds number $Re$ are specified for each case separately. The above variables have been made dimensionless using reference scales, i.e. a reference length $\delta_1^*$, which is the inflow displacement thickness, density $\rho_\infty$, velocity $u_\infty$, temperature $T_\infty$, pressure and energy density $\rho_\infty u_\infty^2$, viscosity $\mu(T_\infty)$ and time $\delta_1^*/u_\infty$. The subscript $\infty$ refers to the free-stream value.

A no-slip isothermal or adiabatic boundary condition is imposed at the wall. Specifically, the velocity components vanish. For an isothermal wall, the wall temperature is prescribed and the pressure is extrapolated from the interior points consistent with the approximate auxiliary condition $\partial_2 p = 0$, coming from the boundary layer approximation of the wall normal ($x_2$) momentum equation. Hence, no conservation equation needs to be solved along the wall. For an adiabatic wall, the calculation of the wall density and wall pressure is provided by solving the mass and energy conservation equations, respectively. The spatial discretization satisfies the requirement of vanishing normal derivative of temperature at the wall. Further details of the numerical treatment along the boundaries can be found in Chapter 3.

2.2 Solution to the compressible boundary layer equations

For compressible boundary layer flow, an exact solution to the Navier-Stokes equations does not exist. However, we can solve a simplified model of the relevant equations. In the present study of compressible flat plate flow, the Reynolds number is sufficiently high that a thin boundary layer will develop along the wall. Hence,
the solution to the compressible boundary layer equations can be used as the initial condition. Although the initial boundary layer solution can deviate significantly from the flow at a later time in large part of the flow domain (e.g. when separation occurs), it gives a good approximation to the solution at the inflow boundary, provided the flow is undisturbed in the inflow region. A correct inflow condition keeps the computational extent to a minimum, while a good approximation of the initial flow shortens the route to the (statistically) stationary state.

Based on the Navier-Stokes coordinate scaling, the gradients in the streamwise and normal directions within the boundary layer are quite different. We can introduce a new scaling \( x_0' = \frac{Re^{1/2}x_0}{2} \), \( u_0' = \frac{Re^{1/2}u_0}{2} \), and the other quantities are retained. Taking the dominant terms in case \( Re \gg 1 \) from the Navier-Stokes equations based on the new scaling, and further assuming that the flow is stationary \((\partial_t f = 0)\) and two-dimensional \((\partial_3 f = 0, u_3 = 0)\), we come to the compressible boundary layer equations,

\[
\begin{align*}
\frac{\partial}{\partial x_0'} (\rho' u_0') &= 0, \tag{2-9} \\
\rho' u_0' \frac{\partial}{\partial x_0'} u_0' &= -\frac{\partial}{\partial x_0'} p + \frac{\partial}{\partial x_1'} (\mu' \frac{\partial}{\partial x_1'} u_0'), \tag{2-10} \\
\frac{\partial}{\partial x_1'} p &= 0, \tag{2-11} \\
\rho' u_0' \frac{\partial}{\partial x_0'} T' &= (\gamma - 1)M^2 [u_0' \frac{\partial}{\partial x_0'} + \mu' (\frac{\partial}{\partial x_1'} u_0')^2] + \frac{1}{Pr} \frac{\partial}{\partial x_1'} (\mu' \frac{\partial}{\partial x_1'} T'). \tag{2-12}
\end{align*}
\]

This system of equations allows for a special, so-called similarity solution. For this purpose we introduce a coordinate transformation \[75\]:

\[
\begin{align*}
\xi &= x_1'(= x_1) ; \quad \eta = \frac{1}{(2x_1')^{1/2}} \int_0^{x_2'} \rho'(x_1', s) ds = \left( \frac{Re}{2x_1} \right)^{1/2} \int_0^{x_2} \rho(x_1, s) ds \tag{2-13}
\end{align*}
\]

and a stream function \( \Psi \) through

\[
\begin{align*}
\rho' u_0' &= \frac{\partial}{\partial x_0'} \Psi ; \quad \rho' u_2' = -\frac{\partial}{\partial \eta} \Psi, \tag{2-14}
\end{align*}
\]

which implies that the continuity equation is identically satisfied. This transformation implies

\[
\begin{align*}
\frac{\partial}{\partial x_1'} &= \frac{\partial}{\partial \xi} + (\frac{\partial}{\partial \eta}) \frac{\partial}{\partial \eta}, \tag{2-15} \\
\frac{\partial}{\partial x_2'} &= \frac{\rho'}{(2\xi)^{1/2}} \frac{\partial}{\partial \eta}. \tag{2-16}
\end{align*}
\]

Substituting this transformation into the system of equations 2-9 - 2-12 we find, using the representation \( \Psi = (2\xi)^{1/2} F(\xi, \eta) \):

\[
\begin{align*}
\frac{\partial}{\partial \eta} (\rho_0 \frac{\partial}{\partial \eta} F) + F \frac{\partial}{\partial \eta} F &= \frac{2\xi}{\rho'} \frac{\partial}{\partial \xi} p + 2\xi \{ \partial_\eta F \partial_\eta \eta F - \partial_\xi F \partial_\eta \eta F \}, \tag{2-17} \\
\partial_\eta p &= 0, \tag{2-18}
\end{align*}
\]
\[ \partial_\eta \left( \frac{\chi}{\rho'} \partial_\eta T' \right) + F \partial_\eta T' + (\gamma - 1)M^2 \chi (\partial_\eta F)^2 = \\
- (\gamma - 1)M^2 \frac{2\xi \partial_\eta F}{\rho'} \partial_\xi p + 2\xi \{ \partial_\eta F \partial_\xi T' - \partial_\xi F \partial_\eta T' \}, \] (2-19)

where we put \( \chi = \rho' \mu' \). These equations greatly simplify if we construct similarity solutions, i.e. we consider solutions in which \( F \) and \( T' \) depend on \( \eta \) only. Moreover, we restrict to constant pressure boundary layers in which \( p = 1/(\gamma M^2) \). In that case we finally arrive at

\[ d_\eta (\chi d_\eta F) + F d_\eta F = 0, \] (2-20)

\[ d_\eta \left( \frac{\chi}{\rho'} d_\eta T' \right) + F d_\eta T' + (\gamma - 1)M^2 \chi (d_\eta F)^2 = 0, \] (2-21)

which is a system of ordinary differential equations for \( F \) and \( T' \). The boundary conditions for the above equations can be specified as follows. At no-slip walls we have \( u'_1 = d_\eta F = 0 \) and in view of the nondimensionalization we have \( u'_1 = d_\eta F \to 1 \) as \( \eta \to \infty \). Moreover, at adiabatic walls \( d_\eta T' = 0 \), whereas at isothermal walls the relation \( T' = T_w \) holds. In view of the scaling we have \( T' \to 1 \) as \( \eta \to \infty \) and \( u'_2 = 0 \) at the wall implies \( F = 0 \) at the wall. These boundary conditions fully specify the similarity solution.

The main problem is that some of these boundary conditions are available at the wall \( (\eta = 0) \), whereas others are given in the far-field. A shooting method is adopted for solving this system of equations. The solution to this system of equations can be used to reconstruct the desired flow field since use can be made of the relations \( u'_1 = d_\eta F \) and \( \rho' = 1/T' \) in view of the constant pressure. This results in \( u'_1, \rho', T' \), and \( p \), which are valid for both parallel and non-parallel flow. For \( u'_2 \) a different procedure is followed. For parallel flow we specify \( u'_2 = 0 \), whereas for non-parallel flow we specify the normal velocity profile according to Stewartson ([75]),

\[ u'_2 = -T' \{(2\xi)^{-1/2} F + (2\xi)^{1/2} (\partial'_\xi \eta) u_1 \}. \] (2-22)

The energy density \( e' \) follows from (2-4), thus defining an initial field for the simulation. As a result the field described by \( \{ \rho', \rho' u'_1, e' \} \) as a function of \( \eta \) is obtained and in order to find the dependence on the original coordinate, \( x_1, x_2 \), the transformation is inverted using

\[ x_2 = \left( \frac{2x_1}{Re} \right)^{1/2} \int_0^\eta T(s) ds, \] (2-23)

where the integration is performed using the trapezoidal rule. Notice that only the ratio \( (2x_1)/Re \) is of relevance, not the value of \( x_1 \) itself. This implies that the solution can be obtained at some location \( x_1 \) and based on this the solution at a different location \( \hat{x}_1 \) can be found straightforwardly provided we scale \( \hat{x}_2 = (\sqrt{x_1/\hat{x}_1})x_2 \). This expresses the similarity property of the solution. We can then choose \( x_1 \) at the
inflow boundary such that the displacement thickness equals unity, which is generally convenient for the initialization. With this procedure we obtain the desired solution at some $x_1$ location, as a function of $x_2$ for a specific grid in the $x_2$ direction. For a parallel flow the full mean initial field is found by copying the above result to all $(x_1, x_2)$, whereas for a non-parallel flow the similarity property is used to define the mean field at the other streamwise locations. In the latter case, if desired, the values of $u_2$ for the freestream portion of the inflow profile are approximated by the outer expansion of the perturbation theory for the incompressible flow over a flat plate (Van Dyke [19], p.135):

$$u_2 = (u_2)_{edge} \left(1 - \frac{3}{8} \frac{x_2}{(x_1)_{in}}\right), \quad (2-24)$$

where $(u_2)_{edge}$ and $(x_1)_{in}$ are the value of $u_2$ at the edge of the boundary layer and the distance from the leading edge of the flat plate to the inflow position, respectively. For a compressible flow, this is a good approximation, provided that the density is nearly constant in the freestream part of the flow (Rudy and Strikwerda [65]).

The correctness of the inflow profiles, i.e. the compressible Blasius boundary layer, is examined by applying the calculation at a low Mach number (we take $M = 0.05$) and comparing the result with the incompressible theory. For incompressible boundary layer the following asymptotical values are valid ([68]):

$$\delta^* = 1.721 \left(\frac{x_1}{Re}\right)^{\frac{1}{2}}, \quad (2-25)$$

$$\theta = 0.6641 \left(\frac{x_1}{Re}\right)^{\frac{1}{2}}, \quad (2-26)$$

$$(u_2)_{max} = \frac{0.8604}{(x_1 Re)^{\frac{1}{2}}}, \quad (2-27)$$

$$c_f = \frac{0.332}{(x_1 Re)^{\frac{1}{2}}}, \quad (2-28)$$

where $\theta$ is the boundary layer momentum thickness and $c_f$ is the skin friction coefficient. Using $Re = 100$, this suggests that the value $\delta^* = 1$ corresponds to

$$x_1 = 33.76,$$
$$\theta(x_1) = 0.3859,$$
$$(u_2)_{max}(x_1) = 0.014800,$$
$$c_f(x_1) = 0.00571,$$

The corresponding values from the compressible boundary layer calculation with $M = 0.05$ are 33.73, 0.3857, 0.014808 and 0.00574, respectively, which confirm the correctness of the procedure followed.

To examine the performance of the similarity transformation, and therefore the quality of the initial condition, we compare the similarity solution to the compressible
boundary layer equations and a converged Navier-Stokes solution. The Reynolds number is 100 based on the inflow displacement thickness and the Mach number is 0.5. For the Navier-Stokes calculation, the domain length and height are 30, covered by $64 \times 32$ grid points in the $x_1$ and the $x_2$ direction, respectively. The grid is uniform in the $x_1$ direction and stretched in such a way in the $x_2$ direction that the resolution becomes higher toward the wall. Given the Reynolds number and the Mach number, the inflow boundary is located at $x_1 = 30$ and the outflow boundary at $x_1 = 60$. The inflow profiles are those of the compressible Blasius solution, and we compare the data at $x_1 = 55.65$. For the treatment of the artificial, i.e. inflow, outflow and freestream, boundaries, we use the reference setting as defined in Chapter 3. The convergence to a steady state is reached when the root mean square of the wall pressure,

$$ R(p) = 2 \left( \frac{1}{N} \sum_{i=1}^{N} \left( \frac{p^{i+\Delta t} - p^i}{\Delta t} \right)^2 \right)^{\frac{1}{2}}, \quad (2-29) $$

is smaller than $10^{-8}$, where $N$ is the number of grid points on the wall. The comparison for the velocity components, $u_1$ and $u_2$, is shown in Fig. 2-1a, and the comparison for density and energy can be seen in Fig. 2-1b. The quantities are normalized by their value at the edge of the boundary layer. In spite of the quite low Reynolds number and the coarse grid used in the Navier-Stokes calculations, the agreement is quite good. We observe a better agreement for higher Reynolds numbers and/or grid resolutions.

**Figure 2-1:** Velocity components (a), density and energy (b) at $x_1 = 55.65$, normalized by the edge values, obtained from similarity solution (solid) and Navier-Stokes computation (dashed).
2.3 Artificial boundary conditions

Numerical simulations necessitate a truncation of the computational domain by artificial boundaries. The artificial boundaries bounding the rectangular computational domain used for the present applications are the inflow, outflow and upper boundaries as sketched in Fig. 1-5. Since these boundaries are close to the plate, the flow outside the computational domain is different from an undisturbed, uniform flow; it is affected by disturbances inside the domain. On the other hand, in order to calculate the interior flow, we need some external flow information. This mutual requirement of informations makes the condition along the artificial boundaries unknown exactly in most flow applications and therefore should be approximated. We remark that although in this section we consider artificial boundary conditions, not all boundaries discussed are artificial boundaries. A suction/blowing or a pressure gradient upper boundary is a physical boundary as shifting the location of this boundary will affect the physics of the flow inside the computational domain. Their inclusion in this section is due to the fact that they are modelled in the same way as the artificial boundaries.

Mathematically, the boundary conditions of a system of equations are subject to a certain requirement in order for the problem to be well posed. The number of physical boundary conditions for the well-posedness requirement for the Euler and Navier-Stokes equations has been derived by Strikwerda in [78] and stated in Table 1 for two and three-dimensional cases. These boundary conditions are provided by some information about the external flow, adjacent to the boundaries. We call these boundary conditions physical boundary conditions. In some cases, however, no accurate external flow information is available, such as at the outflow boundary. Although mathematically only a certain number of boundary conditions is required, depending on the local flow condition, numerically, we should specify all the dependent variables. We call the conditions completing the specification of the dependent variables the numerical boundary conditions. Whereas the mathematical requirement for well-posedness has found a general acceptance, there is still no general method to specify the numerical boundary conditions and to define the physical boundary conditions if the required external information is lacking. Different methods are used in literature.

For the present applications an extrapolation method and a characteristic method are considered. The extrapolation method can be used for the specification of the numerical boundary conditions, but it cannot serve as an alternative physical boundary condition, in case no external information is available. On the contrary, the characteristic method can provide numerical boundary conditions as well as alternative physical boundary conditions. In the derivation of the characteristic method a distinction can be made between reflecting and non-reflecting conditions. This knowledge provides us with some freedom in selecting one which is suitable for a local flow condition (for instance, the inviscid method of characteristics is not suitable in boundary layers). Moreover, we can derive a new type of boundary condition which is of interest for the present applications, such as a blowing and suction boundary
Table 2-1: Number of physical boundary conditions required for well-posedness for two and three-dimensional flows.

<table>
<thead>
<tr>
<th>Boundary type</th>
<th>Dimension</th>
<th>2-dimensional</th>
<th>3-dimensional</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Euler</td>
<td>Navier-Stokes</td>
</tr>
<tr>
<td>Supersonic inflow</td>
<td></td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>Subsonic inflow</td>
<td></td>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>Supersonic outflow</td>
<td></td>
<td>0</td>
<td>3</td>
</tr>
<tr>
<td>Subsonic outflow</td>
<td></td>
<td>1</td>
<td>3</td>
</tr>
</tbody>
</table>

condition, by combining appropriate properties of the basic principles.

From numerical calculations, we observe that fulfilling the number of physical boundary conditions according to the theory for a hyperbolic system (Euler equations) is sufficient. Satisfying the number of conditions for the Navier-Stokes equations by adding some viscous conditions (see [59]) results only in small differences. Therefore, we follow the inviscid approach. The equivalence of the results can be understood from the fact that in the present applications the viscous effects are restricted to the region near the wall.

In the following description of the methods, we consider two-dimensional flow, corresponding to four dependent variables. The extension to three-dimensional flow is straightforward.

2.3.1 Extrapolation method

Along an artificial boundary, some dependent variables are unspecified by the physical boundary conditions. To complete the specification of the variables, we simply extrapolate dependent variables from interior points to the boundary. The choice of dependent variables for the physical and numerical boundary conditions is not unique. The following are appropriate combinations based on our experience.

First, we consider a subsonic inflow boundary. According to Table 1, we should prescribe three physical boundary conditions. In the case of an isothermal wall, the appropriate physical boundary conditions at the inflow boundary are provided by the velocity components and the temperature. The fourth variable can then be specified by extrapolating the pressure or the density from the interior points, which represents the numerical boundary condition. In the case of an adiabatic wall, it is more suitable to prescribe the the pressure or the density instead of the temperature to accommodate the adiabatic condition, $\frac{\partial T}{\partial z} = 0$ at the wall.

In the case of a subsonic outflow boundary, we need one physical and three numerical boundary conditions. The physical boundary condition can be satisfied...
by prescribing one dependent variable, for instance the pressure, although this will cause reflections of existing disturbance waves at the outflow boundary, as will be discussed in Section 2.1. The numerical conditions are satisfied by extrapolating the velocity components and the temperature from the interior points.

We consider now a subsonic upper boundary. Depending on the sign of the normal velocity, we regard it as an inflow upper boundary (negative sign) or an outflow upper boundary (positive sign). From this point, we can proceed in the same way as in the previous cases. The pressure can be used as a physical boundary condition, for instance by prescribing a pressure jump in the case we want to invoke a steady shock from the upper boundary.

The procedure for a supersonic boundary follows accordingly. We prescribe all the dependent variables at the inflow boundary (no numerical boundary conditions) and extrapolate them from the interior points at the outflow boundary (no physical boundary conditions).

2.3.2 Characteristic method

The characteristic method described here was originally proposed by Hedstrom ([33]) for one-dimensional cases. Thompson performed an extension to multi-dimensional problems and non-rectangular coordinate systems. The advantage of the characteristic method is that it can provide numerical as well as alternative physical boundary conditions in case the external flow is unknown. In the latter case, the so-called non-reflecting boundary condition replaces the required physical boundary condition. This boundary condition is extracted by invoking the non-reflecting principle (Hedstrom) in the characteristic form of the Euler equations. Some authors combine the extrapolation technique and the non-reflecting property of the one-dimensional characteristic method for their boundary conditions, for example Rudy & Strikwerda ([65]). Others use a quasi multi-dimensional characteristic method (Thompson [79] and Poinsot & Lele [59]). Both are described in the following.

**One-dimensional**

The one-dimensional characteristic method is not directly applicable to two-dimensional flows. However, it provides a useful non-reflecting condition, which can be applied indirectly, i.e. in combination with the extrapolation method, to two-dimensional flows. We depart from the one-dimensional Euler equations, which form a hyperbolic system. In conservative form, it can be written as,

$$\partial_t \tilde{U} + \partial_1 F = 0,$$

with

$$\tilde{U} = \begin{pmatrix} \rho \\ \rho u_1 \\ e \end{pmatrix}, \quad F = \begin{pmatrix} \rho u_1 \\ \rho u_1^2 + p \\ (e + p)u_1 \end{pmatrix},$$

where
In primitive form, the system becomes,

$$\partial_t U + A \partial_t U = 0,$$

(2-31)

with

$$U = \begin{pmatrix} \rho \\ u_1 \\ s \end{pmatrix}, \quad A = \begin{pmatrix} \rho & 0 \\ u_1 & \rho \\ p/(\rho s) & 0 \end{pmatrix},$$

(2-32)

where \(s = p\rho^{-\gamma}\) is a measure of the entropy and \(c = (\gamma p/\rho)^{\frac{1}{2}}\) is the sound speed. The eigenvalues of \(A\) are real and equal to \(\lambda_1 = u_1 - c\), \(\lambda_2 = u_1\) and \(\lambda_3 = u_1 + c\), and the corresponding left eigenvectors are given by

\[
\begin{align*}
    l_1 &= [-c, \rho, -\frac{p}{\rho s}]^T, \\
    l_2 &= [0, 0, 1]^T, \\
    l_3 &= [c, \rho, \frac{p}{\rho s}]^T,
\end{align*}
\]

respectively. Let \(L\) be a matrix whose rows are the left eigenvectors \(l_i\). Then \(A\) can be diagonalized by the transformation

$$L AL^{-1} = \Lambda,$$

(2-33)

where \(\Lambda\) is a diagonal matrix with \(\Lambda_{ii} = \lambda_i\), and the columns of \(L^{-1}\) are the right eigenvectors of \(A\). Multiplying Eq. (2-31) by \(L\) gives

$$L \partial_t U + \Lambda L \partial_t U = 0,$$

(2-34)

or

$$l_i \partial_t U + \lambda_i l_i \partial_t U = 0,$$

(2-35)

or more specifically,

$$\begin{pmatrix} -c & \rho & -\frac{p}{\rho s} \\ 0 & 0 & 1 \\ c & \rho & \frac{p}{\rho s} \end{pmatrix} \partial_t \begin{pmatrix} \rho \\ u_1 \\ s \end{pmatrix} + \Lambda \begin{pmatrix} -c & \rho & -\frac{p}{\rho s} \\ 0 & 0 & 1 \\ c & \rho & \frac{p}{\rho s} \end{pmatrix} \partial_t \begin{pmatrix} \rho \\ u_1 \\ s \end{pmatrix} = 0.$$

(2-36)

Taking \(s\) as a primitive variable simplifies the eigenvalue calculation, but is inconvenient for numerical work. Therefore we eliminate \(s\) in favor of \(\rho\) and \(p\) and obtain the characteristic equations

$$\partial_t p - \rho c \partial_t u_1 + \lambda_1(\partial_t p - \rho c \partial_t u_1) = 0,$$

(2-37)

$$\partial_t p - c^2 \partial_t \rho + \lambda_2(\partial_t p - c^2 \partial_t \rho) = 0,$$

(2-38)

$$\partial_t p + \rho c \partial_t u_1 + \lambda_3(\partial_t p + \rho c \partial_t u_1) = 0.$$

(2-39)
We can write (2-35) as
\[ \partial_t V_i + \lambda_i \partial_1 V_i = 0, \]  
(2-40)
where \( dV_i = l_i dU \). (2-40) is as set of wave equations with characteristic velocities \( \lambda_i \). In the \((x,t)\)-plane, each wave amplitude \( V_i \) is constant along the curve \( dx/dt = \lambda_i \).

The boundary condition is then specified by invoking the non-reflecting principle, due to Hedstrom ([33]). This principle can be formulated in the following way (see Thompson [79]): \textit{the amplitudes of the incoming waves are constant, in time, at the boundaries.} Mathematically, this implies
\[ \frac{\partial V_k}{\partial t} = 0, \]  
(2-41)
or
\[ l_k \frac{\partial U}{\partial t} = 0, \]  
(2-42)
with \( k \) is the subscript corresponding to the characteristic wave directed inward. For example, if we consider the subsonic outflow boundary, this yields
\[ \partial_t p - \rho c \partial_t u_1 = 0. \]  
(2-43)
The general equations at the boundary become
\[ l_i \partial_t U + \ell_i = 0 \]  
(2-44)
where
\[ \ell_i = \begin{cases} \lambda_i l_i \partial_1 U & \text{for outgoing waves} \\ 0 & \text{for incoming waves} \end{cases} \]  
(2-45)
From (2-38)-(2-39) and (2-44) we find
\[ \partial_t p = -\frac{1}{2}(\ell_3 + \ell_1), \]  
(2-46)
\[ \partial_t \rho = \frac{1}{c^2}(-\frac{1}{2}(\ell_3 + \ell_1) + \ell_2), \]  
(2-47)
\[ \partial_t u = -\frac{1}{\rho c} (\ell_3 - \ell_1), \]  
(2-48)
and \( \partial_t \tilde{U} \) in (2-30) can be obtained by
\[ \partial_t (\rho u_1) = u_1 \partial_t \rho + \rho \partial_t u_1, \]  
(2-49)
\[ \partial_t e = \frac{1}{2} w_1^2 \partial_t \rho + \rho u_1 \partial_t u_1 + \frac{1}{\gamma - 1} \partial_t p. \]  
(2-50)
Note that the condition \( \ell_k = 0 \) acts as a physical boundary condition and the outward directed \( \ell_i \), which is calculated using interior variables, acts as a numerical boundary condition.
Although this one-dimensional characteristic method is basically designed for one-dimensional problems, Rudy & Strikwerda ([65]) used the idea of the non-reflecting condition to specify the pressure at the subsonic outflow boundary, while the other variables are determined using the extrapolation technique. Specifically, $u_1$, $u_2$ and $T$ are linearly extrapolated from the interior data, and the non-reflecting condition (2-43) is modified to

$$
\partial_t p - \rho c \partial_x u_1 + \beta (p - p_\infty) = 0,
$$

where

$$
\beta = \kappa \frac{c^2 - u_1^2}{c L_x}.
$$

Here, $u$ and $c$ are the local velocity and sound speed on the outflow boundary, $L_x$ is the length of the domain and $\kappa$ is a constant as a tuning parameter. For $\kappa = 0$, Eq. (2-51) turns into the original non-reflecting condition (2-43). If $\kappa > 0$ then the condition (2-51) is partly non-reflecting; variation of the outflow pressure is restricted around $p_1$. The theoretical value of $\kappa$ is analyzed by Rudy & Strikwerda in [66].

At the subsonic inflow boundary, Rudy & Strikwerda ([65] also employed the characteristic method to specify the numerical boundary condition. Specifically, for an isothermal wall, $u_1$, $u_2$ and $T$ are prescribed at the inflow boundary, and the pressure is found from the following condition for the amplitude of the outgoing wave

$$
\partial_x p - \rho c \partial_x u_1 = 0.
$$

It means that there is no outgoing wave, as it is the change in amplitude which indicates a wave. The reason for this choice is unclear.

**QUASI TWO-DIMENSIONAL**

The derivation for the quasi two-dimensional case is analogous to the one-dimensional case. We consider a boundary perpendicular to the streamwise direction. Let us write the two-dimensional Navier-Stokes equations in the following form

$$
\partial_t \rho + d_1 + \partial_2 m_2 = 0,
$$

$$
\partial_t e + \frac{1}{2} (u_k u_k) d_1 + \frac{d_2}{\gamma - 1} + m_1 d_3 + m_2 d_4 + \partial_2 [(e + p) u_2] = \partial_i (u_j \tau_{ij}) - \partial_i q_i,
$$

$$
\partial_t m_1 + u_1 d_1 + \rho d_5 + \partial_2 (m_1 u_2) = \partial_j \tau_{ij},
$$

$$
\partial_t m_2 + u_2 d_1 + \rho d_4 + \partial_2 (m_2 u_2) + \partial_2 p = \partial_j \tau_{2j},
$$

where $m_i = \rho u_i$ is the $x_i$ direction momentum density (flow rate). The above system is different from the system of equations (2-1)-(2-3) in the terms containing
derivatives with respect to the $x_1$ direction ($d_1$ to $d_4$). The vector $d$ is written in terms of the amplitudes of characteristic waves:

$$ d = \begin{pmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \end{pmatrix} = \begin{pmatrix} d_1 \\ \frac{1}{2} (j_4 + j_1) \\ \frac{1}{2\rho c} (j_4 - j_1) \end{pmatrix} = \begin{pmatrix} \partial_1 m_1 \\ \partial_1 (c^2 m_1) + (1 - \gamma) u_1 \partial_1 p \\ u_1 \partial_1 u_1 + \frac{1}{\rho} \partial_1 p \\ u_1 \partial_1 u_2 \end{pmatrix} \tag{2-58} $$

The $j_i$'s are the amplitudes of characteristic waves with respect to the $x_1$ direction associated with each characteristic velocity $\mu_i$. These velocities are:

$$ \begin{align*}
\mu_1 &= u_1 - c, \\
\mu_2 &= \mu_3 = u_1, \\
\mu_4 &= u_1 + c,
\end{align*} \tag{2-59, 2-60, 2-61} $$

where $c$ is the speed of sound, given by:

$$ c^2 = \frac{\gamma p}{\rho}. \tag{2-62} $$

The amplitudes of the characteristic waves $j_i$ are defined as:

$$ \begin{align*}
j_1 &= \mu_1 (\partial_1 p - \rho c \partial_1 u_1), \\
j_2 &= \mu_2 (c^2 \partial_1 p - \partial_1 p), \\
j_3 &= \mu_3 \partial_1 u_2, \\
j_4 &= \mu_4 (\partial_1 p + \rho c \partial_1 u_1),
\end{align*} \tag{2-63, 2-64, 2-65, 2-66} $$

$\mu_1$ and $\mu_4$ are the velocities of acoustic waves, $\mu_2$ is the convection velocity while $\mu_3$ is the velocity at which $u_2$ is advected in the $x_1$ direction.

Clearly, on the inflow boundary $j_1$ is the amplitude of the outgoing wave and the others are the amplitudes of the incoming waves, while on the outflow boundary $j_1$ is the incoming one and the others are the outgoing ones. The $j_i$'s of both incoming and outgoing waves crossing the boundaries should now be determined in order to provide the value of the vector $d$. The amplitudes of the outgoing waves are known from the computation within the flow domain, while those of the incoming waves should then be determined from the known outgoing ones and the information outside the domain. There is no exact method to specify the values of $j_i$'s of the incoming waves for the multidimensional NS equations. However this can approximately be done by examining the local associated one-dimensional inviscid (LODI) relations. This is the approximation that one must make in this method. The LODI system can be cast in many different forms depending on the choice of variables. The important
relations are:

\[
\begin{align*}
\partial_t p &= -\frac{1}{2}(j_4 + j_1), \\
\partial_t u_1 &= -\frac{1}{2\rho c}(j_4 - j_1), \\
\partial_t u_2 &= -j_3, \\
\partial_t \rho &= -\frac{1}{c^2}\left[\frac{1}{2}(j_4 + j_1) + j_2\right],
\end{align*}
\]

(2-67) (2-68) (2-69) (2-70)

from which the corresponding equation for the temperature is

\[
\partial_t T = -\frac{T}{\rho c^2}\left[-j_2 + \frac{1}{2}(\gamma - 1)(j_4 + j_1)\right].
\]

(2-71)

Note that these relations are analogous to (2-47)-(2-48) in the one-dimensional method.

Given the above information, Thompson implemented the non-reflecting boundary conditions according to (2-44) and (2-45), which means that no external flow information is used. Poinsot & Lele ([59]) extended the application to the Navier-Stokes equations, by adding some conditions for the viscous terms, and to cases where some known external information is exploited. In the present work, we follow a different procedure in that the viscous conditions are excluded and a criterion when the non-reflecting property should be imposed is added.

The present procedure consists of four steps:

**Step 1:**
Select the inviscid physical boundary conditions to be imposed and eliminate the corresponding conservation equations from the system of equations (2-54)-(2-57). Each physical boundary condition has a typical counterpart conservation equation. If one chooses \(u_1, u_2\) and \(T\) to be imposed, the equations to be eliminated are the \(x_1\) and the \(x_2\) component of the momentum equations, and the energy equation respectively. Table II in [59] provides examples of such choices.

**Step 2:**
For each physical boundary condition, use the corresponding LODI relations to express the unknown \(j_i\)'s as a function of the known \(j_i\)'s and the imposed variables.

**Step 3:**
If the external information does not complete the physical boundary conditions, use the non-reflecting condition (\(j_i\)'s corresponding to the incoming waves are set to zero) to replace the missing physical conditions. If desired, we also can use a partly reflecting condition instead of perfectly reflecting, in which an incoming wave is related to some external information. For instance, at a subsonic outflow boundary \(j_1 = \beta(p - p_\infty)\) instead of \(j_1 = 0\), where \(\beta\) is given by Eq. (2-52).

**Step 4:**
Compute the viscous terms in the same way as in the interior domain and use the remaining conservation equations after substituting the values of the \(j_i\)'s obtained from step 2 and 3 to specify all the variables which are not imposed.
In this way we can freely choose the imposed variables, which match our interest, and allow an inflow/outflow alternation in time as well as in space along the boundary. In the following, we apply the quasi two-dimensional method for subsonic artificial boundaries. The supersonic boundary treatment is in general easier.

**Subsonic inflow:**

**Step 1:** According to Table 1, three physical boundary conditions should be imposed. The inflow temperature and the velocities $u_1$ and $u_2$ are imposed, so that we do not need to solve the conservation equations for energy (2-57) and momentum (2-55) and (2-56).

**Step 2:**

The unknown outgoing waves $j_2$, $j_3$, and $j_4$ are specified by the known incoming wave $j_1$ in combination with the imposed external information. As the inflow velocities $u_1$ and $u_2$ are imposed, the LODI relations (2-68) and (2-69) suggest:

$$j_4 = j_1 - 2pc\partial_t u_1$$

$$j_3 = -\partial_t u_2$$

As the inflow temperature is imposed, the LODI relation (2-71) gives:

$$j_2 = \frac{1}{2} (\gamma - 1)(j_4 + j_1) + \frac{pc^2}{T}\partial_t T.$$  

For a time independent inflow, $\partial_t u_1 = \partial_t u_2 = \partial_t T = 0$.

**Step 3:**

Since all physical boundary conditions are satisfied by the external flow information, in this case the compressible laminar boundary layer solution, we do not need to impose the non-reflecting condition.

**Step 4:**

The remaining equation is the conservation equation of mass, which does not contain viscous terms. The density $\rho$ can now be advanced in time by solving (2-54) after substituting the wave amplitudes $j_1$, $j_2$, and $j_4$ in $d_1$. The $\partial_2$ term is calculated in the same way as in the interior domain. $j_1$ is computed from interior points (known outgoing wave) using Eq. (2-63). $j_4$ and $j_2$ have been determined at step 2. In this case $j_3$ is not needed.

**Subsonic non-reflecting outflow:**

**Step 1:** From Table 1, we need one physical boundary condition. However, the external flow adjacent to the outflow boundary is unknown. Some flow information far downstream can be estimated, for instance the pressure is approximately $p_{\infty}$, but
this condition does not fix any of the dependent variables on the boundary and we retain all the conservation equations (2-54)-(2-57).

**Step 2:**
No real physical boundary condition is available to relate the unknown incoming waves to the known outgoing waves.

**Step 3:**
Since no real physical boundary condition can be used, we impose a partly non-reflecting condition. Therefore we employ the pressure at infinity to obtain the amplitude variation of the incoming wave $j_1$:

$$j_1 = \beta(p - p_\infty) \quad (2-75)$$

where $\beta$ is defined in (2-52). A perfectly non-reflecting condition is obtained for $\beta = 0$. If $\beta \neq 0$ (2-75) keeps the value of $p$ close to $p_\infty$. The higher the value of $\beta$ the smaller the difference. A typical value of $\kappa$ in Eq. 2-52 for the subsonic outflow boundary is 1000.

**Step 4:**
The viscous terms are computed according to the interior domain. All the $j_i$’s with $i \neq 1$ are estimated from the interior points. $j_1$ is given by (2-75) and (2-54)-(2-57) are used to advance the solution in time at the outflow boundary.

**Subsonic upper boundary:**

For the upper boundary, the treatment is applied in the $x_2$ direction instead of the $x_1$ direction. The $\partial_2$ terms are now subjected to the boundary conditions, whereas the $\partial_1$ terms are computed as in the interior domain. Three types of upper boundary condition are used in the present applications: non-reflecting, pressure prescription and blowing/suction boundary conditions. The first is the original artificial boundary condition by Thompson ([79]), the others are specially derived for the present applications. For all the three types, the normal velocity can be positive (outflow) as well as negative (inflow).

In the non-reflecting upper boundary condition, the external flow is unknown. Therefore, no real physical boundary conditions are imposed. If $u_2 < 0$ (inflow), the three required physical boundary conditions are provided by setting the amplitude of the incoming waves, propagating with velocity $u_2 - c$ and $u_2$, to zero. The amplitude of the outgoing wave, propagating with velocity $u_2 + c$ is calculated using the interior information. The treatment for $u_2 > 0$ (outflow) , where we have three outgoing waves and one incoming, is similar.

For the pressure prescription boundary condition, we use a partly reflecting condition which contains the information about the pressure distribution. No real physical boundary conditions are prescribed, so we solve all the conservation equations. The incoming wave amplitude corresponding to velocity $u_2 - c$ is now equal to $\beta(p - p_u)$ with $p_u$ is the imposed pressure distribution along the upper boundary. $\beta$ is selected such that the difference between $p$ and $p_u$ is small. The value of $\kappa$ in
Eq. 2-52 for the pressure prescription boundary is the same as that for the subsonic outflow boundary ($\approx 1000$). The amplitudes of the other two incoming waves in the case of $u_2 < 0$ are set to zero. In this way we get the desired pressure distribution along the upper boundary and at the same time minimize reflections.

In the blowing/suction boundary condition, alternation between blowing ($u_2 > 0$) and suction ($u_2 < 0$) is prescribed along the upper boundary. Therefore, we do not need to solve the $x_2$ momentum equation at the boundary. Analogous to the LODI relation (2-68), the prescribed normal velocity relates the incoming wave amplitude $j_1$ to the outgoing $j_4$, which is known from the interior information. In the region where $u_2 > 0$, the other two waves, $j_2$ and $j_3$, are known outgoing waves, while in the region where $u_2 < 0$ these waves are incoming and are set to zero. In other words, in the latter case we incorporate the non-reflecting condition to $j_2$ and $j_3$. The calculation of the dependent variables at the boundary is then obvious.

2.3.3 Numerical comparison

The various methods for the specification of boundary conditions in the previous subsections provide us with some alternatives. We perform a numerical comparison between those methods, to arrive at the most suitable combination. Therefore we perform a steady state calculation of undisturbed laminar flow along an isothermal wall using the various methods of boundary treatment. The same calculation as in the comparison between the Navier-Stokes and the similarity solution to the compressible boundary layer equations is carried out. We identify the methods for boundary condition as follows.

- **Inflow:**
  - $iextr$: inflow extrapolation method ($u_1$, $u_2$, $T$ are prescribed and $p$ is extrapolated).
  - $ihybr$: inflow hybrid method ($u_1$, $u_2$, $T$ are prescribed and $p$ is calculated using (2-53)).
  - $ichar$: inflow characteristic method ($u_1$, $u_2$, $T$ are prescribed and $p$ is updated using the quasi 2D characteristic method)

- **Outflow:**
  - $oextr$: outflow extrapolation method ($p = p_\infty$ is prescribed and other variables are extrapolated).
  - $ohper$: outflow hybrid method of perfectly non-reflecting ($p$ follows from (2-51) with $\beta = 0$ and other variables are extrapolated).
  - $ohpar$: outflow hybrid method of partly reflecting ($p$ follows from (2-51) with $\beta \neq 0$ and other variables are extrapolated).
Table 2-2: Combination of methods for artificial boundary condition treatment. * denotes the reference combination. + means the solution converges and – does not converge.

<table>
<thead>
<tr>
<th>No.</th>
<th>variation of inflow</th>
<th>outflow</th>
<th>upper</th>
<th>convergence</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Inflow boundary</td>
<td>iextr</td>
<td>ocpar</td>
<td>fperf</td>
</tr>
<tr>
<td>2</td>
<td>ihybr</td>
<td>ocpar</td>
<td>fperf</td>
<td>+</td>
</tr>
<tr>
<td>3</td>
<td>ichar</td>
<td>ocpar</td>
<td>fperf</td>
<td>–</td>
</tr>
<tr>
<td>4</td>
<td>Outflow boundary</td>
<td>iextr</td>
<td>oextr</td>
<td>fperf</td>
</tr>
<tr>
<td>5</td>
<td>iextr</td>
<td>ohpar</td>
<td>fperf</td>
<td>+</td>
</tr>
<tr>
<td>6</td>
<td>iextr</td>
<td>ohper</td>
<td>fperf</td>
<td>+</td>
</tr>
<tr>
<td>1</td>
<td>iextr</td>
<td>ocpar</td>
<td>fperf</td>
<td>+ *</td>
</tr>
<tr>
<td>7</td>
<td>iextr</td>
<td>ocper</td>
<td>fperf</td>
<td>–</td>
</tr>
<tr>
<td>8</td>
<td>Freestream boundary</td>
<td>iextr</td>
<td>ocpar</td>
<td>fextr</td>
</tr>
<tr>
<td>1</td>
<td>iextr</td>
<td>ocpar</td>
<td>fperf</td>
<td>+ *</td>
</tr>
<tr>
<td>9</td>
<td>iextr</td>
<td>ocpar</td>
<td>fpres</td>
<td>+</td>
</tr>
</tbody>
</table>

- **ocper**: outflow perfectly non-reflecting of the quasi 2D characteristic method.
- **ocpar**: outflow partly reflecting of the quasi 2D characteristic method.

Freestream:
- **fextr**: upper extrapolation method \((p = p_\infty \text{ is prescribed and other variables are extrapolated})\).
- **fperf**: upper perfectly non-reflecting of the quasi 2D characteristic method.
- **fpres**: upper pressure prescription (partly reflecting) of the quasi 2D characteristic method.

The combinations of inflow, outflow and upper boundary conditions presented in Table 2-2 are performed. The convergence of each combination is given.

Note that some unconverged solutions occur in the variation of inflow and outflow boundary conditions. In setting 3, the inflow characteristic method causes large numerical instabilities (wiggles) in the boundary layer. This can be understood, since the method is based on inviscid flow theory. In setting 4, fixing the outflow pressure results in wave reflections at the outflow boundary, causing an extremely slow convergence. In setting 7, the non-reflecting characteristic method at the outflow results is an underspecification of variables.

Next, we compare the converged solutions in Fig. 2-2a, Fig. 2-2b and Fig. 2-2c. We use the wall pressure to examine the performance of the inflow and outflow
boundary conditions, and the pressure across the boundary layer, at a streamwise position corresponding to half of the domain length, to examine the quality of the upper boundary conditions. In the comparison of the methods for the inflow bound-

![Figure 2-2](image)

**Figure 2-2:** a) Wall pressure by various inflow techniques: extrapolation method (1, solid) and hybrid method (2, dashed). b) Wall pressure by various outflow techniques: partly non-reflecting hybrid method (5, dashed-dotted), perfectly non-reflecting hybrid method (6, dashed) and partly non-reflecting characteristic method (1, solid). c) Pressure distribution in the normal direction by various upper boundary techniques: extrapolation method (8, dashed-dotted), partly non-reflecting (pressure prescription) characteristic method (9, dashed) and perfectly non-reflecting characteristic method (1, solid).

ary (Fig. 2-2a), the hybrid technique produces a large numerical instability, which discourages the use of the characteristic principle in the viscous part of the inflow boundary. For the outflow boundary (Fig. 2-2b), the non-reflecting condition of the hybrid method underspecifies the outflow variables and results in large wiggles. On the contrary, the hybrid and the characteristic method with the incorporation of the
partly non-reflecting conditions yield good results. Experimenting with supersonic flows, however, we confirm that the perfectly non-reflecting outflow boundary conditions also yield smooth, converged results, as the underspecification appears only in the subsonic part near the wall. In the comparison of the upper boundary conditions (Fig. 2-2c), all the methods give satisfactory results. Along the upper boundary, where the flow is inviscid, the characteristic methods appear to advantage.

In conclusion, based on the above numerical comparison, we give our preference to the following combination. We use for the subsonic inflow boundary the extrapolation method and for the subsonic outflow boundary the partly non-reflecting of the quasi two-dimensional characteristic method. Although the partly non-reflecting of the one-dimensional characteristic method also gives a good result for the outflow boundary condition, we prefer the quasi two-dimensional method due to the multidimensionality character of the latter. We expect, that for flows where the velocity components tangential to the boundary are of comparable magnitude to the velocity component normal to the boundary, the quasi two-dimensional method performs better than the one-dimensional method. For the subsonic upper boundary we also use the quasi two-dimensional characteristic method, which is more flexible than the extrapolation method. The choice between the perfectly non-reflecting, pressure prescription (partly non-reflecting) and suction/blowing depends on the problem under consideration. The extrapolation method can be used at the upper boundary if the physical boundary condition can be fully specified by the external flow (see the case of a steady shock in Chapter 6).
In this chapter, we present the numerical methods developed for the spatial simu-
lations using orthogonal grids. The ingredients of the method are: high order finite
difference central and upwind schemes, second order explicit temporal integration,
and a buffer-domain technique. A highly accurate discretization method is necessary
in order to efficiently perform simulations in the turbulent regime, which demands
an enormous computational effort, while a buffer domain technique is essential to
damp wave reflections at the outflow boundary due to strong instationarities.

3.1 Spatial discretization and temporal integration

We begin with the description of the spatial discretization for the Navier-Stokes
equations, consisting of a second and a fourth order central scheme, and a third
order upwind scheme. Specifically, the conservation equations can be written in the
following form:

$$\partial_t U + \partial_j f_j(U) = 0,$$

(3-1)

where the vector $U$ represents the conserved variables $(\rho, \rho u_i, e)^T$ and $f_j$ the flux
in the $x_j$ direction, consisting of the inviscid and the viscous contributions. It is
important that dissipation and dispersion errors are kept to a minimum. Best suited
for this purpose are, for instance, spectral methods [28], compact finite difference [48]
and high order central difference approximations. In addition, it is also important
that the discretization can be written as close as possible to a conservative form. In
terms of accuracy and efficiency, spectral methods perform the best but is restricted
to orthogonal grids while compact finite difference is slightly better than high order
central difference but is still not flexible for non-uniform grids. In view of the
conservativity and the extension of the method to a general geometry in the future,
we favor a finite difference approach which is much more flexible than the other
methods. In this section two spatial discretization methods are described which are
respectively second order and fourth order accurate on smooth grids.

The second order accurate discretization used here is a finite volume method,
which is conservative, and can readily be formulated for an orthogonal non-uniform
The discretization for the inviscid terms is the cell vertex trapezoidal rule which is a weighted second order central difference. In vertex \((i,j)\) the derivative of a variable \(f\) with respect to e.g. \(x_1\) is defined as

\[
(\partial_1 f)_{i,j} = \frac{s_{i+1,j} - s_{i-1,j}}{x_{1(i+1)} - x_{1(i-1)}} \tag{3-2}
\]

with \(s_{i,j} = \frac{1}{2} \left( \frac{x_{2(j)} - x_{2(j-1)}}{x_{2(j+1)} - x_{2(j-1)}} (f_{i,j} + f_{i,j-1}) + \frac{x_{2(j+1)} - x_{2(j)}}{x_{2(j+1)} - x_{2(j-1)}} (f_{i,j+1} + f_{i,j}) \right). \tag{3-3}

The viscous terms contain second order derivatives which are approximated by the consecutive application of two first order numerical differentiation methods. In centre \((i,j)\) the discretization of \(\partial_{1}^2 f\) has the form

\[
(\partial_{1}^2 f)_{i,j} = 2 \frac{s_{i+\frac{1}{2},j} - s_{i-\frac{1}{2},j}}{x_{1(i+1)} - x_{1(i-1)}} \tag{3-4}
\]

with \(s_{i+\frac{1}{2},j} = \frac{1}{2} \left( (\partial_{1}^m f)_{i+\frac{1}{2},j-\frac{1}{2}} + (\partial_{1}^m f)_{i+\frac{1}{2},j+\frac{1}{2}} \right). \tag{3-3}

The 'inner' derivatives \(\partial_{1}^m f\) are calculated with the same discretization rule as (3-3) but now applied to control volumes centered around vertices \((i + \frac{1}{2}, j + \frac{1}{2})\): \(\partial_{1}^m f\)

\[
(\partial_{1}^m f)_{i+\frac{1}{2},j+\frac{1}{2}} = \frac{g_{i+1,j+\frac{1}{2}} - g_{i,j} + \frac{1}{2}}{x_{1(i+1)} - x_{1(i)}} \tag{3-5}
\]

with \(g_{i,j+\frac{1}{2}} = \frac{1}{2} (f_{i,j} + f_{i,j+1}). \tag{3-4}

The scheme for the viscous terms prevents odd-even decoupling. Each of the above finite difference schemes for first order spatial derivatives in \((i,j)\) can be written in the following form:

\[
(\partial_1 f)_{i,j} = \sum_{k=-n}^{k=n} w_k g_{i+k,j} \tag{3-5}
\]

where \(n = 1\) for the above three-point second order approximation. We call the \(w_k\) differencing weights and the \(h_k\) averaging weights.

A fourth order scheme for the calculation of the inviscid terms can be formulated in the form (3-5) using a five-point molecule \((n = 2)\). The weights can be computed by imposing certain requirements on the discretization scheme. For instance, the requirement that the difference scheme is exact for a given basis function \(m(x_1)\) yields the linear relation

\[
\sum_{k=-2}^{k=2} w_k m(x_{1(i+k)}) = \frac{\partial m}{\partial x_1} (x_{1(i)}) \tag{3-6}
\]
Numerical methods

for the differencing weights and the linear relation

$$\sum_{k=-2}^{k=2} h_k m(x_{1(i+k)}) = m(x_{1(i)})$$ (3-7)

for the averaging weights, respectively. The basis functions used here are polynomials in $x_1$. This leads to

$$(\partial_1 f)_{i,j} = \frac{4}{3} \left( \frac{s_{i+1,j} - s_{i-1,j}}{x_{1(i+1)} - x_{1(i-1)}} \right) - \frac{1}{3} \left( \frac{s_{i+2,j} - s_{i-2,j}}{x_{1(i+2)} - x_{1(i-2)}} \right)$$ (3-8)

with $s_{i,j} = \sum_{k=-2}^{k=2} h_k f_{i,j+k}$, (3-9)

where the averaging weights $h_k$ read:

$$h_{-2} = -\frac{1}{8} \frac{x_{2(j)} - x_{2(j-2)}}{x_{2(j+2)} - x_{2(j-2)}}$$

$$h_{-1} = \frac{1}{2} \frac{x_{2(j)} - x_{2(j-1)}}{x_{2(j+1)} - x_{2(j-1)}}$$

$$h_0 = \frac{5}{8}$$

$$h_1 = \frac{1}{2} \frac{x_{2(j+1)} - x_{2(j)}}{x_{2(j+1)} - x_{2(j-1)}}$$

$$h_2 = -\frac{1}{8} \frac{x_{2(j+2)} - x_{2(j)}}{x_{2(j+2)} - x_{2(j-2)}}$$

It should be noticed that differentiation with respect to $x_1$ is carried out using differencing weights in the $x_1$ direction and averaging weights in the $x_2$ direction. This scheme is fourth order accurate for smooth grids [87].

The weights for the second order derivatives of the viscous terms are computed analogously by employing four-point molecules to approximate each of the derivatives $\partial_1^m f$ and $\partial_2^m f$. The ‘inner’ derivative $\partial_1^m f$ is calculated in cell-centres $(i + \frac{1}{2}, j + \frac{1}{2})$ making use of the information on vertices $[i - 1 \ldots i + 2, j - 1 \ldots j + 2]$, whereas the second derivative $\partial_2^2 f$ is computed in vertices $(i, j)$ using the derivatives $\partial_1^m f$ in cell centres $[i - \frac{3}{2} \ldots i + \frac{3}{2}, j - \frac{3}{2} \ldots j + \frac{3}{2}]$. Based on the same grid resolution, the fourth order method requires approximately twice the computational effort of the second order method on a vector computer. The elaboration of the fourth order central scheme can be found in Appendix A.

In flows containing shock waves, we use a third order MUSCL (Monotonic Upstream Schemes for Conservation Laws) scheme for the inviscid terms of the governing equations. The viscous terms are discretized according to the fourth order central scheme. For convenience, we describe the method for the one-dimensional case. Multi-dimensional applications follow accordingly. For more detailed description of the method one is referred to [47]. We consider the following equation

$$\partial_t U + \partial_1 g = 0,$$ (3-11)

where $g$ is the inviscid part of the flux $f$ in (3-1) in the $x_1$ direction. The above equation can be discretized as follows

$$\Delta_t \partial_t U + g_{i+\frac{1}{2}} - g_{i-\frac{1}{2}} = 0,$$ (3-12)
where $\Delta_i$ is the grid spacing at point $i$. The flux $g$ consists of convective and dissipative terms

$$g_{i+\frac{1}{2}} = C_{i+\frac{1}{2}} - D_{i+\frac{1}{2}}, \quad \text{(3-13)}$$

with

$$C_{i+\frac{1}{2}} = \frac{g(U_l) + g(U_r)}{2}, \quad \text{(3-14)}$$

and

$$D_{i+\frac{1}{2}} = \frac{1}{2} |A(U_l, U_r)| (U_r - U_l), \quad \text{(3-15)}$$

where $U_l$ and $U_r$ are left and right state vectors which are obtained from interpolations of the state vectors at grid points. The interpolation determines the accuracy order of the method. $A(U_l, U_r)$ is an approximation of the flux Jacobian matrix of $g$ in (3-11) at the cell face $i + \frac{1}{2}$. The absolute value of the flux Jacobian matrix approximation is defined as $|A| = R|\Lambda|L$, where $R$ and $L$ are right and left eigenvector matrices of $A$ and $|\Lambda|$ a diagonal matrix containing the absolute values of the eigenvalues of $A$. The interpolation of the state vectors is as follows

$$U_l = U_i + \frac{1}{4} \left[ (1 - \eta) \text{Lim}(\Delta U_{i-\frac{1}{2}}, \omega \Delta U_{i+\frac{1}{2}}) + (1 + \eta) \text{Lim}(\Delta U_{i+\frac{1}{2}}, \omega \Delta U_{i-\frac{1}{2}}) \right], \quad \text{(3-16)}$$

$$U_r = U_{i+1} - \frac{1}{4} \left[ (1 - \eta) \text{Lim}(\Delta U_{i+\frac{1}{2}}, \omega \Delta U_{i+\frac{3}{2}}) + (1 + \eta) \text{Lim}(\Delta U_{i+\frac{3}{2}}, \omega \Delta U_{i+\frac{1}{2}}) \right], \quad \text{(3-17)}$$

with $\Delta U_{i+\frac{1}{2},j} = (U_{i+1,j} - U_{i,j})$ and the limiter is defined as

$$\text{Lim}(a, b) = \text{minmod}(a, b) = \text{median}(a, 0, b). \quad \text{(3-18)}$$

The parameters $\eta$ and $\omega$ must obey

$$-1 \leq \eta \leq 1, \quad 1 \leq \omega \leq \frac{3}{1-\eta} \quad \text{(3-19)}$$

to ensure monotonicity [92]. For the special choice of $\eta = \frac{1}{3}$, the scheme is third order accurate in smooth regions. At extrema in numerical solutions, the scheme is first order accurate as the limiter function becomes zero at these points. A more specific discussion of this scheme can be found in [47].

The discretization near the boundaries is performed in the same manner as in the interior domain by creating dummy points outside the boundaries. This is sketched in Fig. 3-1 where the edges contain the dummy points. Dummy variables are obtained through an extrapolation of the interior variables to the dummy points. First, the extrapolation to the upper edge and the wall edge is carried out. The extrapolation to the inflow edge and the outflow edge then follows. The extrapolation to the upper edge is linear, while the extrapolation to the inflow edge and the outflow edge is third order. For the extrapolation to the wall edge, the velocities
are considered as odd functions, whereas the density and energy are considered as even functions. This is to facilitate the application of the adiabatic wall condition, \( \frac{\partial^2 T}{\partial x^2} = 0 \). For an isothermal wall, the wall temperature is fixed and we use the approximate condition \( \frac{\partial^2 p}{\partial x^2} = 0 \) at the wall, which is discretized in a second order accurate way. In the three dimensional case, periodic conditions are applied in the spanwise direction.

![Figure 3-1: Edges adjacent to the interior domain, containing dummy points.](image)

After the spatial discretization is performed, the system of partial differential equations (3-1) can be combined into a system of ordinary differential equations for the discrete state vector \( u \)

\[
\frac{du}{dt} = F(u),
\]

where \( F \) is the discrete flux vector. This system of differential equations is solved using a four-stage compact-storage Runge-Kutta method. It performs within one time step \( \Delta t \)

\[
u^{(n)} = u^{(0)} + \alpha_n \Delta t F(u^{(n-1)}), \quad (n = 1, 2, 3, 4)
\]

with \( u^{(0)} = u(t) \) and \( u(t + \Delta t) = u^{(4)} \). With the coefficients \( \alpha_1 = 1/4, \alpha_2 = 1/3, \alpha_3 = 1/2 \) and \( \alpha_4 = 1 \) this yields a second order accurate time integration method ([37]).

### 3.2 Buffer domain technique

In the vicinity of the outflow boundary a buffer domain is optionally employed. The objective of the buffer-region is to prevent reflection of the fluctuations which have built up inside the computational domain as a consequence of physical instabilities in the flow. Common ways to achieve this goal are a considerable increase of the viscosity ([51]), a gradual change of the governing equations into a parabolic system ([51], [60] and [77]) and removing the disturbance component from the convective velocity ([60] and [77]). The last method is originally applied by Streett and Macaraeg ([77]) for incompressible flow.
Experiments indicate that the larger the viscosity increase the more effectively
the buffer damps the fluctuations. However, for calculations with explicit time stepp-
ing, the increase of viscosity yields a considerable limitation in the allowable time
step. This viscous limitation on the time step becomes rapidly more restrictive than
the inviscid limitation. Consequently, this viscous time step restriction deteriorates
the overall efficiency of the numerical method. The technique of applying the parab-
olizing procedure has only a gradual effect and consequently requires the use of quite
extended, and hence costly, buffer domains in order to be effective.

We propose a more efficient method in which the disturbances of all the solution
components are gradually reduced to zero within the buffer domain by directly
multiplying the disturbances with an appropriate damping function. This gives a
better result than the removal of disturbances only from the convective velocity,
even without parabolizing the governing equations. This method requires only a
short buffer domain to damp wave reflections at the outflow boundary, as will be
shown in the computational results. In this direct approach we need a reference
flow, relative to which the disturbances can be damped within the buffer domain.
The reference profile used here is the initial base flow in the case of parallel flow and
a time-averaged flow in the case of non-parallel flow, which is further described in
Section 5.3. The amplitudes of the Tollmien-Schlichting waves are gradually reduced
by multiplying the disturbances of all components with a damping function which
gradually decreases from 1 to 0. Hence, near the boundary all fluctuations are
effectively removed from the signal and the flow is relaminarized. This approach
can be described in the following formula:

\[ U = U_{ref} + \zeta(x_1)(\hat{U} - U_{ref}), \quad (3-22) \]

where \( U = (\rho, \rho u_i, e)^T \), \( U_{ref} \) is the similarity solution to the boundary-layer equa-
tions, \( \hat{U} \) is the solution calculated without applying the buffer treatment and \( \zeta \) is
a damping function. The damping given in (3-22) is applied in every stage of the
Runge-Kutta scheme. In the damping function we use a buffer domain coordinate
\( x_b \) which is defined as:

\[ x_b = \frac{x_1 - x_s}{x_e - x_s}, \]

where \( x_s \) and \( x_e \) are the values of \( x_1 \) at the beginning and the end of the buffer
domain respectively. Thus \( x_b \) ranges from 0 to 1 marking respectively the beginning
and the end of the buffer domain. The damping function \( \zeta \) is required to satisfy the
following constraints:

\[ \zeta(0) = 1, \]

\[ \frac{\partial \zeta}{\partial x_b}(0) = 0, \]

\[ \frac{\partial \zeta}{\partial x_b}(x_b) \leq 0 \quad \text{if} \quad x_b > 0, \quad (3-23) \]

\[ \zeta(1) = 0. \]
The first constraint lets disturbances enter the buffer domain at the original level of amplitude, the second constraint prevents a discontinuity at the beginning of the buffer domain, the third constraint forces $\zeta$ to decrease monotonically and the fourth constraint guarantees a zero disturbance amplitude at the outflow boundary. The reduction of disturbances is carried out in every stage and hence has a cumulative effect. Already reduced disturbances at a certain time will be multiplied again by the damping function at the next stage and so on. Hence, the amplitude of the disturbances decreases more rapidly than the decrease of the damping function. However, this successive reduction of the amplitude will not result in a sudden suppression of the amplitude of the disturbances. Numerical experiments show that a balance arises between the growth of disturbances fed by incoming disturbances from upstream the buffer domain and the damping of the disturbances within the buffer domain. We next proceed with specifying the damping function $\zeta$ in a few steps. The simplest damping function which satisfies the above four constraints, $\zeta(x_b) = 1 - x_b^2$, results in a too abrupt decrease of the disturbance amplitudes. To achieve a more gradual decrease, we use

$$\zeta = (1 - C_1 x_b^2)(1 - \frac{1 - e^{C_2 x_b^2}}{1 - e^{C_2}}) \quad 0 \leq C_1 < 1, \quad C_2 > 0.$$  \hfill (3-24)

By varying $C_1$ and $C_2$ different shapes of the damping function can be obtained. As an illustration, the shape of the damping function for various values of $C_1$ and $C_2$ is depicted in Fig. 3-2 and Fig. 3-3. By selecting appropriate values of $C_1$ and $C_2$ we can control the reduction rate of the disturbances. Numerical experiments lead to $0 \leq C_1 \leq 0.1$ and $10 \leq C_2 \leq 20$ as appropriate. Notice that $C_1 = 0$ is allowed, since the second term in (3-24) itself satisfies all four constraints (3-23).

As a final step in the specification of the buffer function we make it insensitive to the mesh size. We note that the result of the buffer procedure for a certain flow configuration depends on the number of time steps per disturbance period, since the buffer function is applied after every Runge-Kutta stage. Hence, when we use a higher grid density, the buffer procedure is applied more frequently. Besides, the number of time steps per period depends also on the flow conditions ($M_\infty$, $Re$, etc). In order to make the buffer procedure independent of the number of time steps per disturbance period, we employ a damping function $\tilde{\zeta}$ given by

$$\tilde{\zeta} = \tilde{\zeta} \tilde{C}_3 / N,$$  \hfill (3-25)

where $\tilde{C}_3$ is a tuning parameter and $N$ is the number of time steps per disturbance period, which is unknown prior to the simulation. $N$ can be approximated by $N = T / \Delta t$, where $T$ denotes one period of the inflow perturbation and $\Delta t$ the allowable time step. Since $T = 2\pi / \omega$, where $\omega$ is the circular frequency of the disturbance, this leads to

$$\tilde{\zeta} = \zeta C_3 \omega \Delta t,$$  \hfill (3-26)

where $\zeta$ is given by (3-24) and $C_3$ is a tuning parameter, which is kept constant throughout the simulations. The tuning of $C_3$ is conducted in Chapter 4.
Figure 3-2: Damping functions within the buffer domain using various values of $C_2$ at $C_1 = 0$.

Figure 3-3: Damping functions within the buffer domain using various values of $C_1$ at $C_2 = 10$.

Our numerical tests have shown that this direct approach is substantially more efficient than the approach of increasing viscosity with respect to both computational effort and reflection properties. The validation of the buffer domain approach including the calibration of the tuning parameters and the investigation of its performance for various buffer domain lengths and various lengths of the physical domain is presented in Chapter 4. We have applied this buffer domain technique to more complex flows, such as separated flows (see Chapter 5) and unsteady shock boundary layer interaction, and obtained satisfactory results. It should be noted that the reference flow $U_{ref}$ depends on the flow under consideration, whereas the damping function is generally applicable.
In this chapter we present simulations of a spatially evolving boundary-layer over a flat plate under zero pressure gradient. The simulations are performed to illustrate the accuracy and efficiency of the numerical approach followed. The computational domain has the form of a rectangle (Fig. 1-5). The domain is bounded by a solid wall at the lower side, while the other sides are open boundaries treated as artificial boundaries, as discussed in Chapter 2. We focus on two aspects, i.e. the spatial discretization method and the treatment of the artificial boundaries, primarily the outflow boundary. If not explicitly mentioned, for convenient, by the numerical methods of spatial DNS in the following we mean the numerical discretization as well as the numerical treatment of all the boundaries.

Specifically we compare the second and the fourth order spatial discretization methods which are described in Chapter 3, using small perturbation stability theories (linear stability theory and linear parabolized stability equations) as references. Regarding the artificial boundaries, our experiences with the various boundary treatments resulted in some preferences regarding the choice of the boundary treatment (Chapter 2). For the inflow boundary, an extrapolation method appears preferable whereas quasi two-dimensional characteristic methods appear more suitable at the upper and the outflow boundary. These methods are employed for the calculations in this and next chapters. Since in the case of zero pressure gradient the upper boundary is an artificial boundary, in contrast to a non-zero pressure gradient boundary, it is more convenient to use the term freestream or farfield boundary instead of upper boundary. Moreover, the buffer domain technique presented in Chapter 3 is extensively validated in this chapter.

The simulations are performed at Mach number 0.5 and 4.5. In the supersonic flow we consider parallel base flow with small disturbances, while in the subsonic flow we study both parallel and non-parallel base flows with small as well as large disturbances. This diverse set of flow conditions results in suitable testcases with which the performance of the discretization schemes and the artificial boundary treatments can systematically be examined. This set of testcases corresponds to
simulations which are successively more critical to the performance of the developed

techniques. A subsonic flow simulation for instance is more critical for the per-
formance of the outflow boundary treatments than a supersonic flow due to larger
upstream influences. Furthermore, large disturbances form a more difficult test for
the outflow treatments and the discretization scheme. Specifically, for the validation
the parallel flow results are directly compared with linear stability theory predic-
tions. On the other hand, results of non-parallel flow simulations are compared with
the solution of linear parabolized stability equations. In particular we pay attention
to a correct prediction of the growth rates which forms a very critical test. Nonlinear
effects in the latter flow become appreciable when we introduce large disturbances
at the inflow boundary.

The chapter is organized as follows. In section 1 we describe the linear stability
theory and parabolized stability equations. In Section 2 and Section 3 results on
parallel and non-parallel flow simulations are presented. We summarize our findings
in Section 4.

4.1 Small perturbation stability theories

The behavior of small perturbations in a laminar boundary layer flow can be de-
scribed by small perturbation stability theories. The solutions of these theories are
valid in the linear regime of the flow, where the perturbations are small compared to
the mean flow. Hence DNS can be validated by these theories in the linear regime.
Two linear perturbation theories are considered to serve as references in the DNS
validation: linear stability theory (LST) and (linear) parabolized stability equations
(PSE). Both theories are similar in that they consider Fourier modes as basic ele-
ments of the perturbations. The difference is that within the linearity restriction,
the first is designed for an exact description of small perturbations in a parallel flow,
whereas the last takes the non-parallelism in the basic flow up to a certain order
of approximation into account. Moreover, these linear theories can be used for the
calculation of $e^n$ factors, a commonly used method by Smith ([69]) and Van Ingen
([36]) for the prediction of transition from laminar to turbulent flow. Mathemati-
cally, we solve an eigenvalue problem for the LST and parabolic partial differential
equations for the PSE. Brief descriptions of the two theories are presented in the
following subsections. More specific information regarding LST can be found in [54],
and on PSE in [34].

4.1.1 Linear stability theory

For the outline of the linear stability theory we depart from the Navier-Stokes equa-
tions (2-1)-(2-3), where the quantities are made dimensionless by the corresponding
freestream values. In a laminar flow containing unsteady small perturbations, the
instantaneous density, pressure, temperature, velocities and the dynamic viscosity
can be presented as the sum of a base flow and a fluctuating quantity,

\[
\rho = \tilde{\rho} + \bar{\rho}, \quad p = \tilde{\rho} + \bar{\rho}, \quad T = \tilde{T} + \bar{T}, \\
\bar{u}_1 = \bar{U}_1 + \tilde{u}_1, \quad \bar{u}_2 = \bar{U}_2 + \tilde{u}_2, \quad \bar{u}_3 = \bar{U}_3 + \tilde{u}_3, \\
\mu = \bar{\rho} + \tilde{\mu},
\]

where the bars denote the base flow which obeys the Navier-Stokes equations and the tildes denote the perturbations. We substitute these into the governing equations (2-1)-(2-3) and neglect the terms containing products of perturbations. Subtracting the equations for the base flow from this, we can write the resulting linearized perturbation equations in the following form

\[
P\tilde{q}_t + Q\tilde{q} + R(\partial_t \tilde{q}) + S(\partial_{ij} \tilde{q}) = 0,
\]

with \( \tilde{q} = [\tilde{\rho}, \tilde{u}_1, \tilde{u}_2, \tilde{u}_3, \tilde{T}]^T \) and the matrix \( P, Q, R \) and \( S \) have base flow quantities as entries. Note that instead of using density and energy, we employ pressure and temperature, which is more convenient for the perturbation equations. The substitution is easily performed by means of the definition of the total energy density (2-4) and the equation of state (2-8). Moreover, the viscosity perturbation has been expressed in terms of the temperature perturbation,

\[
\tilde{\mu} = \frac{d\mu}{dT} \tilde{T},
\]

where \( d\mu/dT \) follows from Sutherland’s law (2-6). We consider the stability of locally parallel, two-dimensional, compressible boundary layer flow. This implies that in the system matrices \( P, Q, R \) and \( S \),

\[
\bar{U}_1 = \bar{U}(x_2), \quad \bar{U}_2 = 0, \quad \bar{U}_3 = 0, \\
\bar{p} = \bar{p}(x_2), \quad \bar{T} = \bar{T}(x_2).
\]

We assume that the pressure, velocity and temperature fluctuations are presented by a harmonic wave of the form

\[
(\bar{\rho}, \bar{u}, \bar{v}, \bar{w}, \bar{T}) = [\bar{\rho}(x_2), \bar{u}(x_2), \bar{v}(x_2), \bar{w}(x_2), \bar{T}(x_2)]e^{i(\alpha x_1 + \beta x_3 - \omega t)},
\]

where \( \alpha \) and \( \beta \) are the wavenumbers in \( x_1 \) and \( x_3 \) direction and \( \omega \) is the circular frequency which, in general, are all complex. In temporal stability theory, \( \alpha \) and \( \beta \) are real and \( \omega \) is complex, whereas in spatial stability theory the reverse is true. For the temporal setting, \( \omega_i \equiv \Im(\omega) \) represents the growth rate of the harmonic wave, whereas for the spatial setting, \( \alpha_i \equiv \Im(\alpha) \) and \( \beta_i \equiv \Im(\beta) \) represent the growth rate in the streamwise and the spanwise direction, respectively.

Substituting (4-5) in the perturbation equations (4-1), we can write the result as the following system of equations

\[
\omega \varphi + (AD^2 + BD + C)\varphi = 0,
\]

(4-6)
where $\varphi = [\tilde{p}, \tilde{u}, \tilde{v}, \tilde{w}, \tilde{T}]^T$ and $D \equiv d/dx_2$. $A$, $B$ and $C$ are $5 \times 5$ matrices which depend on the base flow and the wave numbers $\alpha$ and $\beta$. The boundary conditions for (4-6) are

$$\tilde{u} = \tilde{v} = \tilde{w} = \tilde{T} = 0, \quad \frac{\partial \tilde{p}}{\partial x_2} = \zeta_0, \quad \text{for } x_2 = 0, \quad (4-7)$$

$$\tilde{u}, \tilde{v}, \tilde{w}, \tilde{T} \to 0, \quad \frac{\partial \tilde{p}}{\partial x_2} = \zeta_\infty, \quad \text{for } x_2 \to \infty, \quad (4-8)$$

where $\zeta_0$ and $\zeta_\infty$ follow from the evaluation of the normal momentum equation at the two boundaries. For an adiabatic wall, the use of $\tilde{T} = 0$ at $x_2 = 0$ is justified by the assumption that the temperature fluctuations will not penetrate into the wall due to the thermal inertia of the solid body.

For temporal stability theory we fix the wave numbers and solve the equations for $\omega$. In that case, equations (4-6)-(4-8) represent an eigenvalue problem resulting in the complex dispersion relation

$$\omega = \omega(\alpha, \beta). \quad (4-9)$$

The above system of equations is discretized with a second order central finite difference method. After the discretization, the compressible stability equations (4-6) along with the boundary conditions (4-7) and (4-8) are formulated as a matrix eigenvalue problem,

$$A\varphi = \omega B\varphi, \quad (4-10)$$

where $\omega$ is the eigenvalue and $\varphi$ is the discrete eigenfunction. It can be shown that the rank of the system matrices $A$ and $B$ is $5N-4$, with $N+1$ is the number of grid points in the $x_2$ direction including the boundary points.

From this point, two classes of numerical method can be followed for computing the eigenvalues: global and local methods. Using a global method we get all the eigenvalues of the system matrix, in exchange for a quite high computational effort. In a local method, a guess for an eigenvalue is required. Only the eigenvalues which happen to lie in the neighborhood of the guessed eigenvalue are computed. The local method is much cheaper than the global method, but an accurate guess for the eigenvalue should be provided in order to get a converged solution. We use both methods in our calculations. We apply the standard QZ algorithm ([27]) as a global method and perform the calculation on a coarse grid. From the resulting eigenvalues, we select an appropriate one as the guessed eigenvalue for a local method. Then we employ the inverse Rayleigh iteration in the local method and perform the calculation on a finer grid. The local calculation method is repeated until the eigenvalue corresponding to the desired grid is obtained.

Making use of the calculation procedure for the temporal stability theory, the spatial stability calculations form a rootfinding problem. More specifically, in the spatial stability theory we fix $\omega$, which is now real, and seek the corresponding complex wave numbers, $\alpha$ and $\beta$. Through imposing the angle of the oblique waves,
which is the angle between $\alpha$ and $\beta$, we relate the spanwise wave number, $\beta_r \equiv \Re(\beta)$, to the streamwise wave number, $\alpha_r \equiv \Re(\alpha)$. $\beta_r$ which represents the growth rate in the spanwise direction is set to zero in accordance with the assumption of periodicity in the spanwise direction. The task is now to find a complex $\alpha$ which corresponds to the desired frequency, $\bar{\omega}$. The procedure is as follows. We take a guess for $\alpha$ and compute the eigenvalue $\omega$ in the frame of temporal stability theory. Based on the difference $\Delta \omega = |\omega - \bar{\omega}|$, we provide a better guess for $\alpha$. This iteration proceeds until $\Delta \omega$ is sufficiently small to satisfy a stopping criterion. By solving this rootfinding problem, we obtain the wave numbers, $\alpha$ and $\beta$, corresponding to the selected frequency $\omega$.

![Complex eigenfunction of a 2D unstable mode for compressible Blasius base flow at $Re = 875$, $M = 0.5$ and $\omega = 0.1$; a) pressure, b) streamwise velocity, c) normal velocity and d) temperature.](image)

**Figure 4-1:** Complex eigenfunction of a 2D unstable mode for compressible Blasius base flow at $Re = 875$, $M = 0.5$ and $\omega = 0.1$; a) pressure, b) streamwise velocity, c) normal velocity and d) temperature.

As an example, we show in Fig. 4-1a - 4-1d eigenfunctions of a two-dimensional harmonic wave ($\beta = 0$) resulting from the spatial stability calculation for com-
pressible Blasius base flow. These eigenfunctions correspond to the case of subsonic parallel base flow, with \( M = 0.5, Re = 875 \) and \( \omega = 0.1 \), which is presented in Section 4.2.2.

### 4.1.2 Linear parabolized stability equations

The parabolized stability equations (PSE) are designed to remove the restriction of the linear stability theory to locally parallel base-flow. The PSE account for the non-parallel base-flow, provided that the growth rate, the wave numbers and the wave functions vary slowly in the streamwise direction. Moreover, PSE is restricted to convectively unstable flows. The parabolic character of the equations disallows upstream influences. PSE can be devided into linearized and full PSE. The first only accounts for the base-flow nonparallelism, whereas the second accounts for nonparallelism and nonlinearity simultaneously. Here, we restrict to linearized PSE.

For the description of linearized PSE (in the following we just call it PSE), we follow a similar path as that of the linear stability theory. The difference is that we drop the parallel base-flow restriction (4-3), (4-4), and assume that instead of (4-5) the perturbations are spatially harmonic waves whose growth rate, streamwise wave number and wave function depend weakly on \( x_1 \). More specifically, the perturbations have the form

\[
\bar{q}(x_1, x_2, x_3, t) = q(x_1, x_2)\psi(x_1, x_3, t), 
\]

with

\[
\psi = \exp \left[ \int_{x_1}^{x_1'} \alpha(s) ds + \beta x_3 - \omega t \right],
\]

where \( \bar{q} \) is the perturbation vector as defined previously and \( q \) is the corresponding wave function. The derivatives of the perturbation with respect to the streamwise direction are

\[
\begin{align*}
\partial_{x_1} \bar{q} &= (i\alpha q + \partial_1 q)\psi, \\
\partial_{x_1}^2 \bar{q} &= (-\alpha^2 q + 2i\alpha(\partial_1 q) + \partial_1^2 q + i(\partial_1 \alpha)q)\psi.
\end{align*}
\]

Substituting the components of (4-11) into the linearized perturbation equations (4-1), we can rewrite the resulting equations in the following compact form

\[
Kq + (\partial_1 \alpha)Lq + M(\partial_1 q) + N(\partial_1^2 q) = 0,
\]

where the operators \( K, L, M \) and \( N \) contain only derivatives with respect to \( x_2 \). The PSE approximation assumes that the streamwise variation of \( q \) and \( \alpha \) is weak. Hence, the derivatives \( \partial_1 q \) and \( \partial_1 \alpha \) are sufficiently small that their higher derivatives with respect to \( x_1 \) are negligible. Therefore we neglect \( \partial_{x_1}^2 q \), reducing (4-15) to

\[
Kq + (\partial_1 \alpha)Lq + M(\partial_1 q) = 0.
\]
The removal of $\partial^2 \tilde{q}$ characterizes the parabolicity of the perturbation equations. The residual ellipticity of the equations is retained since $\partial^2 \tilde{q}$ is not zero. Hence, Tollmien-Schlichting waves can be simulated by PSE, in contrast to the boundary layer equations or parabolized Navier-Stokes equations, which fail to obtain TS waves, as $\partial^2 \tilde{q} = O(|\alpha|)$ is not negligible.

With the boundary conditions for the wave function given by (4-7) and (4-8), the problem is now to find $q$ and $\alpha$, which correspond to a desired frequency $\omega$. $\beta$ is related to $\alpha$ as in the LST. Due to the parabolic character of the equations, we can consider the problem as an initial-boundary value problem for $q$ and use a marching procedure to solve it. However, we need an additional condition for $\alpha$ to close the equations. This is provided by a normalization procedure. Specifically, the complex wave number $\alpha$ in the LST is given by the logarithmic derivative

$$-i \partial_1 (\ln \tilde{q}) = -i \frac{\partial_1 \tilde{q}}{\tilde{q}} = \alpha,$$  \hspace{1cm} (4-17)

But from (4-11) and (4-13),

$$-i \partial_1 (\ln \tilde{q}) = \alpha - i \frac{\partial_1 q}{q},$$  \hspace{1cm} (4-18)

The last term causes a dependency of $\alpha$ on $x_2$. To remove this dependency, we manipulate Eq. (4-18) according to

$$-i \int |q|^2 \partial_1 (\ln \tilde{q}) dx_2 = \alpha - i \frac{\int q^* \partial_1 q dx_2}{\int |q|^2 dx_2},$$  \hspace{1cm} (4-19)

where the asterisk denotes the complex conjugate. The integration is performed over the domain in $x_2$. Then we select

$$\int q^* (\partial_1 q) dx_2 = 0,$$  \hspace{1cm} (4-20)

or less stringent, by replacing $q$ by the perturbation velocity vector $v$,

$$\int v^* (\partial_1 v) dx_2 = 0,$$  \hspace{1cm} (4-21)

representing the normalization for $v$. The additional condition for $\alpha$ is now: we update $\alpha$ by the iteration

$$\alpha^{n+1} = \alpha^n - i \frac{\int (q^n)^* (\partial_1 q^n) dx_2}{\int |q^n|^2 dx_2},$$  \hspace{1cm} (4-22)

until (4-21) is satisfied. This condition for $\alpha$ together with (4-16) forms the closed system of parabolized stability equations. The update process for $\alpha$ is performed at every streamwise step in the marching procedure. The initial condition, i.e. the condition of $q$ and $\alpha$ at the inflow boundary, is provided by an LST calculation.
4.2 Parallel base flow

The simulations of parallel base flow with small perturbations are performed in the supersonic and the subsonic regime with Mach number 4.5 and 0.5, respectively, to study the effect of essentially different inflow and outflow characteristics on the flow.

4.2.1 Supersonic flow with small perturbations

First, it is important to verify the accuracy of the discretization scheme and the performance of inflow- and outflow boundary treatments with small disturbances. Therefore, a simulation of unsteady flow has been performed at \( M_\infty = 4.5 \) and \( T_\infty = 61.15K \) in which time dependent perturbations are added to a parallel base flow. This flow falls within a second-mode instability region (Mack mode [53]). Since a flow over a flat plate is physically non-parallel, forcing terms are introduced in the right hand side of the momentum and energy equations in the Navier-Stokes system to keep the flow parallel. Denoting the forcing term of the \( u_1 \) and \( u_2 \) momentum equations and the energy equation by \( F_1, F_2 \) and \( F_e \), respectively:

\[
\begin{align*}
F_1 &= -\partial_2 (\frac{\mu}{Re} \partial_2 \bar{u}_1) \\
F_2 &= 0 \\
F_e &= -\partial_2 [\bar{u}_1 \frac{\mu}{Re} \partial_2 \bar{u}_1 + \bar{q}_2],
\end{align*}
\]

where \( \bar{u}_1 \) and \( \bar{q}_2 \) are the streamwise velocity and the wall-normal heat flux of the mean initial field, respectively.

The present physical parameters have also been used by Guo et al. in [31]. Specifically, the reference Reynolds number is \( Re = 8000 \). The similarity solution at \( x_1 = 100 \) is used as the parallel base flow. This value of \( x_1 \) marks the position of the inflow boundary. Eigenfunction perturbations derived from compressible linear stability theory are added to the base flow defining the total initial flow. The perturbation waves, traveling in the streamwise direction can be described according to

\[
v = \epsilon \text{Real}(\psi(x_2) \exp[i(\alpha(x_1 - x_o) - \omega t)]),
\]

where \( \alpha \) is the wave number, \( \omega \) the circular frequency, \( \epsilon \) the initial perturbation amplitude \( x_o \) the value of \( x_1 \) at the inflow boundary (\( x_1 - x_o = 0 \) at the inflow boundary) and \( \psi \) the complex eigenfunction vector. \( v \) represents a component of perturbations \( (p', u'_1, u'_2, T') \). In the present case of spatial simulations \( \alpha \) is complex whereas \( \omega \) is real. Hence, if \( \text{Im}(\alpha) < 0 \) the perturbation-amplitude will grow exponentially in the streamwise direction.

For the circular frequency of the linear perturbations, we use \( \omega = 1.766 \). The compressible linear stability theory provides an eigenvalue,

\[
\alpha = \alpha_R + i\alpha_I = 1.94247 - i0.02503,
\]

which is an unstable mode. The value of \( \epsilon \) used here is \( 10^{-4} \). In this test case of parallel flow the real and imaginary parts of the eigenfunctions, \( \psi_R^v \) and \( \psi_I^v \), are...
only dependent on \( x_2 \). It can easily be derived that the amplitude of a disturbance component \( v_i \) grows in the \( x_1 \) direction according to the following expression:

\[
R_i = e^{-\alpha_1 x_1} \sqrt{(\psi_{Ri})^2 + (\psi_{Ii})^2}
\]  

and since \( \psi_R \) and \( \psi_I \) are constant in the \( x_1 \) direction, the growth of the amplitude is exponential in \( x_1 \). This does not hold when we consider non-parallel flows as will be illustrated in subsection 4.3.1.

The length of the computational domain is selected to be 8 Tollmien-Schlichting (T-S) wavelengths, of which the last is optionally used as buffer domain. The height of the computational domain is \( L_2 = 4 \). We first use the second order scheme and a grid with 192 \( \times \) 128 points in the streamwise and the normal direction, respectively. Each T-S wave is thus represented by 24 points. The grid is uniform in the \( x_1 \) direction and stretched in the \( x_2 \) direction according to a rational stretching function,

\[
x_2 = L_2 S_r y/(1 + S_r - y),
\]  

where \( S_r \) is a stretching parameter and \( 0 \leq y \leq 1 \), uniformly distributed. The grid becomes more uniform with increasing \( S_r \). An appropriate value of \( S_r \) in this testcase is 0.5, leading to the stretching ratio \( \Delta x_2 \max /\Delta x_2 \min = 8.8 \). The maximum aspect ratio of the grid is \( \Delta x_1 /\Delta x_2 = 12.87 \). The number of time steps per period, following from the stability requirement of the Runge-Kutta method, is about 750. The calculations are performed without buffer domain. The growth of the disturbance amplitude deviates from the LST result increasingly in the streamwise direction as shown on the top in Fig. 4-2 (in this figure, the lines corresponding to the different grids and schemes have been shifted relative to each other to show the differences more clearly). The phase of the T-S waves deviates from the LST result as well, as depicted in Fig. 4-3. The disturbance data is taken from the \( u_1' \) component at a location near the maximum amplitude. The equivalent amplitude growth and T-S waves calculated from other components of the solution are quite similar to those of \( u_1' \). Addressing the deviations to insufficient grid density, we increase the grid density in the \( x_1 \) direction up to 384 \( \times \) 128 grid points. Although the phase is now in better agreement with the LST result, this does not improve the amplitude growth. On the other hand, increasing the grid density in the \( x_2 \) direction (192 \( \times \) 256 grid points) yields a better amplitude growth but the deviation in the phase is about as large as with the original grid. From this knowledge it is apparent that the second order spatial discretization scheme requires a very fine grid, with at least 384 \( \times \) 256 grid points, to achieve acceptable accuracy. In order to yield more accurate results with a moderate grid density we employ the fourth order spatial discretization scheme as described in section 3. Using the original grid 192 \( \times \) 128, this scheme produces much better results than the second order scheme on finer grids. Both the amplitude growth and the T-S wave phase of the fourth order scheme are in excellent agreement with the LST results as shown in Fig. 4-2 (below) and Fig. 4-3, respectively. The number of time steps per period for the fourth order method on this coarse grid is around 1500. For a comparable accuracy, the second order method requires the grid
384 × 256, which has four times as many points as the grid 192 × 128. In combination with the fact that the fourth order method requires about twice the calculation time on a given grid than the second order method, it follows that the fourth order method needs half the computational effort to achieve the same accuracy.

![Figure 4-2](image1.png)

**Figure 4-2:** Comparison of the growth of disturbance amplitude, taken from $u'_1$ component, for the second order scheme on various grids and the fourth order scheme on the coarsest grid.

![Figure 4-3](image2.png)

**Figure 4-3:** The same comparison as in Fig. 4-2 for the development of TS waves in the streamwise direction.

Fig. 4-4a shows the relative deviation of the local growth rates from the LST spatial growth rate, $-\alpha_i$, using the fourth order method. This parameter is critical for the performance of the inflow and outflow boundary conditions. The growth rates are calculated from the first order derivative with respect to $x_1$ of the local amplitude growths using a central difference scheme. The maximum deviation of the growth
rates is about 2.5% over 87% of the domain. Strong oscillations originating from the outflow boundary conditions are restricted within the last 13% of the domain. Although we do not employ any buffer domain treatment in these calculations, the maximum deviation compares favorably with the results reported in [31]. Especially near the inflow boundary, the error is kept to a minimum thanks to the accurate numerical treatment near this boundary. This is confirmed by varying the degree of the extrapolation polynomials used at the boundaries from the default value 4. Lower degree polynomials yield larger errors near the inflow and outflow boundaries. On the other hand, increasing the polynomial degree to more than 5 can lead to numerical instabilities at the boundaries. Small oscillations which are appreciable along the second half of the domain are damped when the buffer domain treatment is employed (the buffer domain validation is discussed in the next section), as shown in Fig. 4-4b. However, small waves remain present along the second half of the domain. Increasing the resolution in the streamwise direction decreases the errors along the first half of the domain. Varying the length of the computational domain indicates that the small waves are residual standing waves originating from the interaction between the inflow and outflow boundaries. Excellent agreement between the fundamental wave for $u_1$ at a certain streamwise position and the corresponding LST result is shown in Fig. 4-5. The streamwise position is selected such that the comparison is made in the neighborhood of a local wave-extremum. Equivalent results are produced by considering other components.

From these results we learn that the fourth order spatial discretization requires less computational effort than the second order method for a comparable accuracy. Furthermore, the inflow and outflow boundary treatments perform well for high su-

Figure 4-4: a) Growth rate deviation in $u_1'$ component, without buffer domain, b) with buffer domain. The result with a finer grid in the $x_1$ direction is also presented.
personic flows. The outflow boundary exhibits only a small upstream influence and hence employing a buffer domain is not essential in order to damp wave reflections. The good performance of the outflow boundary conditions results apparently from the physics of the flow passing through the outflow boundary. In this flow application, most of the flow passing through the outflow boundary is supersonic and only a small part near the wall is subsonic. Therefore, the source of upstream influences is restricted near the wall. Flow simulations at a subsonic reference Mach number provide a more difficult test of the inflow/outflow boundary treatments and the buffer domain. This will be considered in the next sections.

\[ u_1' \]

\[ x_1 = 121.29 \]

Figure 4-5: Comparison of the disturbance profile of \( u_1' \) fundamental wave at \( x_1 = 121.29 \) between DNS and LST.

4.2.2 Subsonic flow with small perturbations

Similar calculations as in the previous section are performed at a reference Mach number of 0.5. The Reynolds number of the simulation is supercritical, equal to 875. The circular frequency \( \omega \) of the linear perturbations is 0.1. Compressible linear stability theory provides an eigenvalue,

\[ \alpha = \alpha_R + i\alpha_I = 0.2649 - i0.005246, \]  

which is an unstable mode. The corresponding eigenfunctions are shown in Fig. 4-1a - Fig. 4-1d as an illustration in the discussion of linear stability theory.

The length of the computational domain is equivalent to 7 T-S waves of which the last wavelength is optionally reserved for a buffer domain. The waves are represented on a grid with 32/wave \( \times \) 128 grid points in the streamwise and the normal direction, respectively. The height of the computational domain is 30. The grid
is similar as in the previous section with the stretching parameter $S_r = 0.2$ yielding a ratio $\Delta x_2_{max}/\Delta x_2_{min} = 34.42$. The maximum aspect ratio of the grid is $\Delta x_1/\Delta x_2 = 18.94$. Comparison of different spatial discretization schemes and the influence of a buffer domain on the results are depicted in Fig. 4-6 and Fig. 4-7 for $u_1$ and $u_2$ disturbances, respectively. In these figures, two groups of lines corresponding to whether a buffer domain is applied (B1) or not (B0) are shown. Calculations

![Figure 4-6](image1)

**Figure 4-6:** Comparison of the amplitude growth taken from the $u'_1$ between treatment without buffer domain (B0) and with buffer domain (B1) using the second order scheme and the fourth order scheme.

![Figure 4-7](image2)

**Figure 4-7:** The same comparison as in Fig. 4-6 for the amplitude growth of $u'_2$.

without buffer domain exhibit strong standing waves in the amplitude growths of disturbances for both the second order and the fourth order method. Reducing the time steps and increasing the grid density have no appreciable influence on the results, indicating that the observed waves are mainly due to the inflow-outflow
boundary coupling and not related to the accuracy of the time integration and the spatial discretization. The indication that the observed waves are standing waves is confirmed by varying the length of the computational domain, which results in a different shape of these waves. Comparison with the corresponding results of a supersonic flow in the previous section, shows that an entirely subsonic boundary-layer provides more wave reflections than a partly supersonic. Subsonic flows, contrary to supersonic, have a downstream domain of dependence. This implies that some information from outside the flow domain is required. In most applications, such as ours, this information can only be obtained by some approximations. The lack of exact information downstream the outflow boundary results in wave reflections. To damp these reflections the buffer domain treatment described in section 4 is applied. The standing waves are substantially suppressed due to the buffer treatment as illustrated in Fig. 4-6 and Fig. 4-7. The performance of the second order and the fourth order methods are comparable with respect to the growth of the disturbance amplitude, promoting the use of the second order method over the fourth order due to lower computational cost. However, a comparison of the T-S waves from both methods shows that the second order method results in larger phase errors than the fourth order method. The latter agrees excellently with the result from LST. Supported by this result, we use only the fourth order spatial discretization method in the next calculations. Moreover, the use of a buffer domain treatment is obviously essential for accurate results.

The calculations with buffer domain are performed with $C_1 = 0$ and $C_2 = 10$ in (3-24) and $C_3 = 330$ in (3-26). The latter value corresponds to $C_3 \omega \Delta t \approx 1$. This value of $C_3$ is kept the same for all calculations employing the buffer domain, since the results are quite insensitive to variations in $C_3$. We proceed with the calibration of $C_1$ and $C_2$. Much higher or lower values of $C_2$ deteriorate the performance of the buffer domain as illustrated by the amplitude growths depicted in Fig. 4-8. This can be explained as follows. A very low $C_2$ suppresses the disturbance amplitude directly after the start of the buffer domain which leads to larger upstream influences, whereas a very high $C_2$ reduces the amplitude only near the outflow boundary. A large increase of $C_1$ has the same effect as a decrease of $C_2$, as illustrated by the amplitude growths shown in Fig. 4-9. A small positive $C_1$, for example 0.01, results in a slight improvement in the growth rate, but as other results do not change, we use $C_1 = 0$ and $C_2 = 10$ in the case of small disturbances. It is, however, expected that the influence of the variation of the damping function parameters on the results become more appreciable if we consider large disturbances.

The performance of the buffer domain is further validated by varying the length of the buffer domain. Increasing this length to 2 TS waves yields a slight improvement in suppressing wave reflections. Extending the buffer domain to more than 2 TS waves gives no appreciable further improvement. Furthermore, we also varied the length of the physical domain to examine the upstream influence of the buffer domain. Although small standing waves observed in the amplitude growth are slightly changed by this variation, their amplitude is unchanged, which indicates that the upstream influence of the buffer domain is insignificant.
Finally, we compare the disturbance profile of the fundamental wave $u_1$ using the fourth order method with the LST result at a specific streamwise location and time. The streamwise location is selected such that the comparison is made in the neighborhood of a local wave-extremum. Excellent agreement in this comparison is shown in Fig. 4-10. As in the previous section, similar results are produced by other components. The buffer domain validated in this subsection, which uses $C_1 = 0$ and $C_2 = 10$, will also be employed for the calculations discussed in the next subsections and further specified when we consider large disturbances.
4.3 Non-parallel base flow

In this section, the results of calculations for non-parallel base flow are presented. The DNS results are compared with the results of PSE. For the comparison of growth rates, results from locally parallel LST are also incorporated. Disregarding the effect of linearization, we should, however, not expect exact agreement for the following reasons. In the PSE method, the upstream influence of instability waves is neglected to parabolize the governing equations for the disturbances, whereas LST completely neglects the non-parallel effect caused by the boundary-layer growth. In LST, locally parallel flow is assumed using a different length scale, $\delta^*(x_1)$, caused by the boundary-layer growth. The final results are then rescaled by a reference length.

4.3.1 Subsonic flow with small perturbations

We perform calculations at Mach number 0.5 and reference Reynolds number 750. The position of the computational domain relative to the stability diagram of the linear stability theory is shown in Fig. 4-11. The test case corresponds to the ray along a dimensionless frequency $F = 0.8$, where $F = 10^4 \omega' \mu/(\rho U_\infty^2)$ with $\omega'$ is the angular frequency in dimension. In dimensionless angular frequency, $\omega = \omega' \delta_1^* / U_\infty = 0.06$. The resulting eigenvalue from linear theory is

$$\alpha = \alpha_R + i\alpha_I = 0.1709 + i0.003549. \quad (4-29)$$

corresponding to point 1 in Fig. 4-11. This is a slightly stable mode close to the lower branch of the neutral curve. As we proceed downstream, the disturbance modes become unstable. The flow domain extends to a local Reynolds number about 2.25
times as large as the inflow Reynolds number, corresponding to a damping mode, somewhat beyond the upper branch of the neutral curve (point 2). The extent of the domain is equivalent to 25 disturbance wavelengths downstream of the inflow boundary. The last 4 wavelengths define the buffer domain. In the PSE calculations, 100 grid points were used in the wall-normal direction which extends to a height of 80, and 4 marching steps were taken per disturbance wavelength in the streamwise direction. Identical disturbances to the inflow of DNS are used to start the PSE calculations.

Figure 4-11: Position of the computational domain relative to the stability diagram of linear stability theory.

In contrast to calculations in the parallel flow case, the base flow of the non-parallel flow case is not the same as the similarity solution to the compressible boundary layer, which is used as the initial condition. Two ways can be followed for the specification of the base flow: by performing a steady state calculation first without imposing perturbations and by time averaging in the statistically stationary state. Although it implies that more work should be carried out, we preferred the latter method, for the following reasons. Firstly, performing a steady state calculation first for the base flow is somewhat unpractical and time consuming. Secondly, this method is not generally applicable. For instance for higher Reynolds number cases or flows subjected to adverse pressure gradients, which contain highly unstable modes, it is impossible to obtain a steady state solution using the present method. Small numerical errors will act as noise, which then amplifies as propagating physical waves ([88]). The third reason is, we want to examine the adequacy of the time averaging procedure, since it can be directly applied to more complex flows, such as separated flows ([88]) and turbulent flows. The time interval for the averaging process is taken here as one disturbance period and we repeat this process until the mean field has converged, which takes approximately ten disturbance periods. The
data sampling of the perturbations is subsequently carried out.

To appreciate the effect of the grid density three different resolutions are used, namely 16/wave $\times$ 64, 24/wave $\times$ 96 and 32/wave $\times$ 128 grid points. As an illustration of the computational time, the coarsest grid, for instance, requires about one hour CPU time on a single processor of a Silicon Graphics R8000. Growth rates resulting from these grids are compared with results from PSE and locally parallel LST in Fig. 4-12. The growth rates are taken from the $u'_2$ component. The Reynolds number $Re^*$ against which the growth rates are plotted is based on the local displacement thickness growing in the streamwise direction. Results from other components are, however, slightly more sensitive to the upstream influence of the outflow boundary and exhibit small standing waves around the curve of growth rates taken from $u'_2$. Small transients occur near the inflow boundary, in which the prescribed inflow adjusts to the solution of the governing system. The results of the grid 16/wave $\times$ 64 exhibit rather large deviations from the results of the finest grid, whereas using the grid 24/wave $\times$ 96 leads to nearly equivalent results. The result of PSE is closer to DNS than the result of locally parallel LST. The growth rates from LST are systematically lower than those of PSE. The same phenomenon has also been observed by Bertolotti in [5] and Fasel in [22]. Noting that the growth rate is a very critical comparison parameter, the agreement between DNS and PSE is very good in spite of the fact that the comparison is performed within a subsonic regime which is more critical for inflow and outflow boundary conditions. In the literature, comparison between DNS and PSE is commonly carried out in supersonic flows such as presented in [31] and [60]. A similar comparison between DNS and PSE at a subsonic Mach number was carried out by Van der Vegt in [81]. The current

![Figure 4-12: Comparison of growth rate between DNS, PSE and locally parallel LST. Three DNS results, taken from the $u'_2$ component, correspond to three different grids.](image)
Perturbed laminar flow under zero pressure gradient results compare favorably with the latter, especially regarding the upstream influence of the buffer domain which is very small in the present results. The maximum relative difference in the phase of TS waves resulting from grid 24/wave × 96 and grid 32/wave × 128 is also small, about 0.6%. The relative difference of the phase is defined as the shift of the TS waves in the streamwise direction relative to one wave length. It appears sufficient to use the coarser grid for the non-parallel test case. This phase difference is not appreciable if we plot the TS waves of the two finer grids in the same figure, therefore only the TS waves of the grid 24/wave × 96 is presented in Fig. 4-13 showing the development of the $u'_2$ disturbance in the streamwise direction. Corresponding to the development of the growth rates which shows successively negative, positive and again negative $\alpha_i$’s as a function of $x_1$, the disturbance decays, grows up to its maximum amplitude and then decays rapidly to zero. The decay near the outflow boundary is partly physical and within the buffer domain it is further increased by the buffer treatment. Note that in spite of the physical damping, the disturbance amplitude is still quite large when entering the buffer domain.

![Figure 4-13](image)

**Figure 4-13:** Development of the $u'_2$ disturbance in the streamwise direction, taken along $x_2 = 0.1604$.

In contrast to the growth rates $\alpha_i$ which are the same for all components, the development of the disturbance amplitude in the $x_1$ direction is different per component. This is clarified by considering the disturbances analogous to (4-24) for non-parallel flows:

$$v = e^{-\int \alpha_i dx_1} (\psi^v_R \cos(\int \alpha_R dx_1 - \omega t) - \psi^v_I \sin(\int \alpha_R dx_1 - \omega t)),$$

(4-30)

The corresponding amplitude of a disturbance component $v_i$ as function of $x_1$ is now

$$R_i = e^{-\int \alpha_i dx_1} \sqrt{(\psi^v_R)^2 + (\psi^v_I)^2}.$$

(4-31)

Here, besides the growth rates $\alpha_i$, the functions $\psi_R$ and $\psi_I$ also depend on $x_1$. Moreover, this dependence is unique per component. Hence in a non-parallel flow,
we expect that the streamwise position of the maximum amplitude for various components is not necessarily identical. For this reason it is important, along with growth rate comparison, to compare also the streamwise development of the disturbance amplitude of DNS and PSE for different components. This comparison is shown in Fig. 4-14. As expected, different disturbance components exhibit different streamwise developments of amplitude and different streamwise positions of global maxima. Although the maximum amplitudes of all components resulting from the PSE slightly underestimate those of the DNS, the DNS results generally compare well with the PSE results.

![Figure 4-14: DNS and PSE disturbance amplitude of various components versus Re*.](image)

Similar to the previous sections, we also compare the disturbance profiles of the DNS with the corresponding PSE results. This comparison is shown in Fig. 4-15, presented as the absolute value of disturbance components $u'_1$, $u'_2$ and $T'$. We perform the comparison at a streamwise location corresponding to $Re^* = 1570.2$ which is beyond the amplitude maxima and near the beginning of the buffer domain. The complex structure of the disturbance profiles is accurately captured, and a critical examination shows that the PSE results again slightly underestimate the DNS results. Performing the comparison at various streamwise locations yields similar results, indicating that the upstream influence of the buffer domain is insignificant and that the observed deviation is not related to the buffer domain. In contrast to parallel flow comparisons between DNS and LST, in which we are certain that the modelling error due to linearization is of the order of the square of the disturbances, here we face an uncertainty regarding the physical effects of the approximation beyond linearization in the PSE method, namely the parabolization. This might form an explanation for the slight underestimation of the PSE. As a consequence of neglecting upstream influences in the PSE method, we might expect
a better agreement between DNS and PSE when we would consider higher Mach numbers, since the upstream influences are smaller with increasing Mach numbers.

![Figure 4-15: PSE and DNS disturbance profiles at x_1 = 988.34 (Re = 1570.2).](image)

### 4.3.2 Subsonic flow with large perturbations

A non-parallel flow simulation in the nonlinear regime is conducted by exactly repeating the linear case with an increased disturbance amplitude ε = 0.01. The grid 32/wave × 96 is used to capture the higher amplitude of the fluctuations. The effects of nonlinearity are examined by comparing the results with those of the linear case. A comparison of growth rates between the linear and nonlinear case is shown in Fig. 4-16a, taken from the u^2_0 component. In the beginning, the growth rates of the large disturbances roughly follow those of the small disturbances, but after reaching a maximum value, contrary to the linear case, they exhibit only a small decrease and end up still considerably growing (α_i < 0) before entering the buffer domain. The deviation from the linear case is also enhanced by the occurrence of small standing waves in the growth rates of the nonlinear case, indicating that flow with large disturbances experiences larger upstream effects in spite of the buffer domain treatment. A nonlinear effect is also appreciable in the shape of the TS waves. This is illustrated in Fig. 4-16b by plotting the same TS wave as in Fig. 4-13. The nonlinear TS wave exhibits an irregular pattern in the streamwise direction instead of a sinusoidal form as in the linear case. Besides, it grows monotonically up to the buffer domain. A nonlinear effect can be seen as well if we compare the mean flow of the linear and nonlinear cases. We can see a deformation in the perturbation vorticity contours due to the nonlinearity in Fig. 4-17. The absolute value of the perturbation vorticity for the linear case decreases already upstream of the buffer domain associated with damping modes, whereas the corresponding value in the nonlinear case grows monotonically until the buffer domain. Considering the
sign of the perturbation vorticity, it consists of positive and negative parts, marching alternatingly in the streamwise direction. Contrary to the linear case in which the positive and negative parts of the perturbation vorticity are nearly equal, the negative part in the nonlinear case is more pronounced than the positive part.

Regarding the occurrence of small standing waves in the growth rates of the nonlinear case, a further investigation has shown that this is due to a much larger decrease of the disturbance amplitude within the buffer domain than in the linear case. The larger amplitude must be suppressed to zero within the same buffer domain length. Consequently, we should employ a higher grid density in the buffer domain or extend the length of the buffer domain to anticipate this amplitude reduction. However, this implies a higher computational cost. A more elegant remedy, avoiding an increase in the computational cost, is to attempt a more gradual decrease of the
perturbed laminar flow under zero pressure gradient. This can be achieved by activating the parameter $C_1$ in the damping function, (3-24), forcing a more gradual suppression of disturbances in the front part of the buffer domain, and at the same time increasing the value of $C_2$, postponing a rapid decrease to zero in the rear part. $C_1 = 0.005$ and $C_2 = 20$ are found to be appropriate values. The performance of the damping function due to this alternative tuning is demonstrated by comparing the growth rates of the nonlinear case resulting from the two sets of parameter in Fig. 4-18. This comparison illustrates a substantial improvement due to a more gradual decrease of the damping function. Note that the growth rates resulting from $C_1 = 0.005$ and $C_2 = 20$ directly after entering the buffer domain decrease more rapidly due to a lower value of the damping function in the interior of the buffer domain. Testing the damping function with these new values in the case of small disturbances we found a slight improvement in the growth rates but inappreciable effects in other quantities. Based on this result, the new parameter values can be used in both linear and nonlinear regimes. Finally, we notice that the effects of nonlinearity discussed above become more profound when increasing the amplitude of forced disturbances.

### 4.3.3 Conclusion

In the previous chapter, numerical methods were developed which are suitable for direct numerical simulations of compressible laminar-turbulent transition in boundary-layers. The important aspects of these methods are the fourth order central difference approximation and the specification of outflow boundary treatments. From the present validation we conclude that the fourth order scheme yields considerably
more accurate results than the second order scheme for the same computational effort. In flow simulations with time dependent perturbations, spurious reflections are generated by the inflow and outflow boundary conditions. The coupling between these boundaries results in the formation of standing waves in the interior domain. This distortion becomes large with increasing amplitude of disturbances and can even result in a divergent solution. The wave reflections can, however, be significantly damped by applying a buffer domain treatment in addition to the outflow boundary condition. On the other hand, the upstream influences become smaller with increasing Mach numbers. Hence, this buffer domain treatment is not required if we consider high supersonic Mach numbers.

The buffer approach employs a direct reduction of the fluctuations of the solution components within the buffer domain, using a damping function. A previous investigation confirms that this direct approach is more efficient than other, indirect, approaches such as increasing viscosity, parabolizing procedure, base flow acceleration and increasing grid spacing ([86]). The validation of the damping function is performed by testing the buffer domain in successively more difficult testcases. In this way the domain of the damping function parameters is narrowed and the eventually selected values of these parameters are applicable for both the linear and nonlinear regimes. Numerical experiments show that the present buffer domain approach exhibits very small upstream influences. Moreover, it is insensitive to the extent of the domain of interest, the reference parameters and the grid density.

The calculation results of flows with small disturbances using the developed numerical methods compare very well with the results of LST and PSE. In agreement with the observation of other authors, the locally parallel assumption of LST underestimates the growth of disturbances in a non-parallel flow. The good agreement between results of the DNS and the PSE supports the use of shape functions resulting from PSE as the imposed disturbances of DNS. The numerical methods have also been successful in performing calculations with large disturbances, advancing the flow into the nonlinear regime. Although, contrary to the outflow boundary, no additional treatment is applied to the inflow boundary conditions to damp spurious reflections generated by this boundary, the error near the inflow boundary is small. This satisfactory result encourages the use of the employed numerical methods in more complex flows such as flows which proceed into the turbulent regime and flows with separation regions ([88]). These applications form the subject of the next chapters.
In Chapter 4 the validity of the developed numerical methods has been assessed for relatively simple flows. The fourth order central discretization method appears more efficient than the second order method, hence the fourth order method is used for the next applications. In this chapter the methods are tested for complex cases. We consider laminar separation bubble flows in two-dimensions and a transition induced by separation in three-dimensions. The simulations are performed in a low Mach-number regime in order to compare the results with incompressible theory as well as with other numerical studies reported in the literature ([4], [55], [58] and [62]). The separation is invoked by an adverse pressure gradient along the upper boundary. In two dimensions, the flow may exhibit a strong unsteady behavior in the form of vortex shedding. In three dimensions, small structures and fluctuations in the turbulent regime after reattachment should be spatially as well as temporally resolved. This forms a difficult test for the selected numerical discretization methods. In addition, the strongly fluctuating flow should not be reflected by the outflow boundary, which is a challenge for the developed buffer domain technique.

Intensive theoretical and numerical studies have been made on laminar separation bubbles in incompressible flows. Semi-empirical methods to predict separation have been proposed by, among others, Stratford ([76]) and Thwaites([80]). Dobbinga et al. ([14]) and Oswatitsch ([57]) presented methods to predict the angle between the wall and the dividing streamline at the separation point. Using direct numerical simulations for incompressible flow, Pauley et al. ([58]) found that, for relatively weak adverse pressure gradients, the separated region builds up into a steady separation bubble. This region grows with increasing adverse pressure gradient, until it reaches a certain critical value at which unsteady separation sets in, characterized by a regular, self-excited vortex shedding. Beyond this critical value the length of the time-averaged bubble decreases with increasing adverse pressure gradient. Furthermore, it was found that the shedding frequency, nondimensionalized by the edge velocity and the boundary-layer momentum thickness at separation, is independent of the Reynolds number and the strength of the pressure gradient ([58] and [63]).
A criterion for the onset of self-excited vortex shedding was proposed in [58]. The instability of incompressible flows containing a laminar separation bubble has also been investigated numerically, among others, by Maucher & Rist ([55]) and Alam & Sandham ([4]). It was found that, if a small amplitude forcing is introduced upstream of the steady bubble, the separated shear layer strongly amplifies the upstream disturbances. Moreover, a periodic vortex shedding was observed by Alam & Sandham. Increasing the suction strength up to a certain critical level, they also observed self-excited vortex shedding, which is in agreement with Pauley et al. Moreover, Alam & Sandham reported differences in the statistic of velocity fluctuations in the region after reattachment in two- and three-dimensions.

In the present work computations of perturbed and unperturbed flows at Mach number 0.2 subjected to various pressure gradients are carried out in two- and three-dimensions to verify these findings. From this comparison we can appreciate the performance of the developed numerical methods which are essentially different from incompressible flow solvers in terms of the governing equations, the simulation algorithm and the boundary conditions, especially along the artificial boundaries.

In addition to the comparison of the basic flow features, we also investigate the effect of different boundary conditions on the resulting laminar separation bubble. In many comparisons between numerical simulation and experiment, an inviscidly derived suction-blowing boundary condition is used in the numerical calculation to match the pressure distribution of the experiment. This method is used for example by Ripley & Pauley ([62]) to compare their numerical results with the experiment of Gaster ([26]). We could also directly prescribe the normal velocity provided by the experiment. To appreciate the effect of the different procedures of defining the freestream boundary condition, we compare three closely related calculations in two-dimensions. In the first calculation, a prescribed pressure distribution is used along the upper boundary. In the second, we use the time-averaged normal velocity on the upper boundary resulting from the first calculation to fix the suction. In the third, a normal velocity distribution is fixed along the upper boundary which is derived from the potential-flow assumption and which matches the prescribed pressure distribution of the first calculation. The occurrence of spontaneous vortex shedding is established at sufficiently high adverse pressure gradients. Finally, we model a transition induced by separation in three-dimensions by imposing three-dimensional perturbations at the inflow boundary. These perturbations trigger secondary instabilities. We compare the result with its two dimensional counterpart in order to study the influence of the three dimensionality in the perturbations. The requirement of the computer resources for the three dimensional simulation is also reported.

This chapter is organized as follows. In Section 1 we present a flow separation induced by a pressure prescription boundary condition in two dimensions. In Section 2 we repeat the simulation by using a blowing/suction boundary condition. In Section 3 we discuss the influence of the pressure gradient and upstream disturbances. In Section 4 the three dimensional simulation of separation induced transition is presented. We state the conclusions in Section 5.
5.1 Pressure prescription boundary condition

The computation is performed on a grid containing $320 \times 96$ cells in the streamwise and the normal direction, respectively. The height of the domain is 30 and the length of the physical domain is 330. The $x_1$ grid is uniform, whereas the $x_2$ grid is stretched according to Eq. 4-27, with stretching parameter $S_r = 0.35$. The inflow boundary is located at $x_1 = 109$ downstream of the flat-plate leading edge. The Mach-number is 0.2 and the Reynolds number is 330. These physical parameters are globally comparable with those used by Pauley et al. (incompressible with $Re$ ranging from 400 to 800) and by Alam & Sandham (incompressible with $Re = 500$). At the inflow boundary a periodic disturbance is superimposed on the compressible Blasius solution which has the form given by Eq. 4-24. The disturbance amplitude $\epsilon$ is selected to be 0.001. Using the circular frequency of 0.0594, we find that the eigenvalue provided by the linear stability theory is given by

$$\alpha = \alpha_r + i\alpha_i = 0.1640 + i0.019687,$$

which is a stable mode. The time development of the dependent variables at a certain point is followed. A statistically stationary solution is reached when each variable fluctuates around a nearly constant value. The sampling of data is then carried out over several disturbance periods. The prescribed pressure distribution along the upper boundary is isentropically related to the following streamwise velocity distribution:

$$u_1(x_1) = \begin{cases} 1 & \text{if } x_{\text{in}} \leq x_1 \leq x_s, \\ 1 + \frac{1}{2}\Delta u_1(\cos\left(\frac{x_1 - x_s}{x_e - x_s} - 1\right) & \text{if } x_s \leq x_1 \leq x_e, \\ 1 - \Delta u_1 & \text{if } x_e \leq x_1 \leq x_{\text{in}} + L_1, \end{cases}$$

(5-2)

where $x_{\text{in}} = 109.3$ denotes the streamwise coordinate of the inflow boundary, $x_s = 119.3$ and $x_e = 369.3$ are the interval boundaries between which the freestream flow is decelerated, and $L_1 = 400$ is the length of the computational domain. The larger the velocity drop $\Delta u_1$ the higher the adverse pressure gradient will be. Varying this parameter, we found that $\Delta u_1 = 0.10$ corresponds to an almost separated flow. In the following, we present the result for $\Delta u_1 = 0.12$, which does lead to separation. Fig. 5-1a shows the resulting time-averaged separation bubble, while the prescribed pressure distribution is shown in Fig. 5-1b along with the time-averaged wall pressure distribution. The maximum height of the separation bubble is 1.2 with the height at location $x_1$ defined as the value of $x_2$ at which $f(x_1, x_2) = \int_0^{x_2} \rho u_1 dx' = 0$. This maximum height is only 20% larger than $\delta_i^+$, the displacement thickness at the inflow boundary. Hence, the prescribed pressure gradient can be classified as weak. The center of the recirculation region, which is defined as the position $(x_1, x_2)$ where $f(x_1, x_2)$ is minimal, is located near the reattachment point. The wall pressure distribution deviates notably from the prescribed distribution at the upper boundary. Proceeding downstream in the separation region, we observe that the pressure gradient decreases until the center of the recirculation region is
reached, behind which it strongly increases right up to the reattachment point. This strong compression near the reattachment region is also observed by Pauley et al. and by Gaster ([26]).

![Figure 5-1](image)

**Figure 5-1:** a) Streamlines of time-averaged bubble due to prescribed pressure, \( \text{Re} = 330 \) and \( M = 0.2 \). b) Pressure distribution, prescribed along the upper boundary (dashed) and along the wall (solid).

To verify the separation point condition, we compare our result with some semi-empirical relations. Here, these relations are described in a non-dimensional form. Stratford ([76]) predicted that separation can be expected if

\[
S(x_1) = C_p \left( x_1 \frac{dC_p}{dx_1} \right)^2 = 0.0104, \quad (5-3)
\]

where

\[
C_p = 2(p - p_s)/(\rho u_s^2). \quad (5-4)
\]

The subscript \( s \) denotes the location where the adverse pressure gradient starts, equal to \( x_s \) in equation (5-2). Our result agrees very well with this prediction, as can be seen in the development of the skin friction \( c_f \) which is plotted in Fig. 5-2 along with the value of the Stratford criterion. The value of this criterion based on our result is 0.0101 at the location where \( c_f = 0 \). We note that the quantities used in calculating the Stratford criterion are averaged over the boundary-layer thickness, since the pressure varies across the boundary layer. The minimum skin friction corresponds to the center of the recirculation region. Another method of separation prediction is proposed by Thwaites ([80]), who found that the value of

\[
m = \text{Re} \left[ \theta^2 \frac{\partial u_e}{\partial x_1} \right]_{\text{sep}} \quad (5-5)
\]

at separation is approximately \(-0.082\), where \( u_e \) denotes the local freestream velocity, \( \theta \) the momentum thickness of the boundary layer and the subscript sep
the separation point. The corresponding value in the present calculation is in the range $-0.103 < m < -0.065$, depending on the exact definition of the edge velocity $u_e$ in the simulation. This is consistent with values found by Pauley et al. ($-0.121 < m < -0.076$) and by Curle & Skan ($-0.171 < m < -0.068$) reported in [12]. Dobbinga et al. ([14]) predicted the angle between the wall and the dividing streamline, $\varphi$, at the separation point as

$$\tan \varphi = \frac{B}{R_{\theta s}}$$

(5-6)

where $R_{\theta s}$ denotes the Reynolds number based on the momentum thickness at the separation point and $B$ is typically between 15 and 20. According to this prediction, the separation angle is in the range of $0.0588^\circ$ to $0.0784^\circ$. The present result provides a separation angle of $0.0662^\circ$, which is within the empirical range. We scrutinize the sensitivity of the result for the grid density by using 64 grid points in the normal direction. We find that the resulting maximum deviations in the boundary-layer integral parameters $\delta^* \text{ and } \theta$ are $1.2\%$ and $2\%$, respectively and, in the maximum height of the separation bubble, $3\%$.

Next, we check upstream influences of the buffer domain by varying its length. These influences are small, as is illustrated in the development of the disturbance amplitude calculated with three different buffer lengths in Fig. 5-3a. The results within the domain of interest remain the same. Note that the vertical axis represents the logarithm of the disturbance amplitude. From this figure we observe that the disturbance amplitude is initially damped, which is consistent with the imposed stable mode. The disturbance mode, however, becomes unstable when $\partial_1 p$ at the wall grows, since this gives rise to an inflection point in the streamwise velocity profile ([53]). The increase of the disturbance amplitude ends near the recirculation center and the amplitude remains nearly constant further downstream. The corresponding
growth rate, which is defined as the derivative of the logarithm of the disturbance amplitude with respect to the streamwise coordinate, is compared with the numerical results of two linear perturbation theories in Fig. 5-3b: linear stability theory (LST) and the linear parabolized stability equations (PSE), which are presented in Chapter 4. As can be seen in Fig. 5-3b, the growth rate of the disturbances from the present

![Figure 5-3: a) Disturbance amplitude in response of the separation bubble flow calculated with three different buffer lengths normalized by its inflow value. b) Corresponding growth rate compared with LST (circles) and PSE (dashed).](image)

calculation agrees well with predictions from LST and PSE up to the beginning of the separation region. Beyond this point the deviation becomes larger as the disturbances increase. From the similarity of the results obtained with LST and PSE,
we conclude that the observed deviation is caused by nonlinear effects downstream of the separation point, especially at the position of maximum reverse flow (minimum skin friction), rather than by the locally parallel assumption of LST. Although we only show the amplitude and the growth rate of the streamwise velocity component, those of the other components behave similarly. The separation region thus acts as an amplifier of the upstream disturbances as suggested by Gaster ([26]) and Rist & Maucher ([55]). We observe, however, that the rate of the amplification strongly decreases behind the center of the time-averaged recirculation region, which is caused by a fast decrease in the strength of the reverse flow. Observing the vortical structure of the flow as shown in Fig. 5-4, we note that periodic vortex shedding occurs near the reattachment region. The shedded vortices are further advected downstream with almost constant thickness at a speed of approximately 40% of the reference velocity. Note that the vortices are damped in the buffer domain. The nearly constant thickness of the vortices corresponds to the almost constant amplitude of disturbances behind the location of vortex shedding. This vortex shedding thus marks a breakdown of the laminar boundary layer. By recording the time-oscillation of the dependent variables, we find that the circular shedding frequency is 0.052753, which differs only 0.2% from the imposed perturbation frequency. This suggests that the vortex shedding is directly induced by the Tollmien-Schlichting waves imposed at the inflow boundary.

5.2 Blowing and suction boundary condition

To appreciate the influence of different realizations of the freestream boundary conditions in generating separation, we perform two additional calculations closely corresponding to the case described above, but now using a suction technique. Only the employed freestream boundary conditions are different, the other physical and geometrical parameters remain the same. This suction technique implies that the normal velocity along the upper boundary is prescribed and the other dependent variables are calculated with the characteristic method, as described in Chapter 2. In the comparison, the previous calculation, which uses the prescribed pressure, is denoted as case A, while the calculations that use the suction technique are denoted as case B and case C. In case B the time-averaged normal velocity along the upper boundary resulting from case A is used as the suction. In case C we derive the suction by assuming incompressible potential flow through a 2D non-uniform channel, as illustrated in Fig. 5-5. The normal velocity along the upper wall defines the suction. The lower wall of this channel is shaped according to the time-averaged displacement thickness $\delta_1$ of case A, in order to take the presence of the boundary layer into account. The channel width $A$ varies according to the streamwise variation of the pressure $\bar{p}$ which is assumed to be constant across the channel width. This pressure distribution $\bar{p}$ is obtained from the mean pressure of case A averaged in the normal direction. We start the derivation of the suction by defining the velocity $q$ along streamlines which correspond to the pressure distribution $\bar{p}(x_1)$ using the
incompressible Bernoulli equation,
\[ q = (1 + 2(p_{\infty} - \bar{p}(x_1)))^{\frac{1}{2}}, \tag{5-7} \]
where \( p_{\infty} = \gamma^{-1} M^{-2} \) is the undisturbed pressure in the far field. In case A, the maximum normal velocity is small compared to the freestream velocity (\( \approx 3.5\% \)). Therefore, we approximate the streamwise velocity of the channel flow by its stream-

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{figure5-5.png}
\caption{2D channel potential flow configuration comparable to a flat plate viscous flow subjected to suction.}
\end{figure}

line velocity \( U_1(x_1) \approx q(x_1) \). Recalling that this streamwise velocity is approximately independent of the normal coordinate and using the continuity equation, we find that
\[ U_1(x_1)A(x_1) = U_o A_o, \tag{5-8} \]
where the subscript \( o \) corresponds to a reference streamwise location. Taking the inflow boundary as this reference point, we relate the channel width \( A(x_1) \) to the streamwise velocity \( U_1(x_1) \) according to,
\[ A(x_1) = \frac{U_o}{U_1(x_1)} A_o. \tag{5-9} \]
We take \( A_o = 29 \), which is different from the height of the computational domain in the DNS in order to take the displacement thickness of the boundary-layer into account. In a potential flow, the slope of the streamline along the wall follows the slope of the wall. Hence,
\[ U_2(x_1) = U_1(x_1) \frac{d}{dx_1}(A(x_1) + \delta_1(x_1)), \tag{5-10} \]
where \( \delta_1(x_1) \) is the displacement thickness distribution of the boundary-layer in case A and \( U_2(x_1) \) the normal velocity along the upper wall. This normal velocity defines the suction along the upper boundary in case C. The time-averaged normal velocity
resulting from case A, which is used as the suction in case B, and the prescribed suction in case C are quite similar, as shown in Fig. 5-6a.

The time-averaged quantities resulting from case A and case B are almost the same, whereas the discrepancy between case C and the other two cases is in general only very small up to the separation point, but becomes more pronounced further downstream. This is illustrated for example by the wall pressure and the skin friction of the mean flow in Fig. 5-6b and Figure 5-7a, respectively. The lines corresponding to case A and case B cannot be distinguished. As case B and case C employ the same suction technique, the slightly larger suction near the reattachment point in case C is the main reason for the deviation in the pressure downstream of the reattachment point. A similar behavior is also observed in the skin friction of the time-averaged flows. Due to the larger suction, case C provides a lower minimum skin friction, representing a stronger reverse flow, and a longer separation region.

![Figure 5-6: Mean freestream normal velocities (a) and mean wall-pressure distributions (b) corresponding to case A (solid), case B (dashed-dotted) and case C (dashed).](image)

The disturbance amplitude in case A is also the same as in case B as shown in Fig. 5-7b. As a consequence of the larger separation region and stronger reverse flow in case C, its disturbance amplitude is slightly higher than in the other cases in the region of separation and further downstream.

From these results we conclude that flows subjected to the pressure boundary condition and the suction boundary condition are equivalent if the freestream pressure in the first case and the freestream normal velocity in the latter case exactly correspond. The difference in the time dependence of the freestream variables (in case A the freestream pressure is steady, while in case B the normal velocity is steady) has apparently a negligible influence. Performing the same comparison for a higher
pressure gradient, we found that the equivalence of the two boundary conditions is independent of the pressure gradient.

5.3 Effect of pressure gradient and upstream disturbances

In this subsection we investigate the effect of the pressure gradient along the upper boundary on perturbed and unperturbed flows. Therefore, we repeat case B, but without imposing any disturbances, further referred to as case B', representing an unperturbed flow under a weak adverse pressure gradient. In addition, we perform perturbed (case B2) and unperturbed flow (case B2') simulations subjected to a stronger adverse pressure gradient realized by a relatively large suction. The suction in case B2, which is also used in case B2', corresponds to a decelerating freestream flow as described by equation (5-2) with $\Delta u_1 = 0.16$. Apart from these modifications, the physical flow parameters and the flow configuration are kept the same. The characteristics of the above mentioned cases are summarized in Table 5-1.

We compute the unperturbed flows, case B' and case B2', by removing the imposed disturbances at the inflow boundary. The suction used in cases B2 and B2' is compared to the freestream normal velocity prescribed in the cases B and B' in Fig. 5-8a. The time-averaged separation bubbles resulting from the cases B, B', B2 and B2' are represented in Fig. 5-8b and the corresponding skin friction
Laminar separation and transition

<table>
<thead>
<tr>
<th>Case</th>
<th>separation generator</th>
<th>perturb.</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>prescription of weak adv. press. grad. (a.p.g)</td>
<td>yes</td>
</tr>
<tr>
<td>C</td>
<td>suc./blow. derived from pot. theory imitating a.p.g. in case A</td>
<td>yes</td>
</tr>
<tr>
<td>B</td>
<td>suc./blo. using time averaged upper $u_2$ in case A</td>
<td>yes</td>
</tr>
<tr>
<td>B'</td>
<td>the same as case B but inflow perturbations added</td>
<td>no</td>
</tr>
<tr>
<td>B2</td>
<td>suction/blowing, stronger adv. press. grad. than in case B</td>
<td>yes</td>
</tr>
<tr>
<td>B2'</td>
<td>the same as case B2</td>
<td>no</td>
</tr>
</tbody>
</table>

Table 5-1: Upper boundary conditions and existence of inflow perturbations, corresponding to different cases.

Figure 5-8: a) Prescribed freestream normal velocity in case B2 (dashed) and case B (solid). b) Boundary of separation bubble resulting from case B (solid), B' (dotted), B2 (dashed) and B2' (dashed-dotted).

distributions are given in Fig. 5-9. We recall that the length of the region of negative skin friction represents the length of the separation bubble, while the minimum value of the skin friction represents the strength of the reverse flow. The comparison shows that a stronger adverse pressure gradient results in a higher separation bubble and a stronger reverse flow independent of whether the flow is perturbed or unperturbed. We note that the reattachment point in case B2' is quite close to the beginning of the buffer domain. In order to study the influence of the position of the buffer domain on the basic separation process, we repeated the calculation of case B2' with a different length of the physical domain. This has virtually no influence on the reattachment point. We observed, that the domain of noticeable upstream influence of the buffer is about 30 dimensionless units and is hence strongly localized.

Under the same pressure gradient, the location of the separation point is not affected by the presence of upstream disturbances. However, the unperturbed flows generate longer separation regions and weaker reverse flows than the corresponding...
perturbed flows. This effect of upstream disturbances is also observed in the incompressible flow simulation of Alam & Sandham. From the present results we note that this upstream perturbation effect is stronger with increasing adverse pressure gradient. We also observe that the recirculation centres of the perturbed flows are located near the reattachment point, while those of the unperturbed flows are near the middle of the separation region.

In case B' the absence of imposed disturbances results in the disappearance of vortex-shedding such as observed in case B and the instantaneous flow has the same steady character as the time-averaged flow, whereas under the larger suction in case B2' it gives rise to the occurrence of self-excited vortex shedding. While in case B2 the vortex shedding is periodic in time, the shedding in case B2' is irregular. This is illustrated in Fig. 5-10 in which the time development of the streamwise velocity near the wall at \( x_1 = 430 \) is shown. The shedded vortices in case B2 (forced vortex shedding) are stronger than those in case B2' (self-excited) as can be seen in Fig. 5-11. Moreover, the vortex shedding in case B2 occurs earlier than in case B2'. The spatial wave number of the disturbances resulting from case B2 and case B2' is plotted in Fig. 5-12a. The observed three peaks in case B2 correspond, for increasing wave numbers, to the disturbances further downstream of the vortex shedding location, upstream of it and at the location of vortex shedding. In case B2' the spectrum shows only one dominant wave number which corresponds to the disturbances downstream of the self-excited vortex shedding. The multiple peaks in the spatial wave number in case B2 implies that the propagation velocity of the disturbances is location dependent, since we found that the temporal wave number of the disturbances is independent of location. The lowest propagation velocity is at the location of vortex shedding.
The breakdown of the perturbed laminar shear layer under strong suction (case B2) occurs earlier than under weaker suction (case B), as can be seen in the streamwise velocity disturbance amplitude in Fig. 5-12b. This figure also confirms the earlier breakdown of the perturbed shear layer (case B2), in the form of vortex shedding, compared to the unperturbed one (case B2'). The disturbance amplitudes of other components behave similarly. The different level in the disturbance amplitude in case B2 and case B2' corresponds to the observed disturbance drop upstream of the reattachment region. Focusing on the region downstream of the inflow boundary in case B2', we find that the perturbations oscillate with nearly uniform wavelength. This observation, in combination with the disturbance spectrum and the disturbance amplitude development, suggests that the origin of the self-exciting vortex shedding in case B2' is formed by Tollmien-Schlichting waves.
The waves are naturally generated from the existing numerical noise in the flow and their amplitude is enhanced by the strong adverse pressure gradient.

A parameter characterizing the vortex shedding is the so-called Strouhal number, which is defined as the shedding frequency non-dimensionalized by the boundary-layer momentum thickness and the local freestream velocity at the separation point:

\[ St = f \left( \theta / u_e \right)_{\text{sep}}, \]  

(5-11)

where \( f \) denotes the shedding frequency. Pauley et al. ([58]) suggested that the Strouhal number is independent of Reynolds number and the pressure gradient. The Strouhal numbers deduced from case B2 and B2’ are 0.00672 and 0.00756, respectively. From this we conclude that the presence of small upstream disturbances can affect the Strouhal number, even if the Reynolds number and the pressure gradient are the same.

### 5.4 Transition induced by separation bubble; 3D simulation

The simulations performed so far are two dimensional. These two dimensional simulations were an essential step in the development of the numerical methods. From the physical point of view, however, two dimensional simulations are often not realistic as flows in many practical applications contain three dimensional phenomena. Moreover, two dimensional flows in practice are predominantly two dimensional only...
as far as large scales (averaged quantities) are concerned but three dimensional on the small scale level (fluctuating quantities). This also holds for a laminar separation bubble flow, with associated turbulent reattachment. Hence, in spite of the enormous computational demand, we perform a three dimensional simulation of a laminar separation bubble flow. Moreover, we want to study the feasibility of the numerical methods for turbulence simulations, which necessarily require full resolution in three dimensions.

5.4.1 Configuration

We perform a three dimensional simulation within the computational box shown in Fig. 5-13. The physical configuration is similar to the two dimensional cases described previously. Specifically, a suction dominated normal velocity distribution shown in Fig. 5-14a is prescribed along the upper boundary in order to produce an adverse pressure gradient. The wall is isothermal and the wall temperature is equal to its adiabatic value at the inflow boundary while the reference temperature is 276K. A periodic boundary condition is employed in the spanwise direction. We set the Mach number to 0.2, so that the effect of compressibility is negligible. Based on the inflow displacement thickness, the Reynolds number is 330, and the inflow and outflow boundaries are located at $x_1 = 109.33$ and 348.14, respectively. This is equivalent to $R_{x_1}$ (Reynolds number based on $x_1$ coordinate) ranging from $3.68 \times 10^4$ to $8.03 \times 10^4$ or $R^*$ (based on the laminar spreading of displacement thickness, Eq. 5-14) ranging from 330 to 488. The height of the domain is 20 and the spanwise extent is $[-\pi/|\beta|, \pi/|\beta|]$, where $\beta$ is the spanwise wave number of the inflow perturbations, given below. Three dimensional linear eigenfunction perturbations of the following form are superimposed upon the inflow Blasius profiles:

$$v = \text{Real}[\epsilon_{2d}\psi_{2d}(x_2)e^{-i\omega t} + \epsilon_{3d}(\psi_{3d}^+(x_2)e^{i(\beta^+x_3-\omega t)} + \psi_{3d}^-(x_2)e^{i(\beta^-x_3-\omega t)})], \quad (5-12)$$

where $v$ represents a component of perturbations ($p', u'_1, u'_2, u'_3, T'$), while $\psi_{2d}$ and $\psi_{3d}^{\pm}$ are 2D and 3D eigenfunction vectors corresponding to the perturbation component. Note that the contribution of the wave number $\alpha$ (complex) in the perturbations is incorporated by multiplying Eq. 5-12 with $e^{i(\alpha(x_1-x_0))}$ within the square brackets, where $x_0$ is the streamwise coordinate at the inflow boundary. In Eq. 5-12, however, $e^{i(\alpha(x_1-x_0))} = 1$ as $x_1 - x_0 = 0$ at the inflow boundary. The circular frequency $\omega = 0.15$ ($F = 10^4\omega/Re = 4.55$) and the wave numbers $\beta^\pm = \pm \alpha_r = \pm \Re(\alpha)$. The eigenvalues resulting from this choice of parameters are:

$$\alpha_{2d} = 0.3455 + i0.00842,$$

$$\alpha_{3d} = 0.3170 + i0.01811 \quad (\beta^\pm = \pm 0.3170),$$

which are both damping modes. These high frequency waves are selected to keep the spanwise extent of the computational domain small. These linearly stable modes give rise to growing perturbations under the influence of the adverse pressure gradient. The streamwise extent of the computational domain correspond to 13 waves.
of the 2D mode or 12 of the 3D mode. We did not observe a transition process through separation induced instabilities without forcing perturbations at the inflow boundary. Spalart & Yang [71] reported the absence of natural transition at zero pressure gradient for perturbation amplitudes below $4 \times 10^{-3}$. In the case where the Reynolds number is high or the adverse pressure gradient is strong, however, we expect that a spontaneous transition, without forcing inflow perturbations, may occur such as a spontaneous vortex shedding does in the two dimensional simulations. In order to generate secondary instabilities in the present low Reynolds number flow, we set the perturbation amplitudes to $\epsilon_{2d} = 0.01$ and $\epsilon_{3d} = 0.01$ for the two and three dimensional waves, respectively. A $402 \times 64 \times 48$ grid is used in the $x_1, x_2$ and $x_3$ direction, respectively, using the same stretching in the normal direction as in the two dimensional separation bubble cases (stretching parameter $S_r = 0.35$). With
these grid points, the grid resolution is \((\Delta x_1, \Delta x_2, \Delta x_3)=(0.597, 0.082, 0.413)\) based on the inflow displacement thickness, or equivalent to \((\Delta x_i^+, \Delta x_j^+, \Delta x_k^+)=\) \((9.76, 1.34, 6.75)\), where \(x_i^+ = x_i u_r Re\). Here, we use \(u_r = 0.05U_\infty\), which is a typical value in the turbulent region in the present simulation. As a comparison, the resolution required for the DNS in the temporal setting used by Spalart [72] is \((\Delta x_i^+, \Delta x_j^+, \Delta x_k^+)=\) \((20, 1, 6.7)\) and for the DNS in the spatial setting by Kloker [42] is \((\Delta x_i^+, \Delta x_j^+, \Delta x_k^+)=\) \((8.3, 1.7 \sim 3.4, 3.8)\).

In addition to the 3D simulation, we also performed the 2D counterpart simulations with small amplitude \((\varepsilon = 0.0001)\) and large amplitude \((0.01)\) perturbations in order to reveal the influence of three dimensionality in perturbations. The result can be seen in the inviscid streamwise velocity distribution (Fig. 5-14b) and the development of the skin friction coefficient and shape factor (Fig. 5-15a and 5-15b) as a function of \(x_1\). It appears that any change aimed at an increase of the rate of perturbation amplification in a laminar separation bubble flow, whether through an increase of amplitude or by imposed 3D perturbations, results in an enhance of flow deceleration (adverse pressure gradient), an earlier reattachment (from \(c_f\) data) and an earlier as well as larger decrease of the shape factor. This observation is consistent with the results of the perturbed and unperturbed cases in the previous section. Moreover, the 3D perturbations generate stronger reverse flow in the separation bubble. The rapidly varying negative skin friction in the 3D case remains even if we perform the flow averaging over a much longer time interval. We suspect that this is due to high vorticity gradients, marking the start of transition as will be shown later, which would need a somewhat higher grid resolution in this area. The cost aspect, however, prevents us to conduct such a resolution check and we focus on the feasibility study.

The averaging process begins at dimensionless time \(t = 700\), which is 1.25 times as large as the time needed for the perturbation waves to travel from the inflow boundary to the outflow boundary. This can be considered as short, as typically the value 2 is used [18]. We checked, however, the time development of the solution in a laminar and a turbulent region, as presented by the streamwise velocity record at two streamwise locations in Fig. 5-16. At \(t = 400\), the location \(x_1 = 300\) becomes turbulent and the turbulence reaches the outflow boundary at \(t = 515\). From \(t = 800\) the averaged value over the spanwise direction and time has become virtually constant. We take the averaged flow at \(t = 950\) as the base flow and from there start calculating root mean square quantities up to \(t = 1200\). The whole process required about 310 CPU hours on one processor of a Cray C90, 40% of which is used for the data sampling. The total number of time steps is about 86000. For a comparison, Rai & Moin [61] performed a direct simulation of natural transition applying an implicit method with about 16 million grid points and required 800 CPU hours on a Cray YMP only for the data sampling.
5.4.2 Flow visualization

All the visualizations shown here are taken at the end of the simulation, i.e. at $t = 1200$, at which a well developed state has been built up. Contours of the spanwise vorticity, $\omega_3$, in the plane $x_3 = \pi/\beta$, $x_3 = \frac{1}{2}\pi/\beta$ and $x_3 = 0$ and the total vorticity, $\omega_t = (\omega_1^2 + \omega_2^2 + \omega_3^2)^{1/2}$, in the plane $x_2 = 3.54$ are shown in Fig. 5-17. The development of spikes, together with high shear layer roll up, which is seen in the mean separation region ($x_1 = 150 - 195$), forms lambda vortices. In the $x_2$ plane (Fig. 5-17d), two lambda vortices are noticed with the second much stronger than the first, implying that the vorticity gradient is very high. The strong second vortex stretches and bursts at the mean reattachment point, giving rise to small vortex structures, which are characteristic for a turbulent field. Note that the
Laminar separation and transition

Streamwise extent needed for the laminar flow at the separation point to become turbulent is very short, less than $50\ \delta_i$ (inflow displacement thickness). For zero pressure gradient flow, this extent is much longer, through a gradual development of several lambda vortices and laminar streaks. There are low speed and high speed streaks. In the low speed streaks the velocity is lower than the mean velocity in a horizontal plane parallel to the wall, while the apposite is true for the high speed streaks. The low speed streaks structure in the plane $x_2 = 0.814$ (log law region) in Fig. 5-18, shows the absence of laminar-like streaks, which are characterized by long stretched, smooth structures and commonly appear in natural transition. The pattern tends to be more or less isotropic from the reattachment region up to some distance downward ($x_1 \approx 180 - 240$), and the streaks become turbulent-like in their excursion further downstream. They are shorter and exhibit more irregular structures than laminar streaks.

This resembles coherent structures in a zero pressure gradient flat plate turbu-

Figure 5-17: Instantaneous spanwise vorticity contours in the plane $x_3 = \pi/\beta$ (valley plane) (a), $x_3 = \frac{1}{2}\pi/\beta$ (b), $x_3 = 0$ (peak/symmetry plane) (c) and instantaneous total vorticity, $\omega_t$, contours in the plane $x_2 = 3.54$ (d).
Figure 5-18: Contours of positive (dotted) and negative (solid) streamwise velocity fluctuations in the plane $x_2 = 0.814$ ($x_2^+ \approx 14$). The solid lines represent low speed streaks. The domain is duplicated in the spanwise direction.

It appears that the region from $x_1 = 200$ to $x_1 = 300$ with its gradually diminishing favorable pressure gradient acts as a recovery region and the flow approaches a zero pressure gradient turbulent flow. The mean flow and statistics in this region, as shown later, reveal more precisely to what extent the zero pressure gradient turbulent state has been recovered. The dynamics in the $x_1$=constant planes is illustrated in Fig. 5-19 by the velocity vectors at three streamwise locations: the first bursting location ($x_1 = 192.32$), a location of high intensity turbulence, i.e. the beginning of the recovery region ($x_1 = 228.14$) and further downstream ($x_1 = 287.84$). The first velocity vector field shows a downward acceleration of fluid with a high velocity (so called 'sweep') just after the bursting of the strongly stretched vortex near the reattachment point. The velocity vectors in the second plane exhibit more chaotic motions than in the third plane, which is also visible in Fig. 5-17, indicating a higher turbulent intensity in the second plane. It should be noticed that the existence of symmetry in the spanwise direction was not exercised in the simulation explicitly as we simulated the whole spanwise extent. It is questionable whether the symmetry is retained with increasing Reynolds number. In Fig. 5-20 instantaneous maximum and minimum components of the vorticity over the spanwise direction are plotted against the streamwise coordinate at the wall and away from it ($x_2 = 3.54$). Generally, the vorticity along the wall is dominated by the negative spanwise component and near the reattachment region it is higher than its value further downstream. The normal component of vorticity at the wall is by definition zero. The high positive and negative values of $\omega_1$ at the reattachment location correspond to the legs of the strongest lambda vortex before bursting. The large positive $\omega_3$ in the separation region corresponds to counter clockwise rotations due to separation ($\partial u_1/\partial x_2 < 0$). Away from the wall, the spanwise vorticity is still dominated by the negative component. In the rear part of the recovery region
(from $x_1 \approx 270$), the strength of vorticity in all components tends to be statistically constant, indicating that the turbulence has roughly recovered and approaches the zero pressure gradient state.

From the above visualizations we see that in spite of the small mean separation bubble (its height is approximately $2 \delta_i^* \approx 270$) and the low Reynolds number, the bubble is able to induce turbulence, albeit not fully developed. For a fully recovered and developed turbulence, a longer domain and a higher Reynolds number are required.

5.4.3 Mean quantities

The mean quantities are obtained by averaging the solution in the spanwise direction and in time:

$$\overline{v} = \frac{1}{T L_3} \int_{t_o}^{t_o + T} \int_{-L_3}^{L_3} v d x_3 d t \quad (5-13)$$
where $v$ is a component of the solution, $L_3 = 2\pi/\beta$ is the spanwise extent of the computational domain and $T$ is the averaging time interval. $t_a$ is sufficiently large that a statistically stationary state has been reached. The boundary layer integral parameters based on the averaged solution (mean flow) are compared with the corresponding semi empirical relations (Fig. 5-21). Under a zero pressure gradient condition, the relations for a laminar boundary layer read ([68])

\begin{align}
\delta^*(x_1) &= 1.721(Re)^{-1/2}(x_1)^{1/2}, \\
\theta(x_1) &= 0.664(Re)^{-1/2}(x_1)^{1/2}, \\
c_f(x_1) &= 0.664(Re)^{-1/2}(x_1)^{-1/2},
\end{align}

(5-14)

and for a turbulent boundary layer,

\begin{align}
\delta^*(x_1) &= 0.0309(Re)^{-1/6}(x_1 - x_t)^{5/6}, \\
\theta(x_1) &= 0.0222(Re)^{-1/6}(x_1 - x_t)^{5/6}, \\
c_f(x_1) &= 0.037(Re)^{-1/6}(x_1 - x_t)^{-1/6},
\end{align}

(5-17)

where $Re$ is the Reynolds number based on the inflow displacement thickness and $x_t$ is a virtual leading edge for the turbulent boundary layer. The turbulent self-similarity relations are based on the power law profile,

\[ \frac{u_1}{U_\infty} = \left( \frac{x_2}{\delta} \right)^n \]

with $n = 9$ as an appropriate choice, which is also used by Ducros et al. [18]. At the inflow boundary, the boundary layer parameters have consequently the precise values of the laminar self-similarity relations (5-14)-(5-16). The shape factor, $H = \theta/\delta^*$, (Fig. 5-21a) initiates with the value 2.6 and rapidly increases due to the adverse pressure gradient and the flow separation. In the recovery region, $H$ goes down to 1.6, which is slightly higher than predicted by the self-similarity relations ($H = 1.4$). However, this value is fair as in practice it depends on $Re^*$ after transition (Ducros et al. in [18]). Spalart reported $H = 1.43$ for $Re^* \approx 2000$ [72] and $H = 1.66$ for $Re^* = 498$ [73]. Our corresponding $Re^*$ equals 580. The shape factor reaches a value of approximately 4 at the separation point which is typical for laminar separation. As for the shape factor, the displacement thickness (Fig. 5-21b) reaches the maximum value within the separation region whereas the momentum thickness (Fig. 5-21c) increases rapidly near the reattachment point. The initial turbulent boundary layer after reattachment is much thicker than the one after a natural transition. This means that the virtual turbulent leading edge in our case may be located upstream of the current flat plate leading edge. We assume that this virtual turbulent leading edge $x_t = -100$, and the corresponding values for $\delta^*, \theta$ and $c_f$ are represented by the dashed lines. This value for $x_t$ is chosen in such a way that the dashed lines are evenly close to $\delta^*$ and $\theta$ in the turbulent regime (exact matching is impossible due to the difference in the shape factor). The self-similarity relation for $c_f$ is quite insensitive to a change in $x_t$. Nevertheless, the skin friction coefficient (Fig. 5-21d)
Figure 5-21: Streamwise evolution of shape factor, displacement thickness, momentum (loss) thickness and skin friction of the mean flow (a, b, c and d, respectively), the present case (solid) is compared with the empirical laws for zero pressure gradient (z.p.g) laminar flow (dashed dotted) and for the z.p.g turbulent case (dashed).

is in fair agreement with the empirical relations in the turbulent regime, in the sense that it is being recovered toward the similarity prediction.

The mean streamwise velocity profiles $u_1(x_2)$ at equidistant $x_1$ stations in Fig. 5-22 show a clear change from a laminar profile at the inflow boundary to turbulent profiles at some distance after reattachment. The $u_1$ profiles in the downstream direction within the recovery region are plotted in Fig. 5-23 - 5-25 using logarithmic scale, together with the corresponding viscous sublayer profile and a turbulent logarithmic law. This law is defined as

$$
\begin{align*}
  u_1^+ &= x_2^+ & \text{for } 0 < x_2^+ < 5 & \text{(viscous sublayer),} \\
  u_1^+ &= 2.5 \ln(x_2^+) + 5.1 & \text{for } 30 < x_2^+ & \text{(logarithmic layer),}
\end{align*}
$$

where

$$
\begin{align*}
  u_\tau &= (\tau_w/\rho_w)^{1/2}, \\
  u_1^+ &= u_1/u_\tau.
\end{align*}
$$

(5-21) (5-22)
\[ x_2^+ = x_2 u_r Re, \]  
(5-25)

with \( \tau_w \) and \( \rho_w \) the wall shear stress and the density at the wall, respectively. An

\[ U_1 = x_2^+, \]

\[ \text{Log Law} \]

\[ u_1^+ = 2.5 \ln(x_2^+) + 5.1. \]

\[ x_1 \]

\[ U_1 + = x_2^+ \]

\[ \text{Log Law} \]

effect of the separation bubble on the turbulence downstream is that the \( u_1 \) profile just after reattachment drops substantially below the logarithmic law as seen in Fig. 5-23. The increasing \( u_1 \) far from the wall at the first location, \( x_1 = 205.45 \), is due to the still existing influence of the separation upstream (\( u_r \) just after reattachment is low and has not reached the turbulent level yet). The low values within the logarithmic layer resembles a similar effect of surface roughness ([68]). Proceeding downstream, however, the profiles in the logarithmic regime increase gradually until the most downstream profile recorded virtually coincides with the logarithmic law (Fig. 5-25). This gradual change of the streamwise velocity profiles clearly shows the recovery process toward the zero pressure gradient state.

The resemblance of the logarithmic profiles in the reattachment region to those affected by surface roughness suggests that these logarithmic profiles in the reat-
Figure 5-24: As in Fig. 5-23 with $x_1 = 253.21$ (□), 265.15 (*) and 277.09 (▽).

Figure 5-25: As in Fig. 5-23 with $x_1 = 289.03$ (□), 300.97 (*) and 309.91 (▽).

tachment region can be expressed as

$$u_1^+ = 2.5 \ln(x_2^+) + 5.1 + D.$$  \hfill (5-26)

In the case of surface roughness, $D$ is a function of the roughness size. Analogously, in the case of turbulence induced by a laminar separation bubble we could relate the quantity $D$ to the properties of the separation bubble, such as the bubble size, the strength of the adverse pressure gradient, the momentum thickness at the separation point or the level of upstream disturbances. To our knowledge, such a relation is not known yet. To conduct such a parameter study, however, DNS is not practical as a large number of simulations are required. Here, we only conjecture the analogy between turbulence on a rough surface and just after the reattachment of a separated laminar flow. This conjecture is supported by the fact that the skin friction coefficient just after reattachment increases to a level above the zero pressure gradient turbulence level along a smooth wall, which is similar to the effect of surface roughness.
5.4.4 Fluctuations

The velocity fluctuations are presented in Fig. 5-26, Fig. 5-27 and Fig. 5-28 for the streamwise, the normal and the spanwise direction, respectively, using the root mean square values. The r.m.s. operator is defined as

\[
(v)_{\text{rms}} = \left( \frac{1}{T \int T} \int_{t_0}^{t+T} \int_{-L_3}^{L_3} v^2 dx_3 dt \right)^{1/2},
\]

where \( v \) is the fluctuation quantity, i.e. the instantaneous local value minus the corresponding local mean value. The r.m.s. values are initially very high just downstream the reattachment point, especially in the region near the wall \((x_2/\delta < 0.6)\). The high level fluctuations near the reattachment point are in accordance with the previous observation of vorticity fluctuations and the velocity vectors in \(x_1\) planes. Proceeding further downstream, the r.m.s. level of the near wall fluctuations decreases and the quantities in the last three streamwise locations \((x_1 = 289.03, 300.97 \text{ and } 309.91)\) more or less collapse onto a single profile.

We check to what extent these profiles resemble r.m.s. profiles of zero pressure gradient turbulence by a comparison with other numerical and experimental data. We compare the r.m.s. velocity fluctuations in two far downstream stations with

![Figure 5-26: R.m.s. streamwise velocity fluctuations corresponding to the \(x_1\) positions in Fig. 5-23, Fig. 5-24 and Fig. 5-25 (a, b and c, respectively); the sequence of lines with increasing streamwise coordinate is solid, dashed, dotted (in a) and dashed dotted.](image-url)
the experimental data of Eckelmann (channel flow, [20]) and the DNS result of Spalart (boundary layer, [72]) in Fig. 5-29 and Fig. 5-30, respectively. In general, the qualitative agreement is quite good. In both figures, the spanwise component is consistently lower than the references. The under-estimation of the spanwise fluctuations is often observed in numerical simulations (Ducros in [18], Antonia et al. in [3]), which suggests the need for a higher resolution in this direction. The agreement with Eckelmann’s data is slightly better than with Spalart’s, which is probably due to the compatibility of the local Reynolds number. In Eckelmann’s experiment the Reynolds number based on $u_r$ is $Re_r = 209$ (in our case $Re_r = 240$), whereas in Spalart’s DNS the Reynolds number based on the local displacement thickness is $R_{\delta_1} = 2000$ (our $R_{\delta_1} = 600$). It should be noted that since we cannot
Figure 5-29: R.m.s. velocity fluctuations $u_i/u_\tau$ versus $x_2/\delta$ (lines) at $x_1 = 300.38$ and $309.32$ compared to experimental data by Eckelmann (1970) in turbulent channel flow for $Re_\tau = 209$ (symbols) (data from our simulation are equivalent to $Re_\tau = 240$; streamwise (solid, $\Delta$), normal (dashed, $*$) and spanwise (dashed dotted, $\Box$).

Figure 5-30: The same fluctuations as in Fig. 5-29 compared to the DNS of Spalart (1988) for $R^* = 2000$ (the present data are equivalent to $R^* = 600$).

carry out a resolution study - due to cost reasons - there remains some uncertainty about this aspect. In addition, a longer domain computation is recommended to remove the uncertainty concerning the recovery state.

The Reynolds stresses $u'_i u'_j$ also show the same trend as the velocity fluctuations (Fig. 5-31a). Proceeding downstream from the reattachment region the initially high level of Reynolds stress near the wall decreases, while that in the region $x_2/\delta > 0.6$ slightly increases. Comparing the Reynolds stress $(u'_1 u'_2)_{\text{rms}}$ at the station $x_1 = 309.91 (Re_\tau = 240)$ with the DNS result of Rai & Moin for $Re_\tau \approx 650$ (boundary layer, [61]) in Fig. 5-31b yields an under-estimation in the whole range. The data from Eckelmann [20] (channel flow) for $Re_\tau \approx 208$ are also incorporated in order to illustrate the effect of Reynolds number. Clearly, a lower Reynolds number results in a lower level of Reynolds stress. This may explain why our result is closer to the result of Eckelmann, at least in the region close to the wall. The correlation
coefficient, $\frac{-u_1'u_2'/(u_{1rms}u_{2rms})}{},$ however, agrees well with the data of Sabot & Comte-Bellot [67] in spite of the high Reynolds number in their flow ($Re_\tau \approx 37000$), as can be seen in Fig. 5-32. This suggests that the correlation coefficient is less sensitive to the Reynolds number than are the Reynolds stresses. This insensitivity of the correlation coefficient to the Reynolds number is also observed by Kim, Moin & Moser in [39].

Finally, we computed a measure of three dimensionality, $E'$, as a function of $R = (Rx_1)^{\frac{1}{2}}$, given by

$$E'(R) = \left[ \frac{1}{T} \int_{t_0}^{t_0+T} \int_0^{L_2} \int_{-\frac{1}{2}L_3}^{\frac{1}{2}L_3} Edx_3dx_2dt \right]^{\frac{1}{2}},$$  \hspace{1cm} (5-28)
with

\[ E = \| \mathbf{u}(x, t) - \frac{1}{L_3} \int_{-\frac{L_3}{2}}^{\frac{L_3}{2}} \mathbf{u}(x, t) \, dx \|^2 \]  

(5-29)

where \( \mathbf{u} = (u_1, u_2, u_3) \). The result is plotted in Fig. 5-33 together with an approximation of three-dimensionality amplification in a natural transition boundary layer, representing an exponential growth rate \( d[\ln(E')] / dR \approx 2 \times 10^{-2} \). The corresponding growth rate in the present transition is more than five times higher. This means that, computationally, we reduced the price of the transition simulation to only 20% of that needed for the natural transition, provided that the same grid resolution is required. In view of the development of numerical methods for spatial DNS, this is an important benefit.

\[ \begin{array}{c}
\begin{array}{c}
\text{Figure 5-33: Amplification curve of } E' \text{ given by Eq. 5-28 (solid). The dashed} \\
\text{line is representative for the amplification in the case of zero pressure gradient.}
\end{array}
\end{array} \]

5.5 Conclusion

From the two dimensional simulations we conclude that the results of a low Mach-number laminar separation bubble flow agree well with incompressible-flow results. The condition at the separation point is quantitatively consistent with approximate theories and the features of the flow subjected to different adverse pressure gradients confirm the incompressible results reported in the literature. Specifically, under a weak adverse pressure gradient an unperturbed laminar separation bubble flow exhibits a nearly steady character, while under a strong adverse pressure gradient it is steady up to the reattachment point, after which a strongly unsteady behavior in the form of self-excited vortex shedding is found. Under a weak adverse pressure gradient, an unperturbed shear layer exhibits no breakdown. Based on these agreements, we conclude that the compressible simulation method performs satisfactorily.
We further explored the physical features of the flow, focusing on the effect of different freestream boundary conditions, the magnitude of the adverse pressure gradient and the presence of small upstream disturbances. We investigated two different realizations of a freestream adverse pressure gradient: prescribing the pressure and using suction and blowing. The time-averaged results which we derived by using the two boundary conditions are the same if the prescribed pressure corresponds exactly to the prescribed normal velocity. This equivalence is independent of the pressure gradient. The potential-flow assumption produces a normal velocity distribution which is comparable to the normal velocity resulting from the DNS. The results in other quantities are qualitatively the same.

An increase of the adverse pressure gradient results in an earlier occurrence of separation and shear-layer breakdown, a higher separation-bubble, a stronger reverse flow and a larger disturbance amplitude.

If small perturbations are imposed at the inflow boundary, vortex shedding occurs at the end of the separation region for both low and high pressure gradients. The resulting time-averaged bubble is shorter than without these inflow perturbations. The difference in length depends on the applied adverse pressure gradient. Furthermore, under the same adverse pressure gradient forced vortex shedding (perturbed) occurs earlier than self-excited (unperturbed) vortex shedding and the shedded vortices in the first case are stronger. Upstream disturbances also affect the Strouhal number of the vortex shedding.

The three dimensional simulation shows that the numerical methods developed for two dimensional applications can directly be applied to transition and turbulent flow simulations. The required computing time of about 300 CPU hours is, however, very high, considering the low Reynolds number and the small computational domain used. The symmetry property of the fluctuations makes the use of symmetry conditions in the spanwise direction possible, which could halve the computational requirements. For higher Reynolds number simulations, however, the periodic condition should first be applied to check whether the symmetry property still holds. Compared to the simulation of a natural transition, the simulation of a separation bubble transition required only a short streamwise extent for the laminar flow to become turbulent, and hence offers a relatively cheap way to test the numerical method for spatial DNS.

The region downstream of the reattachment point exhibits a recovery region in which the flow approaches the zero pressure gradient wall turbulence. The reattachment region is characterized by high amplitude fluctuations and mean streamwise velocity profiles which resemble profiles affected by surface roughness. At the end of the recovery region, near the beginning of the buffer domain, the mean profiles and the r.m.s. of fluctuations are in fair agreement with zero pressure gradient wall turbulence profiles. The large deviation in the spanwise fluctuations, however, indicates the need of a grid resolution study, especially in the spanwise direction. To confirm the results of comparison at the end of the recovery region, a longer domain is also recommended. Under the same suction and blowing, the resulting inviscid pressure distribution depends on the condition of inflow perturbations. Hence, the
pressure prescription boundary condition along the upper boundary, can be used instead of the suction and blowing condition to accelerate the recovery process as a desired pressure distribution can be imposed at the upper boundary.

The extension of the numerical method to a two dimensional supersonic laminar separation-bubble flow containing stationary as well as instationary shocks is presented in the next chapter.
Chapter 6

Shock boundary-layer interaction under blowing and suction

In Chapter 5, the developed numerical methods have successfully been applied to flows involving unsteady separation. In this chapter the complexity of the flow increases by the presence of strong compressibility effects in the flow, i.e. shocks. The present application therefore forms a more critical test case for the numerical methods. The presence of shocks together with the boundary layer along the wall results in an interaction between them. This kind of viscous-inviscid interaction has been a subject of intensive studies for many years. The reason is that shock boundary-layer interactions (SBLI) occur frequently in high speed engineering problems and may influence the aerodynamic and structural performance considerably.

Since the early investigations by Ferri (1940), Ackeret, Feldmann & Rott (1947), and Liepmann, Roshko & Dhawan (1952), which reveal the difference between the interaction for laminar and turbulent boundary layers, many experimental, theoretical and numerical studies of SBLI appear in the literature. Reviews about this topics can be found in Adamson & Messiter (1980) and Delery & Marvin (1986). These studies, however, concern mostly steady SBLI. Although recently investigations on unsteady SBLI have been performed, the origin of the unsteadiness is restricted to the boundary layer (upstream turbulence/disturbances or vortex shedding), whereas the main, inviscid shock remains steady. This type of unsteady interaction has formed the research subject of a.o. Degrez (1981), Dolling (1993) and Loth & Matthys (1995). In practical situations, unsteadiness of both the inviscid shock and the boundary layer underneath also appear, such as on maneuvering or fluttering control surfaces of high speed vehicles and in propulsion systems. This kind of unsteady interaction may form a pressure load which is more critical for the material structure than the pressure load caused by the viscous unsteadiness alone.

In the present study, we show by performing direct numerical simulations that an unsteady interaction can occur under suction and blowing boundary conditions.
in which both the inviscid shocks and the boundary layer oscillate regularly or irregularly in time [89]. We study the interaction between the generated shock, which can be single or multiple, depending on the physical parameters, and the boundary layer on the lower wall which is adiabatic.

First, we validate the numerical method, especially the upper boundary conditions for flows containing a shock. In the previous chapter, the good performance of the suction and blowing boundary condition is confirmed for shockless flows. The validation is carried out for a well known steady SBLI for which experimental data and other numerical results are available. Apart from validation of the numerical method for flows with shocks, the objective of the present study is the description of some interesting properties of SBLI under suction and blowing. As for this unsteady SBLI no experimental data is available, we rely on a numerical investigation. In addition, the complexity of the SBLI under suction and blowing prevents a complete and rigorous explanation of the observed phenomena at this moment. To achieve this goal, more experimental and numerical work should be performed.

This chapter is organized as follows. In Section 1, we describe the numerical method and boundary conditions. In Section 2 the validation for steady SBLI is presented. In Section 3 we discuss the unsteady SBLI under suction and blowing and we summarize our findings in Section 4.

6.1 Numerical method and boundary conditions

The fourth order central and third order upwind schemes described in Chapter 3 are employed. The first is suitable for flows which do not contain discontinuities, while the second is especially suited for flows containing shocks. The time integration is performed with the compact storage, explicit Runge-Kutta method.

The boundary condition for the lower wall is no-slip adiabatic, which implies that the velocity components vanish while the temperature, density and pressure are calculated by solving the conservation equations of mass and energy at the wall. At the subsonic part of the inflow and outflow boundaries we use the best combination extracted from Chapter 2, namely an extrapolation technique at the inflow and a quasi 2-D non-reflecting characteristic condition at the outflow boundary. At the supersonic part of the inflow boundary all dependent variables are imposed according to the laminar boundary layer (compressible Blasius) profiles. At the supersonic part of the outflow boundary all dependent variables are extrapolated from the interior domain. If the flow is instationary, the buffer domain technique is applied instead of the outflow boundary condition. The buffer domain calms the flow gradually into a laminar flow to prevent reflections of upstream disturbances at the outflow boundary.

Special attention is given to the treatment of the upper boundary. The suction and blowing condition along the upper boundary is realized by prescribing the (subsonic) normal velocity, which is positive for suction and negative for blowing. In the case of unsteady flow, the external flow is in general not known. This can lead to
a delicate situation. In the region of (subsonic) negative normal velocity (blowing), theoretically three variables should be prescribed corresponding to three incoming characteristic waves. Thus, besides the normal velocity, two other variables should be imposed. However, this cannot be done since we have no other information about the external condition than the normal velocity. There are two alternatives. Let only the normal velocity be prescribed and the other variables be extrapolated from the interior solution irrespective of the sign of the normal velocity. This leads to an underprescription of boundary conditions (next denoted as B1). Another choice is to apply the non-reflecting principle to the other two incoming waves, as described in the discussion of the subsonic freestream quasi 2-D characteristic method in Chapter 2 (next denoted as B2). First, we validate these boundary conditions in a well known steady SBLI where the external solution along the upper boundary is known analytically. As a reference, a boundary condition based on the analytical solution is incorporated as well (B3). This reference boundary condition makes use of the extrapolation technique. Specifically, in the region of positive normal velocity, one analytical variable is imposed, for which we choose the temperature, and three variables, namely the velocity components $u_1$ and $u_2$ and the pressure are extrapolated from the interior domain. In the region of negative velocity the temperature and the velocity components are prescribed and the pressure is extrapolated from the interior domain to the boundary. A sketch of the three boundary conditions is given in Fig. 6-1.

**Figure 6-1:** Three different upper boundary conditions under study, B1: underprescription, B2: quasi 2D characteristic, B3: prescription of analytical solution (extrapolation technique). Bold arrows denote prescribed variables while thin arrows denote extrapolations from the interior domain.
6.2 Validation: steady SBLI

For the validation of the numerical method we perform a simulation of steady SBLI. The experimental data of this simulation is known from an experiment by Hakkinen et al. [32]. In addition, corresponding numerical results by Katzer [38] are available. The shock is generated by a wedge or corner and impinges onto a flat plate under a certain angle. For a certain incoming Mach number and a shock angle $\sigma$ (see Fig. 6-2) the relations between the flow properties on both sides of the shock can be derived from gas dynamics [49].

\[
\frac{p_2}{p_1} = 1 + \frac{2\gamma}{\gamma+1}(M^2_1 \sin^2 \sigma - 1) \quad (6-1)
\]

\[
\frac{\rho_2}{\rho_1} = \frac{(\gamma+1)M^2_1 \sin^2 \sigma}{2 + (\gamma-1)M^2_1 \sin^2 \sigma} \quad (6-2)
\]

\[
\frac{T_2}{T_1} = \frac{p_2 \rho_1}{p_1 \rho_2} \quad (6-3)
\]

\[
\frac{u_{n2}}{u_{n1}} = \frac{\rho_1}{\rho_2} \quad (6-4)
\]

Subscripts 1 and 2 denote the location in front of and behind the shock, respectively, while subscript $n$ denotes the direction normal to the shock. These properties are useful as the external information for the upper boundary condition. The analytical quantities along the upper boundary form step functions, with a jump in $p, \rho,$ and $T$ and a drop in the velocity component normal to the shock. The tangential velocity component remains unchanged. The isobars of the flow field are shown in Fig. 6-3, calculated with the MUSCL scheme in both directions as the shock is oblique. Two cases of shock strengths are simulated. In the first case, the Mach number is 2.0 and the wedge angle is such that the shock angle is 32.58°, which yields a ratio of $p_3/p_1$ of 1.4. The subscript 3 and 1 denote the inviscid area defined in Fig. 6-2. The Reynolds number $Re_o = 2.96 \times 10^5$, based on the distance between the flat plate leading edge to the shock impingement point, which is equivalent to $Re = 950$, based on the inflow displacement thickness. In the second case $Re_o = 2.87 \times 10^5$ and the ratio $p_3/p_1 = 1.25$, which results from a shock angle of 31.67°. The convergence criterion for the steady state is

\[
R(p) = 2 \left( \frac{1}{N} \sum_{i=1}^{N} \left( \frac{p^{t+\Delta t}(i) - p^{t}(i)}{\Delta t} \right)^2 \right)^{\frac{1}{2}} < 10^{-6}, \quad (6-5)
\]

where $N$ is the number of grid points on the wall. For the first case we investigate the appropriate grid stretching and resolution, the influence of the size of the computational domain, the performance of the upwind and central discretization schemes and the performance of a number of upper boundary conditions. In addition, a comparison with the above mentioned reference results is made. These aspects are discussed in the following subsections.
Figure 6-2: Impinging steady shock generated by a wedge. The computational domain is bounded by the dashed line.

Figure 6-3: Mach-field of steady SBLI using upwind scheme in both directions. Shock is induced by wedge/corner.

6.2.1 Grid arrangement and location of boundaries

The grid is uniform in the $x_1$-direction and stretched in the $x_2$-direction. Two stretching functions are compared: rational and exponential. The normal grid using the rational function is presented by

$$x_2 = L_2 S_r y / (1 + S_r - y)$$  \hspace{1cm} (6-6)

where $S_r$ is a stretching parameter, $L_2$ the height of the domain and $0 \leq y \leq 1$, uniformly distributed. Using the exponential function, the grid is presented by

$$x_2 = L_2 \frac{1 - e^{(S_e y)}}{1 - e^{S_e}}$$  \hspace{1cm} (6-7)
where $S_e$ is another stretching parameter, and the other quantities are the same as in the rational function. As a reference, a lengthscale ratio given by the triple-deck theory is used. Near the wall, it suggests

$$\Delta x_2 \leq Re_s^{-\frac{1}{2}} \delta_o^*,$$

(6-8)

and

$$\Delta x_2 \approx Re_s^{-\frac{1}{2}} \Delta x_1,$$

(6-9)

where $Re_s$ is the Reynolds number based on the distance of the estimated separation point from the leading edge and $\delta_o^*$ is the displacement thickness at the impingement point. Based on the results of Hakkinen et al. and Katzer, we take $Re_s \approx 2.3 \times 10^5$ which implies that the minimum aspect ratio $\Delta x_2/\Delta x_1 \approx 0.0457$ and the minimum normal spacing $\Delta x_2/\delta_o^* \leq 0.2137$. The domain dimension is $400 \times 115$ based on the inflow displacement thickness. Using a grid resolution of $151 \times 128$, which is comparable to Katzer’s grid ($151 \times 101$ for a comparable domain dimension), the rational and exponential functions with various stretching parameters yield grid properties as presented in Table 6-1.

All the grids satisfy the requirement of $\Delta x_2/\delta_o^* \leq 0.2137$, but only two grids are close to the requirement $(\Delta x_2/\Delta x_1)_{\text{min}} \approx 0.0457$, namely the rational grid with $S_r = 0.2$ and the exponential grid with $S_e = 3$. Both also have comparable minimum normal grid spacing. However, the latter is preferred due to its smaller ratio of maximum and minimum normal grid spacing, which indicates that the grid is more uniform far from the wall. To achieve the same value of maximum aspect ratio, a rational stretching would have needed more points in the normal direction.

Next, the grid density is examined. Varying the number of grid points in the streamwise direction to 101 and 192 yields almost the same result for the skin friction coefficient, $c_f$. A change of the number of grid points in the normal direction to 64, 128 and 192 yields a deviation of 8.5% for 64 points and of 1.4% for 128 points compared to the results with 192 points. The sensitivity of the result to the resolution in the normal direction is the reason for employing more points in this direction than used by Katzer.

<table>
<thead>
<tr>
<th>Stretching</th>
<th>Parameter</th>
<th>$(\Delta x_2)_{\text{min}}/\delta_o^*$</th>
<th>$(\Delta x_2)<em>{\text{max}}/(\Delta x_2)</em>{\text{min}}$</th>
<th>$(\Delta x_2/\Delta x_1)_{\text{min}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rational</td>
<td>$S_r = 0.2$</td>
<td>0.079868</td>
<td>31.9270</td>
<td>0.0588</td>
</tr>
<tr>
<td></td>
<td>$S_r = 0.4$</td>
<td>0.136789</td>
<td>11.4991</td>
<td>0.1006</td>
</tr>
<tr>
<td></td>
<td>$S_r = 0.6$</td>
<td>0.179410</td>
<td>6.8081</td>
<td>0.1320</td>
</tr>
<tr>
<td></td>
<td>$S_r = 0.8$</td>
<td>0.212519</td>
<td>4.8961</td>
<td>0.1563</td>
</tr>
<tr>
<td>Exponential</td>
<td>$S_e = 4$</td>
<td>0.036091</td>
<td>51.2908</td>
<td>0.0265</td>
</tr>
<tr>
<td></td>
<td>$S_e = 3$</td>
<td>0.075719</td>
<td>19.1658</td>
<td>0.0557</td>
</tr>
<tr>
<td></td>
<td>$S_e = 2$</td>
<td>0.150204</td>
<td>7.1617</td>
<td>0.1105</td>
</tr>
</tbody>
</table>

Table 6-1: Grid property resulting from rational and exponential function in the $x_2$-direction.
The influence of the boundaries is examined by shifting their position by about 17% and, as can be seen in the plot of the displacement thickness and separation bubble in Fig. 6-4 and Fig. 6-5, it was found to be small. The effects are appreciable if we lower the upper boundary, as illustrated by the lower peak of the displacement thickness in Fig. 6-5. This is caused by unavoidable reflections of expansion waves at the upper boundary near the outflow.

### 6.2.2 Spatial discretization, upper boundary conditions and comparison with reference results

From our experience, a central discretization can also be used for flows containing shocks if the shocks are not too strong and the resolution is adequate. The use of the fourth order central scheme is preferred, since it is more accurate and cheaper than the third order upwind scheme. Therefore we examine combinations of both schemes...
Comparing the third order upwind scheme (MUSCL) in both directions with a combination of upwind in the streamwise and central in the normal direction, we observe only small differences in the wall pressure and skin friction coefficient as shown in Fig. 6-6. However, the latter produces wiggles in the solution as illustrated by the Mach-field in Fig. 6-10 (compare to Fig. 6-3). Fig. 6-6 shows at the same time that the upper boundary conditions B2 and B3 produce virtually the same results.

Next we examine the performance of the suction and blowing based on the underprescribed boundary condition B1 and the one based on the non-reflecting characteristics by comparing the calculated temperature on the upper boundary with the theoretical temperature, as shown in Fig. 6-11. The upwind scheme is applied

---

**Figure 6-6:** Wall pressure (above) and skin friction coef. (below) for $p_3/p_1 = 1.4$: B3 upwind (solid), B2 upwind (dashed dotted), B2 hybrid (dashed), Katzer’s numerical result (*) and Hakkinen et al.’s experiment (o).
Figure 6-7: Wall pressure (above) and skin friction coefficient (below) for $p_3/p_1 = 1.25$: B3 upwind (solid) and Hakkinen’s experiment (o).

in both directions. The increasing temperature near the outflow is caused by reflections of compression waves at the upper boundary (see Fig. 6-3). B1 exhibits a large overshoot behind the shock and increases toward the outflow boundary. B2, in contrast, yields a much better freestream temperature. This difference is also manifest in the other dependent variables. Fig. 6-11 also illustrates that the pressure jump across the shock at the upper boundary in the case of B2 is not discontinuous compared to the grid spacing. This results in a small shift in the wall pressure and shear stress coefficient of B2 and B3 (Fig. 6-6). In B3 the pressure jump at the upper boundary is discontinuous due to the prescription of analytical variables. The jump of B2 sharpens if we employ the central scheme in the normal direction. This results in a better agreement in the wall pressure and wall shear stress coefficient resulting from B3 and B2 central, as shown in Fig. 6-6. From this we conclude that the non-reflecting characteristic blowing boundary condition (B2) yields a comparable result to the analytical boundary condition (B3). Furthermore, if the shock angle is close to 90° a central scheme is preferred in the normal direction, otherwise
an upwind scheme is necessary to avoid wiggles in this direction.

Wall pressure, shear stress coefficient and streamwise velocity profiles in Figure 6-6 - 6-9 show that the present result agrees well for both cases of shock strength with the numerical result of Katzer (1989) and is comparable to the measurements by Hakkinen et al.. In the plots of the skin friction coefficient also the result for zero pressure gradient is included to show that the presence of the shock has only a local effect. Although Katzer’s results are not taken along in all figures, it is confirmed that the overall agreement between his results and the present results is excellent. Katzer solved the 2D Navier-Stokes equations by using the explicit time-split Mac-Cormack scheme where a control of artificial dissipation is employed as a tuning parameter. Although the agreement in the wall pressure is good, there is a discrepancy in the wall shear stress coefficient in the case of the stronger shock, where the separation region is quite large. Katzer addresses the origin of the difference between

Figure 6-8: \( u_1 \) profiles for \( p_3/p_1 = 1.40 \): B3 upwind (solid) and Hakkinen’s experiment (o).

Figure 6-9: \( u_1 \) profiles for \( p_3/p_1 = 1.25 \): B3 upwind (solid) and Hakkinen’s experiment (o).
the calculated and the measured skin friction to the fact that in the experimental study the wall shear stress has been measured using a Stanton probe contacting the wall. This may influence the length of the separation bubble. In addition, there is a three dimensional effect in the experiment due to the presence of side walls, which is absent in the numerical simulation.

6.3 Unsteady SBLI under suction and blowing

Based on the suction and blowing characteristic boundary condition presented above, it is now possible to prescribe an arbitrary normal velocity distribution along the upper boundary instead of a step function, as in the previous section. To study the influence of this distribution on the flow below we define a consecutive suction and blowing as sketched in Fig. 6-12. Parameters characterizing the suction and blowing are the amplitude $a$, the distance between suction and blowing $d$, its width $w$ and the angle $\phi$. As a physical interpretation of this boundary condition, we can imagine an array of sinks (suction) and sources (blowing) along the upper boundary.
No boundary layer develops along the upper boundary. Such controlled suction and blowing is difficult to realize in a laboratory, but may occur in practical situations such as inside propulsion systems. We can also interpret differently by thinking of a freeslip upper wall (for instance a wall covered by a thin layer of liquid) with its inclination varying in the streamwise direction, as outlined below. In contrast to the controlled suction and blowing, the latter approximate model can be conducted as a laboratorium experiment in an easier way. Besides the freeslip assumption, the approximation in the latter interpretation also implies that a possible time dependency of the upper wall shape is neglected.

6.3.1 Reference configuration

As a reference simulation, we use the following configuration. The height and length of the domain are 30 and 500, respectively, $M=1.3$, $Re=500$ based on the inflow displacement thickness. The suction and blowing parameters are $a=0.12$, $w=300$, $d=22.5$ and $1/\tan(\phi) = du_2/dx_1 = -0.0053$. The grid resolution is $191 \times 64$ in the $x_1$ and $x_2$-direction, respectively, which is found to be sufficient for the present qualitative study as illustrated below. The following is a brief enumeration of some results. By associating suction and blowing with channel divergence and convergence, a supersonic flow is caused to expand and compress, respectively. Applying only suction (the amplitude of blowing is suppressed to zero) does not cause compression and no shock wave is formed, as the exit pressure is free. The flow is isentropic and steady in this case. On the other hand, applying only blowing (compression) generates a steady shock and a separation region upstream of the shock. A consecutive suction and blowing with their distance $d=0$ results in the occurrence of steady shocks. The shock strength is affected by the angle $\phi$. The smaller $\phi$ the stronger the shock. An interesting phenomenon happens when we increase the distance $d$ up to the reference case value, $d=22.5$. The generated shocks then become unsteady, in that they oscillate spatially as well as temporally. This unsteadiness persists with further increasing $d$ and at the same time the extent of the separation region increases. However, if the distance between suction and blowing is extremely large, their interaction decays and the flow is characterized by a local suction and
blowing separately, as described above. Furthermore, we observe that decreasing
the width $w$ of the suction and blowing region suppresses the shock unsteadiness
and decreases the averaged shock strength. This occurs since the distance between
and the amount of suction and blowing also decrease.

A directional shock sensor, defined as

$$s_i(t) = \max \left| \frac{\partial p}{\partial x_i}(t) \right| \frac{\Delta x_i}{\rho_\infty},$$

where $\Delta x_i$ is the grid spacing in the $x_i$-direction, is used as indicator for the shock
strength and the unsteadiness. As an illustration, the shock sensor corresponding
to the reference case is shown in Fig. 6-13. It functions as a numerical as well as

\begin{figure}[h]
\centering
\includegraphics[width=0.8\textwidth]{shock_sensor.png}
\caption{Shock sensor of reference case with threshold value 0.2 and statistically stationary state beginning at $t \approx 5000$.}
\end{figure}

\begin{figure}[h]
\centering
\includegraphics[width=0.8\textwidth]{skin_friction.png}
\caption{Skin friction coefficient of time averaged flow resulting from the resolution $115 \times 32$ (dashed), $191 \times 64$ (solid) and $247 \times 96$ (dashed dotted).}
\end{figure}

physical indicator. As a numerical indicator, the shock sensor decides whether the
upwind or central scheme is used. If $s_i$ is higher than a threshold value, the upwind
scheme is activated in the $x_i$ direction, otherwise the central scheme is used. The
threshold value of 0.2 is found to be appropriate. Using this hybrid scheme, we
avoid wiggles when a shock occurs and maintain high accuracy when the flow is shockless. The data sampling, such as time averaging, is taken after a statistically stationary state is reached. As a physical indicator, the time averaged \( s_i \) measures the shock strength while the shock sensor bandwidth \( \Delta s_i \), defined as the difference between the maximum and the minimum \( s_i \) during the data sampling, measures the level of unsteadiness (\( \Delta s_i = 0 \) means that the shock is steady). The accuracy of the above grid resolution is examined by comparing the skin friction coefficient of the time averaged flow resulting from the resolution \( 115 \times 32, 191 \times 64 \), and \( 247 \times 96 \), as shown in Fig. 6-14. The deviation of the grid \( 191 \times 64 \) from the finer grid is negligible in comparison with the deviation of the coarse grid, which shows that the resolution is sufficient for the present qualitative study.

### 6.3.2 Influence of physical parameters

Next, we consider a consecutive suction and blowing, with a non-zero distance \( d \) so that shock unsteadiness may occur. The suction and blowing can be associated with an upper wall inclination following from the definition of streamline \( \frac{dx_2}{dx_1} = \frac{u_2}{u_1} \). A typical shape of this imaginary upper wall can be seen in Fig. 6-15. The change of the wall shape in time due to the flow unsteadiness is in general found to be small, so that the interpretation of a freeslip, rigid upper wall is justified. Depending on a number of physical parameters, single or multiple shocks are generated and the unsteadiness can enhance or decay. We study the influence of the domain height, Mach number, Reynolds number and the amplitude of suction & blowing by performing a series of simulations. The parameters have been varied independently using the values in the reference configuration as the base variables. Furthermore, the resulting adverse pressure gradient is sufficiently strong, even in the case of \( s_i < 0.2 \), that separation regions always exist. We investigate the dependence of the shock strength and unsteadiness, of the occurrence of vortex shedding and of the properties of the separation region on the above parameters.

![Figure 6-15: Imaginary upper wall, representing a streamline of the potential flow along the upper boundary.](image)
SHOCK UNSTEADINESS, AVERAGE SHOCK STRENGTH AND VORTEX SHEDDING

We find that if the upper boundary is sufficiently high (higher than 60), the occurring shocks are due to compression corners of the associated upper wall. The influence of the boundary-layer thickening on the lower wall is small. Using the above mentioned reference Reynolds and Mach number, we also observe a spontaneous vortex shedding from the time averaged reattachment region, which is typically located under the blowing region (the vortex shedding may disappear at lower Reynolds numbers). This shedding of vortices causes a small fluctuation of the shocks. If, in contrast, the position of the freestream boundary is close to the wall, the blowing suppresses the occurrence of vortex shedding. In addition, the thickening of the boundary layer in the interaction region changes the effective cross section of the domain significantly and the inviscid part of the flow domain can be considered as a Laval nozzle. This change in the flow features with decreasing height of the upper boundary is illustrated by their instantaneous Mach-field and vortex contours in Fig. 6-16 and Fig. 6-17, respectively. It was further observed in the case of a low upper boundary that due to consecutive compressions and expansions in the nozzle, unsteady shocks can occur, as happens in the reference case. A part of the time development of the Mach-field for the reference case is shown in Fig. 6-18. The

Figure 6-16: Mach-field of the flow with corresponding to domain height 120, 90, 60 and 40, from above to below. Dashed lines denote subsonic regions.
unsteadiness of this flow can be understood as follows. The presence of a shock results in a change in boundary layer thickness and the geometry of the associated upper wall. This changes the effective cross section of the nozzle, which in turn causes the shocks to migrate to other streamwise locations, and so on. This process repeats irregularly in time, forming an unsteady shock boundary-layer interaction. It is clear that this type of unsteadiness, which is caused by the continuous change of the channel effective cross section, is different from the shock fluctuation due to vortex shedding in the case of high upper boundary.

The unsteady interaction described above is localized under the suction and blowing region. Increasing the duration of the simulation considerably we observe that this irregular motion does not decay in time and evolves into a statistically stationary behavior. Varying the length of the computational domain, we confirm
that the phenomenon is not caused by a coupling between the outflow and inflow boundaries.

The influence of the parameters on the shock strength and unsteadiness is shown in Fig. 6-19 and 6-20, respectively. A further decrease in the distance between the upper boundary and the lower wall removes the unsteadiness and even the shocks (shock strength lower than 0.2). This is caused by the fact that the boundary layer thickening tends to be parallel to the inclination of the associated upper wall, or in other words, tends to a constant nozzle cross section. The small increase of the shock bandwidth above height 40 is caused by the occurrence of vortex shedding from the separation region. The region of shock unsteadiness and shock strength is also bounded on both sides by the Mach number. Up to $M = 1.7$ there is a subsonic region behind the shocks. At $M = 1.8$ and higher, the compressions are so weak that no subsonic regions exist in the inviscid part. A decrease in the suction & blowing amplitude and the Reynolds number tends to suppress the unsteadiness and shock strength, which can be expected in view of the shock weakening and increasing viscosity, respectively. The flow with high Reynolds numbers ($Re \geq 1500$) is also characterized by the appearance of vortex shedding. The dependence of spontaneous vortex shedding on the height and Reynolds number suggests that both parameters are important for a criterion describing the onset of vortex shedding. In addition, we observe that in all cases of vortex shedding, the shedding frequency as represented by the Strouhal number $St_\theta$ in Equation 5-11 is lower than in the case of subsonic flows.
under adverse pressure gradient in Chapter 5. This can be explained by the fact that the blowing in the present case is much larger than in the subsonic case. This large blowing, under which the region of vortex shedding is located, hampers the shedding.

**Properties of separation region**

The influence of the physical parameters on the size of the separation bubble and the reverse flow intensity is shown in Fig. 6-21 and 6-22 in terms of the bubble height and the minimum skin friction coefficient, respectively. The separation bubble in this case is taken from the time averaged flow, and we recall from Chapter 5 that the bubble height is defined as the maximum value of $x_2$ where $\int_0^{x_2} \rho u_1 dx_2 = 0$. The length of the separation bubble varies proportionally to its height, so the height is representative for the size of the separation bubble. In Fig. 6-21 the separation region grows with increasing the height of the upper boundary up to $h = 60$. Above this height the bubble size exhibits a small decrease due to vortex shedding. This vortex shedding results in a decrease of reverse flow intensity (lower negative $c_f$ in Fig. 6-22). Increasing the amplitude of suction and blowing results in a growing separation bubble and increase in reverse flow intensity due to the stronger shock. In evaluating the effect of Reynolds and Mach number, we note that the bubble height should be read as the ratio between the actual observed height and the inflow boundary layer displacement thickness as the absolute value of this thickness depends...
Figure 6-21: Separation bubble height as function of domain height, Mach, Re and suction & blowing amplitude.

Figure 6-22: Minimum $c_f$ as function of domain height, Mach, Re and suction & blowing amplitude.
on $Re$ and $M$. The ratio of the bubble size to the inflow displacement thickness decreases with increasing Reynolds number. This is a consequence of the fact that the boundary layer grows more slowly at higher Reynolds numbers. The steep decrease in reverse flow intensity above $Re = 1000$ is due to a transition from a closed laminar bubble to a bubble which sheds vortices. The ratio of bubble size to the inflow displacement thickness grows with increasing Mach number up to $M = 1.7$, above which the ratio decreases sharply due to the disappearance of the shock, as shown previously in Fig. 6-19. Accordingly, the intensity of the reverse flow also decreases considerably above this Mach number.

![Graph](image)

**Figure 6-23:** (a) Skin friction coefficient of time averaged flow and (b) r.m.s value of wall pressure (solid) and temperature (dashed) fluctuations from the reference case

A typical time-averaged skin friction coefficient is shown in Fig. 6-23a, along with the corresponding root mean square value of the wall pressure and wall temperature fluctuations in Fig. 6-23b. The root mean square of the fluctuations is defined as

$$\text{rms}(q') = \left( \frac{1}{T} \int_{t_o}^{t_o+T} (q')^2 \, dt \right)^{1/2},$$  \hspace{1cm} (6-11)

where $q'$ represents an instantaneous fluctuation of a wall quantity $q$, $t_o$ the starting time of the averaging and $T$ the time interval over which the averaging is performed. The separation point is located upstream, under the suction region. This suction, which causes the flow to accelerate, prevents the separation point to move further upstream. Proceeding downstream from the separation point the shear stress coefficient exhibits a deep valley indicating a strong reverse flow near a reattachment point and forms the kernel of the time-averaged separation bubble. After this separation kernel, other extrema of negative $c_f$ may occur due to compressions in the inviscid flow. The maxima of pressure and temperature fluctuations along the wall are apparently located at the kernel of the separation bubble, as illustrated by their
r.m.s. values. This correlation between the location of the maximum fluctuation of wall quantities and the location of the recirculation centre is typical for the present flow application.

### 6.3.3 Onset criterion for vortex shedding

For boundary layer flows under adverse pressure gradient, Pauley et al. [58] proposed the following criterion for the onset of vortex shedding (in dimensionless form):

$$ P_{\text{max}} = \theta_s^2 Re \left( \frac{du_{in}}{dx_1} \right)_{\text{max}} \approx -0.24, \quad (6-12) $$

where $\theta_s$ is the momentum thickness at the separation point and $(du_{in}/dx_1)_{\text{max}}$ is the maximum (negative) inviscid velocity gradient along the separation region. However, applying this criterion to the present application results in large discrepancies. It was found that the influence of $\theta_s$ on the vortex shedding is not consistent. This is manifest in an unclear correlation between $P_{\text{max}}$ and the occurrence of vortex shedding, as can be seen in the plot of $P_{\text{max}}$ versus Reynolds number based on the domain height, $R_h$ in Fig. 6-24a. The data have the same Mach number. The overlapping area of $P_{\text{max}}$, in which both vortex shedding and non vortex shedding cases occur is large. The configuration of each case is presented in Table 6-2, while the corresponding values of $P_{\text{max}}$ are given in Table 6-3. Furthermore, the maximum inviscid velocity gradient in the present flows containing shocks is so much
Table 6-2: Time averaged result of $R_{\theta_s}$, $\max(-\frac{du_{in}}{dx})$ and $\max(-\frac{du_{out}}{dx})$ from cases with varying $Re, h, M$ and amplitude of suction and blowing.
Table 6-3: Shedding parameters Eq. 6-12, Eq. 6-13 and Eq. 6-16 corresponding to the cases in Table 6-2.
higher than in shockless flows that even in cases where no vortex shedding occurs, the value of $P_{\text{max}}$ is far beyond the threshold value proposed by Pauley. Employing the maximum (negative) velocity gradient $(du_e/du_{1})_{\text{max}}$ along the boundary layer edge instead of the maximum gradient in the inviscid part

$$P_{\text{max}} = \theta_5^2 Re \left( \frac{du_e}{du_{1}} \right)_{\text{max}}$$

(6-13)
does not improve the result considerably, as shown in Fig. 6-24b. The edge velocity, $u_e$, is defined based on the constant pressure assumption across the boundary layer:

$$u_e = \left[ \frac{2}{(\gamma - 1)M^2} \left( 1 - \frac{\gamma - 1}{p_w^{\gamma}} \right) + 1 \right]^{1/2}$$

(6-14)

with $p_w$ is dimensionless wall pressure $p_w/p_{\infty}$. In Fig. 6-24b, we observe that the low values of $P_{\text{max}}$ are mostly occupied by the data of no vortex shedding and the opposite is true for the high values. The overlapping area is, however, still large. On the other hand, it was shown that the domain height and Reynolds number have an important effect on the occurrence of vortex shedding. This correlation between $Re$ and $h$ can be seen in Fig. 6-24. The vortex shedding occurs at high values of $R_h = Re \times h$. This suggests that replacing the length parameter $\theta_5$ by the domain height $h$ forms a logical step toward a better shedding criterion for flows under suction and blowing.

Based on this motivation, we define the following shedding parameter

$$P_{\text{max}} = h^2 Re \left( \frac{du_e}{du_{1}} \right)_{\text{max}},$$

(6-15)
where $h$ is the height of the upper boundary and $\left(\frac{d\nu_c}{dx_1}\right)_{\text{max}}$ is the maximum (negative) gradient of the edge velocity as given above. Plotting this $P_{\text{max}}$ against $Re, h$ and $R_h$ in Fig. 6-25, we see a better correlation between $P_{\text{max}}$ and the occurrence of vortex shedding. In the last figure, the overlapping interval of the shedding parameter is small. However, including the data with various Mach numbers results in a larger overlapping area as shown if Fig. 6-26a. This suggests that the shedding parameter should take the Mach number explicitly into account. From the data we learn that a decreasing Mach number tends to trigger vortex shedding. This implies that the incorporation of Mach number in the shedding parameter is in such away that a decrease in Mach number increases the shedding parameter value. At this level it is difficult, based on the available numerical data, to determined the influence of Mach number on the vortex shedding quantitatively. For that, many additional computations would be needed. We could, however, argue that the role of Mach number in the shedding parameter is equivalent to its role in the momentum thickness at separation point, $\theta_s$, as it appears in the criterion proposed by Pauley et al. [58] in Eq. 6-12. Assuming that $\theta_s$ is proportional to $1/M^2$ we found that the following criterion is appropriate for the onset of vortex shedding in compressible flows under suction and blowing

$$2100 \leq -P_{\text{max}} = -h^2 Re \left(\frac{d\nu_c}{dx_1}\right)_{\text{max}} (0.11 + M^4)^{-1} \leq 2700.$$

Below 2100 vortex shedding does not occur and above 2700 it always occur. The result using this shedding parameter is shown in Fig. 6-26b. The effect of taking
the Mach number into account is that the overlapping area becomes small. The value 0.11 in the shedding parameter is to facilitate the application of the shedding criterion to the incompressible limit \( M \to 0 \) and this value is calibrated at Mach 0.2, using the cases in Chapter 5. In this way this criterion is not only applicable for shock cases, it is also applicable for low Mach number flows. More data especially in the low Mach number regime are, however, desired to specify more precisely the Mach number term in the shedding criterion. These data can be obtained by performing additional calculations or from other researchers, for instance from Pauley et al. [58] or Alam & Sandham [4].

### 6.4 Conclusion

We summarize our new contributions as follows. The numerical method used provides results which agree well with other numerical and experimental results in steady SBLI. In addition, the good performance of the characteristic suction and blowing boundary condition is established for flows containing shocks. Using the characteristic boundary condition, we show for the first time that a local, unsteady SBLI can occur in a supersonic flow under a consecutive suction and blowing. A substantial unsteadiness is not only found in the separated boundary layer, but also in the inviscid shock. The shock unsteadiness occurs in a closed range of domain height and Mach number and weakens with decreasing suction and blowing amplitude and Reynolds number. The average shock strength decreases with decreasing Re, domain height and suction & blowing amplitude. A maximum shock strength is encountered in a bounded range of Mach number. We observed that the fluctuation of the wall quantities has a maximum value at the core of the time averaged separation bubble. The Reynolds number and the domain height are found to have a dominant influence on the occurrence of vortex shedding. Furthermore, it was found that the shedding criterion due to Pauley et al. is not applicable for the present flows containing shocks. A new shedding criterion suitable for compressible flows under suction and blowing is proposed, based on the present results.

Due to the lack of experimental data and analytical results for SBLI under suction and blowing at this moment, it is recommended to reproduce such complex flow using another numerical method for the purpose of comparison. A code based on an implicit time stepping is currently under development in our group and the above SBLI forms a critical test case for the code. Additional cases of simulations with varying Mach numbers should be performed to further tune the Mach number term in the shedding criterion.
Conclusions and recommendations

We conclude the thesis by extracting important findings from the previous chapters and stating the further theoretical or practical implications of these results, followed by some recommendations for future research. Relating to the purpose of the thesis, i.e. firstly the development and validation of the numerical methods and secondly the study of the physical phenomena, we present the conclusions regarding the numerical and the physical aspect separately as to address the research questions stated in Chapter 1.

7.1 Numerical aspects

We selected the finite difference method for the spatial discretization in view of the extension of the method in the future to a general geometry, which is more flexible than other accurate discretization methods such as pseudo spectral and compact finite difference schemes. In Chapter 4 we compared the second order and the fourth order central discretization scheme and found that the fourth order method needs about the half of computational effort of the second order method to achieve a comparable accuracy. The good performance of the third order upwind scheme in flows containing shocks was demonstrated in the study of shock boundary layer interactions in Chapter 6. The accuracy of the spatial discretizations was confirmed by analyzing the convergence rate with increasing grid resolution. At the edges and the corners of the boundaries we made use of dummy points in combination with the central scheme. Although it gives satisfactory results, the value of other techniques such as one sided differencing, which needs no dummy points, ought to be explored. It might improve the growth rate near the inflow boundary.

The Blasius solution to the boundary layer equations provides appropriate laminar profiles at the inflow boundary, while the similarity transformation yields a good initial condition for a non parallel flow under zero pressure gradient. This similarity solution was mostly used as the initial condition in the case of an arbitrary pressure
gradient since only for special cases a better approximation can be obtained such as in the case of the steady shock boundary layer interaction in Chapter 6, where an analytical solution in the inviscid part exists. For 3D simulations, however, the need for a better initial condition than the similarity solution is more profound as it shortens the route to the (statistically) stationary solution, hence saving CPU time. In lack of a better analytical solution for an arbitrary pressure gradient we recommend to first perform the simulation in two dimensions and use the resulting stationary mean solution as the initial condition for the three dimensional simulation. This is beneficial if only the micro scale structure is three dimensional, while the mean flow remains two dimensional.

Numerical experiments of different techniques in modelling artificial and physical boundary conditions suggests the selection of the extrapolation technique for the subsonic inflow boundary, the partly non-reflecting quasi 2D characteristic method for the subsonic outflow boundary and the quasi 2D characteristic method of arbitrary pressure distribution or suction/blowing for the upper boundary. The major implication of this choice is that we can shift the artificial upper boundary close enough to the wall as the boundary condition allows flow unsteadiness at this boundary, in contrast with a vanishing fluctuation boundary. This allows us to truncate the computational domain so as to include only the interesting region. Moreover, the multi dimensionality of the boundary condition allows the use of a non orthogonal boundary, which is suitable for the future extension to a complex geometry. Using the suction/blowing boundary condition, we can also imitate an arbitrary free-slip curved wall, with the benefit of keeping an orthogonal grid. Otherwise, a non-orthogonal grid or a metric transformation of the equations should be applied which increase the computational effort and may influence the accuracy. For a strongly instationary flow, the partly or even perfectly non-reflecting outflow boundary conditions are not sufficient to prevent wave reflections at the outflow boundary and a buffer domain treatment is required. We developed a buffer domain technique which is insensitive to the extent of the computational domain, the physical parameters of the flow and the grid density. The technique, which is based on a direct reduction of fluctuations, occupies only about 10 % of the computational domain or less for a longer computational domain and is much more efficient than other relaminarization methods such as increasing viscosity and parabolization of the equations. Although the buffer domain technique was developed and tested in the linear and nonlinear regime of perturbations (Chapter 3 and 4), it performs satisfactorily up to the turbulent regime (Chapter 5). The existence of viscous dominated flow near the wall makes the use of inflow characteristic boundary conditions inappropriate. This boundary condition, however, may perform better in free shear flows, e.g. the spatial mixing layer.

In every test case (except in the 3D simulation) the adequacy of the grid resolution was investigated by conducting a resolution study as the first step in the simulations. In this way, the well resolved results are guaranteed and the accuracy of the numerical method can be examined without uncertainties arising from this aspect. The accuracy of the method was illustrated by comparing the DNS results
Conclusions and recommendations

with the corresponding references. Especially, a critical comparison was carried out with small perturbation theories (LST and PSE). Our DNS reproduced, for instance, the growth rate of a second Mack mode at a high supersonic Mach number within 2.5 % error, which is comparable to a reference result (Guo et al., 1994) where a sixth order spatial discretization scheme (Chapter 4) was used. Excellent agreements were also achieved in the comparison with semi empirical theories for separated flows, such as the separation criterion by Stratford (Chapter 5), and in the case of steady shock boundary-layer interaction with the experimental result by Hakkinen and the numerical result by Katzer (Chapter 6). Although no reference was available for comparison at this moment, the unsteady shock boundary-layer interaction under blowing and suction has a value of providing an appropriate test case for the development of an implicit time integration method, which is now in progress in our group. The strong, yet controllable unsteadiness of the flow provides a critical test for the maximum accurate time step in the implicit method and this is carried out in a computationally less demanding two dimensional simulation.

The good agreement between the spatial DNS and the PSE (Chapter 4, encourages the use of PSE eigenvectors as the inflow perturbations for the DNS. For a validation purpose, this could give a better approximation for nonparallel flows. In view of the capability of nonlinear PSE to rapidly and accurately modelling a secondary instability up to the spike stage (Bertolotti, 1990), nonlinear PSE can serve as an advance inflow condition for spatial DNS, and hence reduces the streamwise extent of the computational box in the DNS. The possibility of PSE-DNS coupling ought to be explored.

Based on the developed numerical methods, the required computing time for the 3D simulation of transition induced by separation is quite high, although is still lower than a natural transition (Chapter 3). This separation bubble simulation is, however, valuable for the validation of the numerical methods in the turbulent regime, especially the buffer domain treatment, and for offering complex physics in a quite small computational box. Hence, this flow forms a suitable test case for the validation of LES models and the numerical methods provide a reliable algorithm for this purpose. Of course, in order for the DNS to play a role as an accurate reference, a resolution study should be carried out. Therefore, we can take advantage of the symmetry condition. The computational time is also lower if we perform simulations at higher Mach numbers.

As a closing remark regarding the numerical aspect, the numerical methods for spatial DNS has successfully been applied to flow simulations containing complex physics in a simple geometry. The extension of the method to a complex geometry can proceed flexibly, as this possibility has been taken into account in the development of the method.
7.2 Physical aspects

In Chapter 5, we showed that there is a unique correspondence between a suction/blowing and a pressure prescription boundary condition. This implies that we can obtain a desired pressure distribution by applying the corresponding suction and blowing. For that, we first perform a simulation using the pressure prescription boundary condition with the desired pressure distribution. The resulting mean normal velocity along the upper boundary is then the sought suction and blowing profile, while the mean streamline along the upper boundary can be considered as a freeslip wall. This may help experimentalists to obtain an appropriate suction/blowing or to design wall curvature which can produce a certain pressure gradient.

A physical aspect that was studied in Chapter 5 is the influence of upstream perturbations and pressure gradient on a laminar separation bubble flow. One thing that we learned was that upstream disturbances do not change the location of the separation point but can shift the reattachment point in the upstream direction considerably, resulting in a shorter separation bubble. This effect is enhanced with increasing amplification rate of the perturbations, which can result from a higher initial perturbation amplitude or a secondary instability. The practical implication of this result for aerodynamic applications is that one can reduce the extent of the separation region or prevent a separation over the whole surface, such as occurs in a wing stall, by disturbing the laminar flow before separation. This is different from making the flow turbulent artificially through a fixed transition to prevent laminar separation. In the latter case, the viscous drag is higher than in the former case due to a higher turbulent skin friction and consequently the aerodynamic performance is degraded. We should be aware, however, that in the numerical study, the pressure gradient is generated by suction and blowing, whereas in aerodynamic practice the pressure gradient is commonly due to the surface curvature. Nevertheless, the numerical result gave a valuable hint.

In the 3D simulation of transition induced by separation (Chapter 5) the reattachment region is characterized by a substantially high turbulent intensity, and further downstream it recovers gradually to the zero pressure gradient wall turbulence level. The mean streamwise velocity profiles just downstream of the reattachment point resemble those of turbulent flow over a rough surface, which suggests that in the logarithmic layer they can be described by a log law, $u_1^+ = 2.5 \ln(x_2^+) + 5.1 + D$. In the rough surface turbulence, the term $D$ depends on the roughness size. Further investigations are required to find out whether a similar relation exists between $D$ and the properties of the separation bubble flow, and how the pressure development downward from the reattachment point influences the local turbulent profiles.

We found in Chapter 6 that an unsteady shock boundary-layer interaction can occur in a supersonic boundary layer flow subjected to blowing and suction. The resulting strong unsteadiness can be inviscid dominated, characterized by oscillating shocks, or viscous dominated, characterized by vortex shedding. The size of the separation region, the shock strength and the level of shock oscillation are influenced by the flow Reynolds and Mach numbers, the distance between the suction/blowing
and the opposite wall, and the suction/blowing properties. Shock oscillation occurs only in a certain range of Mach number and the distance of suction/blowing to the opposite wall. The oscillation is stronger with increasing Reynolds number. Furthermore, the shock may disappear if the Mach number is very high. This result indicates the possibility of the simultaneous occurrence of shocks and separation when gaseous fuel is injected into supersonic flow in propulsion systems, giving rise to an undesired momentum loss.

We proposed a criterion for the onset of vortex shedding in flows subjected to blowing and suction. The criterion is designed for a large range of Mach numbers, from incompressible flows up to highly supersonic flows containing shocks. A large number of additional testcases are, however, desired to calibrate the effects of Mach number in the criterion. This criterion could help to predict the occurrence of vortex shedding after separation in a nozzle flow as blowing and suction can be associated with a curved wall. In three dimensions, the shedded vortices give rise to turbulence. Hence, the criterion may for instance predict the occurrence of turbulence in the combustor of a propulsion system, which is responsible for fuel-air mixing.
Appendix A

Fourth order spatial discretization

In this appendix we present the finite-volume-like fourth order spatial discretization scheme for the equations and the characteristic based boundary conditions, mentioned in Chapter 2, on non-equidistant, orthogonal grids. This is an elaboration of the fourth order discretization method, which is presented in Chapter 3. For brevity, we only consider the derivatives in the streamwise and the normal direction, and we use the notation $f_{i,j} = f(x(i), y(j), t)$, where $x, y$ are applied instead of $x_1, x_2$, for the streamwise and the normal direction, respectively. The derivatives in the spanwise direction, $z$, are approximated in the same way as in the streamwise, $x$, direction. The indices run from $i = -3$ to $I + 3$ and $j = -3$ to $J + 3$ in the $x$ and $y$ direction, respectively. The three outer points are dummy points, which are defined to accommodate central discretizations at the boundaries. For brevity, only those indices in the direction being considered are written below.

The first derivative with respect to $x$ is approximated by using the following differencing weights:

$$\left. \left( \frac{\partial f}{\partial x} \right) \right|_{0 \leq i \leq I} = \frac{4}{3} \frac{g_{i+1} - g_{i-1}}{x_{i+1} - x_{i-1}} - \frac{1}{3} \frac{g_{i+2} - g_{i-2}}{x_{i+2} - x_{i-2}} + \mathcal{O}(\Delta x^4),$$

where $g$ is averaged in the $y$ direction according to:

$$g_j \left|_{2 \leq j \leq J-1} = -\frac{1}{8} \frac{y_{j+2} - y_j}{y_{j+2} - y_{j-2}} f_{j+2} + \frac{1}{2} \frac{y_{j+1} - y_j}{y_{j+1} - y_{j-1}} f_{j+1} + \frac{5}{8} f_j,$$

$$-\frac{1}{8} \frac{y_j - y_{j-2}}{y_{j+2} - y_{j-2}} f_{j-2} + \frac{1}{2} \frac{y_j - y_{j-1}}{y_{j+1} - y_{j-1}} f_{j-1} + \mathcal{O}(\Delta y^4),$$

$$g_j \left|_{j=1,J} = \frac{1}{4} f_{j+1} + \frac{1}{2} f_j + \frac{1}{4} f_{j-1} + \mathcal{O}(\Delta y^2),$$

$$g_j \left|_{j=0} = \frac{1}{2} f_{j+1} + \frac{1}{2} f_{j-1} + \mathcal{O}(\Delta y^2).$$

$\Delta x$ and $\Delta y$ above and in the next equations are the maximum grid spacing in the $x$ and the $y$ direction within the stencil, respectively.
The first derivative with respect to $y$ is approximated by using the following differencing weights:

$$
\left( \frac{\partial f}{\partial y} \right)_{i,j} \bigg|_{2 \leq j \leq J-1} = \frac{4}{3} \frac{g_{j+1} - g_{j-1}}{x_{j+1} - x_{j-1}} - \frac{1}{3} \frac{g_{j+2} - g_{j-2}}{x_{j+2} - x_{j-2}} + \mathcal{O}(\Delta y^4),
$$

$$
\left( \frac{\partial f}{\partial y} \right)_{i,j} \bigg|_{j=0,1,J} = \frac{g_{j+1} - g_{j-1}}{y_{j+1} - y_{j-1}} + \mathcal{O}(\Delta y^2),
$$

where the averaging for $g$ in the $x$ direction is calculated according to:

$$
g_i \bigg|_{0 \leq i \leq I} = \frac{1}{8} \frac{x_{i+2} - x_i}{x_{i+2} - x_{i-2}} f_{i+2} + \frac{1}{2} \frac{x_{i+1} - x_i}{x_{i+1} - x_{i-1}} f_{i+1} + \frac{5}{8} f_i - \frac{1}{8} \frac{x_i - x_{i-2}}{x_{i+2} - x_{i-2}} f_{i-2} + \frac{1}{2} \frac{x_i - x_{i-1}}{x_{i+1} - x_{i-1}} f_{i-1} + \mathcal{O}(\Delta x^4).
$$

The second derivatives with respect to $x$, $\partial^2 f / \partial x^2$, consists of an inner derivative $f' = \partial f / \partial x$ and an outer derivative $\partial f'/\partial x$. The outer derivative is approximated by using the following differencing weights:

$$
\left( \frac{\partial^2 f}{\partial x^2} \right)_{i,j} \bigg|_{0 \leq i \leq I} = \frac{9}{8} \frac{g'_{i+1/2} - g'_{i-1/2}}{x_{i+1/2} - x_{i-1/2}} - \frac{1}{8} \frac{g'_{i+3/2} - g'_{i-3/2}}{x_{i+3/2} - x_{i-3/2}} + \mathcal{O}(\Delta x^4),
$$

where the averaging for $g'$ in the $y$ direction equals:

$$
g'_{j} \bigg|_{1 \leq j \leq J} = \frac{1}{8} \frac{y_{j+3/2} - y_j}{y_{j+3/2} - y_{j-3/2}} f'_{j+3/2} + \frac{9}{8} \frac{y_{j+1/2} - y_j}{y_{j+1/2} - y_{j-1/2}} f'_{j+1/2} - \frac{1}{8} \frac{y_j - y_{j-3/2}}{y_{j+3/2} - y_{j-3/2}} f'_{j-3/2} + \frac{9}{8} \frac{y_j - y_{j-1/2}}{y_{j+1/2} - y_{j-1/2}} f'_{j-1/2} + \mathcal{O}(\Delta y^3),
$$

$$
g'_{j} \bigg|_{j=0} = \frac{y_{j+1/2} - y_j}{y_{j+1/2} - y_{j-1/2}} f'_{j+1/2} + \frac{y_j - y_{j-1/2}}{y_{j+1/2} - y_{j-1/2}} f'_{j-1/2} + \mathcal{O}(\Delta y^2).
$$

The inner derivative, $f'$, in the expressions above is approximated by using the following differencing weights:

$$
\left( \frac{\partial f}{\partial x} \right)_{i-1/2,j-1/2} \bigg|_{-1 \leq i \leq I+2} = \frac{9}{8} \frac{g_i - g_{i-1}}{x_i - x_{i-1}} - \frac{1}{8} \frac{g_{i+1} - g_{i-2}}{x_{i+1} - x_{i-2}} + \mathcal{O}(\Delta x^3),
$$

where the averaging for $g$ in the $y$ direction equals:

$$
g_{j-1/2} \bigg|_{-1 \leq j \leq J+2} = -\frac{1}{8} \frac{y_{j+1} - y_{j-1/2}}{y_{j+1} - y_{j-2}} f_{j+1} + \frac{9}{8} \frac{y_j - y_{j-1/2}}{y_j - y_{j-1}} f_j - \frac{1}{8} \frac{y_{j-1/2} - y_j}{y_{j+1} - y_{j-2}} f_{j-2} + \frac{9}{8} \frac{y_{j-1/2} - y_j}{y_j - y_{j-1}} f_{j-1} + \mathcal{O}(\Delta y^3).
$$

Analogously to the $x$ direction, the outer derivative with respect to $y$ is approximated by the following differencing weights:

$$
\left( \frac{\partial f'}{\partial y} \right)_{i,j} \bigg|_{2 \leq j \leq I} = \frac{9}{8} \frac{g'_{j+1/2} - g'_{j-1/2}}{y_{j+1/2} - y_{j-1/2}} - \frac{1}{8} \frac{g'_{j+3/2} - g'_{j-3/2}}{y_{j+3/2} - y_{j-3/2}} + \mathcal{O}(\Delta y^4),
$$

$$
\left( \frac{\partial f'}{\partial y} \right)_{i,j} \bigg|_{j=0,1} = \frac{g'_{j+1/2} - g'_{j-1/2}}{y_{j+1/2} - y_{j-1/2}} + \mathcal{O}(\Delta y^4),
$$

where $O$ represents the big O notation.
where the averaging for $g'$ in the $x$ direction is defined by:

$$ g'_i |_{0 \leq i \leq I} = \frac{1}{8} \frac{x_{i+3/2} - x_i}{x_{i+3/2} - x_{i-3/2}} f'_{i+3/2} + \frac{9}{8} \frac{x_{i+1/2} - x_i}{x_{i+1/2} - x_{i-1/2}} f'_{i+1/2} $$

$$ - \frac{1}{8} \frac{x_i - x_{i-3/2}}{x_{i+3/2} - x_{i-3/2}} f'_{i-3/2} + \frac{9}{8} \frac{x_i - x_{i-1/2}}{x_{i+1/2} - x_{i-1/2}} f'_{i-1/2} + O(\Delta x^3). $$

The inner derivative, $f'$, in the expressions above is approximated in the same way as the inner derivative with respect to $x$.

The cross derivative, $\partial^2 f / \partial x \partial y$, is approximated by taking the outer derivative in $y$ and the inner derivative in $x$ or the opposite way. The same procedure holds for the other cross derivatives, $\partial^2 f / \partial x \partial z$, $\partial^2 f / \partial y \partial z$. Further, the coordinates for the outer derivatives are defined as follows:

$$ x_{i+1/2} = \frac{9}{16} (x_{i+1} + x_i) - \frac{1}{16} (x_{i+2} + x_{i-1}), $$

$$ x_{i-1/2} = \frac{9}{16} (x_i + x_{i-1}) - \frac{1}{16} (x_{i+1} + x_{i-2}), $$

$$ x_{i+3/2} = \frac{9}{16} (x_{i+2} + x_{i+1}) - \frac{1}{16} (x_{i+3} + x_i), $$

$$ x_{i-3/2} = \frac{9}{16} (x_{i-1} + x_{i-2}) - \frac{1}{16} (x_i + x_{i-3}). $$

The same holds for the other directions, except for the $y$ component with the indices $j = 0, 1$:

$$ y_{j+1/2} = \frac{9}{16} (y_{j+1} + y_j) - \frac{1}{16} (y_{j+2} + y_{j-1}), $$

$$ y_{j-1/2} = \frac{1}{2} (y_j + y_{j-1}). $$

Finally, the coordinates for the inner derivatives are:

$$ x_{i-1/2} = \frac{1}{2} (x_i + x_{i-1}) $$

for terms with coefficient $\frac{9}{8}$ and

$$ x_{i-1/2} = \frac{1}{2} (x_{i+1} + x_{i-2}) $$

for terms with coefficient $\frac{1}{8}$. 


References


REFERENCES


[64] U. Rist and U. Maucher, Direct numerical simulation of 2-D and 3-D instability waves in a laminar separation bubble, 74th Fluid Dynamics Symposium on "Application of Direct and Large Eddy Simulation on Transition and Turbulence", Chania, Crete-Greece, 34.1-34.7, April (1994).


Summary

This thesis is aimed at the development of a numerical method for compressible Direct Numerical Simulation (DNS) in the spatial setting and at an increased insight in the physical aspects of the flows studied. The developed numerical method is meant to handle flow separation, shock boundary layer interaction and turbulence. As a model problem, we consider flat plate boundary layer flow under various flow conditions. The numerical method consists of a conservative finite volume method for the spatial discretization and a compact storage Runge-Kutta method for time integration. We studied variations in the boundary conditions and a buffer domain technique at the outflow boundary. The latter is needed in order to prevent wave reflections at the outflow boundary.

The method is validated by imposing small perturbations on a base flow and making use of Linear Stability Theory (LST) and Linear Parabolized Stability Equations (PSE) as references. A fourth order spatial discretization method is found to be much more efficient than the corresponding second order method. We checked the stability of parallel and non-parallel flows. In the case of non-parallel flows we observed that the agreement with PSE is better than with LST, as expected. Simulations at Mach number 0.5 and 4.5 were considered in order to study the influence of the buffer domain treatment on subsonic and supersonic flows. Further validations are carried out for two and three dimensional laminar separation bubble flows at Mach number 0.2. The separation bubble is invoked by using a pressure prescription or suction and blowing conditions along the upper boundary combined with the non-reflecting characteristic method. These boundary conditions allow flow unsteadiness along the upper boundary and can handle self excited shocks in supersonic flows. The results show a good agreement with semi empirical theories and the developed numerical method performs well in the turbulent regime. The good performance of a third order shock capturing (upwind) scheme and the boundary conditions is confirmed in the simulation of steady and unsteady shock boundary-layer interactions by a comparison with experimental and other numerical results.

Considering the physical aspects, it was found that upstream disturbances have only a small influence on the separation point but shift the reattachment point considerably in the upstream direction. This effect is enhanced if the perturbations have higher growth rate, such as those with higher initial amplitude or containing oblique modes which trigger secondary instabilities. The turbulence just after reattachment is characterized by a higher level of fluctuations than in the zero pressure gradient wall turbulence and show a log law profile similar to turbulent flow over a rough surface. Depending on many physical parameters, shock buffeting can occur in a supersonic flow under consecutive suction and blowing. A vortex shedding criterion for varying Mach number, from incompressible to supersonic, under suction and blowing is proposed. This criterion is valid for flows with or without shocks.
Samenvatting

Het werk dat in dit proefschrift beschreven wordt is enerzijds gericht op de ontwikkeling van een numerieke methode voor ruimtelijke Directe Numerieke Simulatie (DNS) van samendrukbare vloeistoffen en anderzijds op het vergroten van het inzicht in de fysische aspecten van de onderzochte stromingen. De numerieke methode moet toepasbaar zijn op stromingen met loslating, schok-grenslaag interactie en turbulentie. Als modelprobleem beschouwen we een grenslaag boven een vlakke plaat onder diverse stromingscondities. De numerieke methode is gebaseerd op een conservatieve eindige volume methode als ruimtelijke discretisatie en een compact-storage Runge-Kutta methode als tijdintegratie. We hebben variaties in de randvoorwaarden en een buffertechniek aan de uitstroomrand bestudeerd. Deze buffertechniek is noodzakelijk om reflecties aan deze rand te voorkomen.

De methode is gevalideerd door kleine verstoringen aan een hoofdstroming toe te voegen en de resultaten te vergelijken met Lineaire Stabiliteitstheorie (LST) en lineaire geparaboliseerde stabiliteitsvergelijkingen (PSE). We vonden dat een vierde orde ruimtelijke discretisatiemethode efficiënter is dan een tweede orde methode. We hebben de stabiliteit van zowel parallelle als niet-parallelle stromingen onderzocht. Voor niet-parallelle stromingen is de overeenstemming met PSE beter dan met LST, zoals verwacht mag worden. Om de invloed van de buffer voor subsone en supersone stroming te onderzoeken hebben we simulaties uitgevoerd bij Machgetal 0.5 en 4.5. Verdere validaties zijn uitgevoerd voor twee-dimensionale stromingen met een laminaire loslaatbel bij Machgetal 0.2. De loslaatbel wordt opgewekt door een voorgeschreven druk of een voorgeschreven zuig- en injectiesnelheid op de bovenrand van het domein in combinatie met een niet-reflecterende karakteristiekenmethode. Deze randvoorwaarden maken instationaire stroming langs de bovenrand mogelijk en kunnen schokken opwekken in het geval van supersone stroming. De resultaten komen goed overeen met semi-empirische theorieën en de ontwikkelde numerieke methode werkt goed in het turbulente gebied. De goede werking van een derde orde shock-capturing methode en de randvoorwaarden wordt bevestigd in de simulatie van stationaire en instationaire schok-grenslaag interactie door vergelijking met experimentele en andere numerieke resultaten.

De verstoringen aan de instroomrand hebben maar weinig invloed op het loslaatpunt, maar geven een grote stroomafwaartse verschuiving van het heraanligpunt. Dit effect wordt nog verder versterkt als de verstoringen een grotere groeisnelheid hebben, bijvoorbeeld als ze grotere amplitude hebben of als ze scheve modes bevatten die secundaire instabiliteit veroorzaken. Direct achter het heraanligpunt is het niveau van de turbulente fluctuaties groter dan in het geval van grenslaagturbulentie zonder drukgradiënt. Het snelheidsprofiel lijkt op dat van een turbulente stroming over een plaat met wandruwheid. Afhankelijk van een aantal fysische parameters kunnen oscillerende schokken in een supersone stroming voorkomen bij opeenvolgende zuiging en injectie aan de bovenrand. Er wordt een criterium voor vervalschudding voorgesteld dat afhankelijk is van het Machgetal en toepasbaar is voor stromingen variërend van incompressibel tot supersoon. Dit criterium is geldig voor stromingen met of zonder schokken.
Ringkasan


Acknowledgement

This thesis is the result of a four-years research project which is a part of CFD research activities within the group of Prof. Zandbergen. Many have contributed in accomplishing this thesis.

First, I am greatly indebted to my promotor, Prof. Zandbergen for the opportunity, to work on the research project, for the valuable advices during the progress meetings, which were very motivating, and for the financial support through his funds during the first year.

This work was further financially supported by the Royal Netherlands Academy of Sciences (Koninklijke Nederlandse Academie van Wetenschappen, KNAW). The three dimensional calculations had not been possible without the computer support of the National Computing Facilities (Stichting Nationale Computer Faciliteiten, NCF).

I would like to express my gratitude to my direct supervisors, Bernard Geurts and Hans Kuerten, who have been patiently listening to my opinions and suggestions, yet subjecting them to lots of critical questions, not only to make me come up with reasonable arguments but, importantly, to avoid some misunderstanding during our conversations in Dutch. I also thank them for giving me the opportunity to attend international conferences.

The eort of Ir. Anton de Bruin from the National Aerospace Laboratory (Nationale Lucht- en Ruimtevaartlaboratorium, NLR), as a member of the supervisory committee is very much appreciated. His expertise in the aeronautical applications has been valuable.

I wish to thank my room mate, Ruerd Heeg, for helping me with the PSE code and for being a companionable colleague.

Further, I am indebted to Dr. Bernhard Müller of Luleå University of Technology, Sweden, who has reviewed the entire manuscript and for the constructive discussions during his stay in Twente.

In this regard, I also appreciate the interest and valuable comments of Prof. Germano of Politecnico di Torino, Italy, to my work, which were encouraging and set me to think further on some points.

Finally, many thanks are due to my mother, brother and sister in Indonesia for their moral support through emails, letters and phone calls, and the 'morning coffee' group for the social interaction.

In oktober 1993 begon hij een promotieonderzoek als Asistent in Opleiding bij de vakgroep Toegepaste Analyse, faculteit der Toegepaste Wiskunde van de Universiteit Twente. Onder promotor Prof. Zandbergen en begeleiding van Dr. B.J. Geurts en Dr. J.G.M. Kuerten werkte hij aan de ontwikkeling van een numerieke methode voor ruimtelijke Directe Numerieke Simulatie. Verder zijn de bestudeerde compressibele wandstromingen op zich fysisch interessant. Tijdens dit promotieonderzoek begeleidde hij ook studenten in het practicum Numerieke Wiskunde en in het werkcoullege Analyse C. Het opdoen van kennis in twee verschillende universiteiten in Nederland gaf de auteur een waardevolle ervaring. De grotere vrijheid als student in Delft vergt een grote eigen verantwoordelijkheid en volwassenheid, terwijl de strakke begeleiding in Twente zeer belangrijk is om het werk binnen de geplande tijd te voltooien.