Roe Scheme in Generalized Coordinates; 
Part II- Application to Inviscid and Viscous Flows

M. J. Kermani* E. G. Plett†

Department Of Mechanical & Aerospace Engineering
Carleton University, Ottawa, Canada

Abstract

The recently developed formulation for the numerical flux resulting from use of the Roe scheme in generalized coordinates as outlined in Part I of this publication, is now applied to several inviscid and viscous test cases. The test cases studied are as follows: inviscid transonic flow over a bump and in a Laval nozzle (the results of which are not detailed in this paper), Blasius flow, plane Poiseuille flow, and laminar boundary layer interaction with an oblique shock over a flat plate. In this paper, the primitive variables are extrapolated to the cell faces by the MUSCL idea using the third order upwind biased scheme. The van Albada flux limiter is used to prevent spurious numerical oscillations. The viscous terms are centrally differenced.

Introduction

The direct application of the Roe scheme, [16], to fluid equations in the physical domain has been reported by others, see for example Refs. [1, 5, 15]. Only a few reports have described the application of the Roe scheme in generalized coordinates and they are very brief, e.g. [17]. Fluid equations in the physical domain and in the computational domain are put side by side term by term in part I of this publication and a formula is obtained to give the numerical flux predicted by the Roe scheme in generalized coordinates. This numerical flux is written in terms of grid-geometry (metrics of transformation from the physical domain to the computational domain) and flow parameter (see Ref. [10] for more detail). That method is now applied to some test cases in this paper.

Governing Equations

The fluid equations for the viscous, unsteady and compressible flow in full conservative form in generalized coordinates with no body force could be approximated by thin layer Navier-Stokes equations as follows, see Ref. [6] for example:

\[
\frac{\partial Q_1}{\partial t} + \frac{\partial F_1}{\partial \xi} + \frac{\partial G_1}{\partial \eta} = \frac{\partial G_{1_{VT}}}{\partial \eta} \tag{1}
\]

where \(Q_1\) is the conservative vector, \(F_1\) and \(G_1\) are the inviscid flux vectors, and \(G_{1_{VT}}\) is the viscous flux vector obtained with the thin-layer approximation, all determined in generalized coordinates. The thin-layer approximation is suitable for high speed flows, in which all the viscous derivatives along the main stream of the flow, such as along \(\xi\), are neglected. Eqn. 1 is discretized over the control volume \((j,k)\) as shown in Fig. 1 as follows:

\[
\frac{\partial Q_1}{\partial t} + \frac{F_{1_E} - F_{1_W}}{\Delta \xi} + \frac{G_{1_N} - G_{1_S}}{\Delta \eta} = \frac{G_{1_{VT_N}} - G_{1_{VT_S}}}{\Delta \eta} \tag{2}
\]

Figure 1: The schematic of inviscid fluxes entering and leaving the control volume in generalized coordinates.

---

*Ph.D. Candidate, AIAA student member.
†Professor, Associate Fellow AIAA.
Copyright ©2001 by M. J. Kermani & E. G. Plett. Published by the American Institute of Aeronautics and Astronautics, Inc. with permission.
where $F_{1,E}$ is the inviscid numerical flux obtained in generalized coordinates at the east cell face, $E$, as described in Ref. [10]. $F_{1,w}$, $G_{1,n}$ and $G_{1,s}$ are the inviscid fluxes at the west, north and south cell faces respectively and $G_{1,v_{1}n}$ and $G_{1,v_{1}s}$ are the viscous fluxes obtained at the north and south cell faces of the control volume. According to the thin-layer approximation, the east and west terms of the viscous fluxes are canceled by each other.

Presentation of Numerical Flux by the Roe Scheme in Generalized Coordinates

The inviscid numerical flux $F_{1,E}$ based on the Roe scheme is written in generalized coordinates, according to Ref. [10].

$$F_{1,E} = \frac{1}{2} \left[ F_{1,E}^{I} + F_{1,E}^{R} \right] - \frac{1}{2} \sum_{n=1}^{4} \left[ \lambda_{E}^{n} \delta w_{E}^{n} T_{E}^{n} \left( \sqrt{\xi_{E}^{2} + \xi_{E}^{2}} J \right) E \right],$$

where $F_{1,E}^{I}$ and $F_{1,E}^{R}$ are the inner and outer values of $F_{1}$ determined at face $E$, $\lambda_{E}$'s are the eigenvalues of the Jacobian matrix determined at Roe's averaged condition, $\delta w_{E}$'s are the wave amplitude and $T_{E}$'s are the eigenvectors corresponding to the eigenvalues ($\lambda_{E}$'s) determined at Roe's averaged conditions. For a complete description of these parameters the reader is referred to Ref. [10]. $F_{1,w}$ is equal to $F_{1,E}$ at the previous node in the $\xi$ direction. $G_{1,n}$ is determined in the same manner as for Eqn. 3. More details are given in Ref. [10]. The viscous fluxes are obtained by central differencing.

Time Discretization

For the time discretization, the following two-step explicit scheme, from the Lax-Wendroff family of predictor-correctors, is used. The predictor step provides the flow condition in an intermediate step $n + 1/2$.

$$\frac{Q_{1}^{n+1/2} - Q_{1}^{n}}{\Delta t/2} + \left( \frac{\partial F_{1}}{\partial \xi} \right)^{n} + \left( \frac{\partial G_{1}}{\partial \eta} \right)^{n} = \left( \frac{\partial G_{1,v_{1}n}}{\partial \eta} \right)^{n+1/2},$$

where $Q_{1}^{n}$ are the eigenvectors corresponding to the eigenvalues ($\lambda_{E}$'s) determined at Roe’s averaged conditions. For a complete description of these parameters the reader is referred to Ref. [10]. $F_{1,w}$ is equal to $F_{1,E}$ at the previous node in the $\xi$ direction. $G_{1,n}$ is determined in the same manner as for Eqn. 3. More details are given in Ref. [10]. The viscous fluxes are obtained by central differencing.

Space Discretization

In this computation a third order upwind-biased algorithm with the MUSCL extrapolation strategy, [21], is applied to the primitive variables pressure ($p$), velocity components ($u, v$) and temperature ($T$), in order to obtain the inner ($L$) and outer ($R$) flow conditions. For example at the east cell face of the control volume, $E$, the $L$ and $R$ flow conditions are determined as follows:

$$q_{E}^{L} = q_{j,k} + \frac{1}{4} \left[ (1 - \kappa) \Delta w_{q} + (1 + \kappa) \Delta E_{q} \right]$$

$$q_{E}^{R} = q_{j+1,k} - \frac{1}{4} \left[ (1 - \kappa) \Delta E_{q} + (1 + \kappa) \Delta E_{q} \right],$$

where $q$ represents a primitive variable, i.e. either ($p, u, v, T$) and $\kappa = 1/3$ for the current study as the third order upwind biased scheme and: $\Delta w_{q} = q_{j,k} - q_{j-1,k}$, $\Delta E_{q} = q_{j+1,k} - q_{j,k}$ and $\Delta E_{q} = q_{j+2,k} - q_{j+1,k}$. In the current computations, the van Albada et. al. flux limiter, Ref. [20], is applied. This limiter has been reported and implemented in two different forms in the literature. The first approach as implemented for example in Ref. [18], recommends:

$$q_{E}^{L} = q_{j,k} + \frac{\phi}{4} \left[ (1 - \kappa \phi) \Delta w_{q} + (1 + \kappa \phi) \Delta E_{q} \right]$$

$$q_{E}^{R} = q_{j+1,k} - \frac{\phi}{4} \left[ (1 - \kappa \phi) \Delta E_{q} + (1 + \kappa \phi) \Delta E_{q} \right],$$

where $\phi$ is the limiter function, which is a function of forward- and backward-differences, as defined by:

$$\phi_{j,k} \equiv \frac{2(\Delta w_{q})(\Delta E_{q}) + \epsilon}{(\Delta w_{q})^{2} + (\Delta E_{q})^{2} + \epsilon}$$

and $\epsilon$ is a small number which prevents indeterminacy in regions of uniform flow, i.e. in regions $(\Delta w_{q}) = (\Delta E_{q}) = 0$. The flux limiter extrapolation formulation given by Eqn. 7 cannot totally prevent the spurious numerical oscillations. Alternatively, it is found that the following form of this flux limiter is better able to prevent the spurious numerical oscillations and give better convergence, [2, 19].

$$q_{E}^{L} = q_{j,k} + \frac{\phi}{4} \left[ (1 - \kappa) \Delta w_{q} + (1 + \kappa) \Delta E_{q} \right]$$

$$q_{E}^{R} = q_{j+1,k} - \frac{\phi}{4} \left[ (1 - \kappa) \Delta E_{q} + (1 + \kappa) \Delta E_{q} \right].$$

Eqs. 9 has been used throughout this paper and Ref. [8] for the extrapolation of primitive variables. To avoid expansion shocks from appearing in those regions of sonic expansions the following entropy correction formula is used throughout this paper and

\[ E \text{American Institute of Aeronautics and Astronautics Paper 2001-0087.} \]
also in Ref. [8]. For more detail relating to the entropy correction formula and its assessment, the reader is referred to Ref. [9].

\[
\lambda_{new} \leftarrow \frac{\lambda^2 + \epsilon^2}{2}, \quad |\lambda| < \epsilon
\]

(10)

where \( \lambda \) is the eigenvalue of the Jacobian flux matrix determined at Roe’s averaged condition, and \( \lambda^L \) and \( \lambda^R \) are the eigenvalues determined at inner or outer flow conditions, respectively.

**Inviscid Flow Test Cases**

The numerical flux Roe, developed in generalized coordinates as outlined in Ref. [10], has been applied to the following inviscid test cases: (1) Transonic channel flow over a bump, and (2) inviscid transonic flow in a converging-diverging nozzle, also called a Laval nozzle. A complete explanation of these test cases and the application of Roe’s numerical flux in generalized coordinates to these test cases are given in Refs. [9, 10], and are not repeated here due to space limitations. In both of these test cases, subsonic flow accelerates to supersonic flows and a terminating normal shock returns the flow to subsonic flow. Both of these test cases contain regions of sonic expansion, i.e. local Mach number 1. The expansion shocks are avoided by the entropy correction formula as given by Eqn. 10. The accuracy assessment for these cases show excellent agreement with analytical results and computations by others.

**Blasius Flow**

The first viscous flow validation case, is the viscous incompressible laminar flow over a flat plate at zero angle of incidence. The results of this test are compared with the analytic results of Blasius. The Mach number of the free stream is 0.5 and the Reynolds number based on the length measured from the leading edge of the plate is 10,000. The geometry of the plate is shown in Fig. 2. The flat plate has a length \( L \) and the computational domain is extended a distance \( L/2 \) upstream of the leading edge. This distance allows the flow to develop from the uniform inflow condition as it approaches the plate.

The number of grids for this test case are 31 along the plate (including grids ahead of the plate) and 21 in the transverse direction (with clustering factor 1.7), as shown in Fig. 3. The grid was generated by an algebraic grid generator code. An initial condition, which is the same as the free stream condition is applied for this problem. The boundary conditions are as follows. Inflow conditions of uniform total pressure and total temperature with zero transverse velocity component were applied at the inlet to the computational domain. Outflow conditions corresponding to the free stream pressure at the top and downstream boundary of the computational domain were specified. No slip adiabatic wall conditions were specified on the plate with symmetry conditions applied across the extended plane ahead of the plate.

The classical Blasius similarity solution provides data for comparison. The text by White (1991) discusses this solution. The computed velocity component along the plate, \( u \), is compared with that of Blasius solution as shown in Fig. 4. In the current computation the boundary layer has been accurately captured within 8 grid points by the third order upwind biased scheme as applied to Roe’s algorithm. With the first order scheme, the boundary layer profile has also been captured fairly well everywhere except at the edge of the layer. Using the van Leer flux vector splitting algorithm with the third order upwind-biased scheme in the same flow condition,
i.e. Mach=0.5 and Reynolds number 10,000, at least 20 nodes were required within the boundary layer in order to reasonably capture the boundary layer profile, see Ref. [18]. The first order upwind scheme of van Leer was totally incapable of capturing the boundary layer profile with this number of nodes (with the results that the boundary layer thickness was predicted to be four to five times larger than that of Blasius solutions, see Ref. [18]). This present success is due to the non-diffusive property of the Roe scheme, which makes it suitable for the viscous flow computations. A comparison for the level of diffusivity of various upwind and central numerical schemes such as those of van Leer’s flux vector splitting, [22], Roe’s flux difference splitting, [16], MacCormack’s predictor-corrector scheme, [14], and Jameson’s central scheme, [7], is given in Ref. [23].

The wall skin friction factor for the laminar flow over the flat plate is also compared to that of Blasius solution in Fig. 5. As shown in these figures the agreement is very good.

**Plane Poiseuille Flow**

The next validation test is the laminar incompressible flow between two parallel plates as shown in Fig. 6. The fully developed flow between the plates is compared with the analytic results of plane Poiseuille flow.

In this example, the plates have a length $0.024$ meter ($2.40 \text{ cm}$) and are separated by $1.6 \times 10^{-4}$ meter. The leading edges of the plates are located at $x = 0.0$. The computational domain is extended to some distance upstream of the leading edges of the plates where uniform inflow is specified.

In this example, flow with average velocity $\approx 26 \text{ m/s}$ enters the domain of computation. This flow gradually develops as it approaches the plates. Over each plate a boundary layer develops and thickens along the plate until the boundary layers from two plates meet each other at some distance from the leading edge of the plate, $L_e$. $L_e$ is the flow development region (also called entrance length). For $x \geq L_e$, the velocity profile remains unchanged with $x$ and flow is called *fully developed*. In the fully developed region a linear pressure drop in the flow direction is expected.

The flow conditions at the inlet plane are set to: $p_{0,\text{in}} = 107,827$ $\text{pa}$, $T_{0,\text{in}} = 292.2$ $\text{°K}$, and $\Phi = 0$, where $p_{0,\text{in}}$ and $T_{0,\text{in}}$ are the total pressure and temperature and $\Phi$ is the flow angle. The Reynolds number in this test case based on the inflow condition and the hydraulic diameter, $D_h$, is: $Re_{D_h} = \frac{\mu_{\text{in}}}{\mu_{\text{in}}} D_h / \mu_{\text{in}} $
575, where \( D_h = 2h \) for a channel consisting of two parallel plates extending to infinity in the \( z \) direction, see Fig. 6. The Reynolds number 575 in this problem is well below the critical Reynolds number, 2300, so the flow could be safely assumed to be laminar.

Due to the flow symmetry above and below the centerline only half of the geometry is computed here. In this study 201 \( \times \) 45 uniform grid are taken along the plate and in transverse direction, respectively. A uniform initial condition of \( p = 101300 \) \( \text{Pa} \), \( T = 287 \) \( ^\circ \text{K} \) with \( \text{Mach} = 0.3 \) is applied for this problem.

Outflow conditions corresponding to the pressure \( p = 101300 \) \( \text{Pa} \) at the outlet of the computational domain were specified. No slip adiabatic wall conditions were specified on the plate with the symmetry conditions ahead of the plate. The symmetry condition also has been applied at the center plane between the plates.

Results and Comparisons

The analytical solution of the plane Poiseuille flow are taken for comparison with the results obtained from the current study in the fully developed region. The text by White (1991) discusses the analytical solution of plane Poiseuille flow in which the velocity profile in the fully developed region is determined as follows. The x-momentum Navier-Stokes equations in the fully developed region is simplified to: 

\[
0 = -\frac{\partial p}{\partial x} + \mu \frac{\partial^2 u}{\partial y^2}.
\]

For \( p \) being a function of only \( x \) and \( u \) a function of only \( y \) we conclude that: 

\[
\frac{\partial p}{\partial x} = \mu \frac{\partial^2 u}{\partial y^2} = \text{constant}.
\]

Therefore, \( \frac{\partial}{\partial x} = \frac{d}{dx} \) and \( \frac{\partial}{\partial y} = \frac{d}{dy} \). Hence, 

\[
0 = -\frac{dp}{dx} + \mu \frac{du}{dy}.
\]

The boundary conditions are: \( u = 0 \) at the bottom wall, i.e. \( y = 0 \), and \( \frac{du}{dy} = 0 \) at the centerline, i.e. \( y = \frac{h}{2} \), see Fig. 6. Therefore, velocity profile in the fully developed region becomes:

\[
u = \frac{1}{2\mu} \left[ -\frac{dp}{dx} \right] y (h - y), \tag{11}
\]

where \( -\frac{dp}{dx} \) is the pressure gradient along the plate which is constant in the fully developed region. Eqn. 11 gives the analytical value for the velocity profile in the fully developed region. To compare the velocity profile given by Eqn. 11, i.e. the analytical by obtained velocity profile, with that of the current computation, the values for \( \frac{dp}{dx}, \mu \) and \( h \) must be given to Eqn. 11. In the current computation the following values are taken:

\[
h = 1.6 \times 10^{-4} \quad \text{m}
\]

\[
\mu = 1.813 \times 10^{-5} \quad \text{N} \cdot \text{s/m}^2. \tag{12}
\]

The input value of \( \frac{dp}{dx} \) to Eqn. 11 is taken from the pressure distribution at the center line of the two parallel plates.

![Figure 7: Comparisons of pressure distribution between the two parallel plates.](image)

![Figure 8: Comparisons of velocity profile \( u \) between the two parallel plates.](image)
plates (obtained from the current computation), as shown in Fig. 7. As shown in Fig. 7, $p$ varies linearly in the $x$-direction in the fully developed region providing a constant value for the $\frac{dp}{dx}$ in this region. This constant is:

$$\frac{dp}{dx} \approx -2.417 \times 10^5 \frac{pa}{m}.$$  \hspace{1cm} (13)

Substituting the values from Eqns. 12 and 13 in to Eqn. 11, the analytical velocity profile becomes:

$$u = 6.666 \times 10^9 y (1.6 \times 10^{-4} - y) \frac{m}{s}.$$  \hspace{1cm} (14)

Comparison of the analytical by the obtained velocity profile and that of the current computation at the exit plane of the parallel plates is shown in Fig. 8, where the agreement may be seen to be very good.

**Shock-Boundary Layer Interaction**

The next test case presents a more severe test in which a shock wave interacts with a laminar boundary layer over the flat plate. This is a phenomenon of great complication because the behavior of the boundary layer mostly depends on the Reynolds number, whereas the Mach number predominantly determines the conditions in the shock wave. In the current test case a shock wave interaction with a laminar boundary layer that develops on a flat plate is studied.

As shown in Fig. 9, an oblique shock enters the domain of interest and impinges on a laminar boundary layer on flat plate. This shock is introduced as an incident shock in Fig. 9. If the incident shock has sufficient strength it will cause the boundary layer to separate at point $S$-upstream of the impingement point. The boundary layer downstream of the impingement point reattaches to the plate at $R$. This produces a separation bubble. This bubble looks like an imaginary bump to the main flow. Therefore, a sequence of processes are induced by this imaginary bump as follows; (1) compression waves caused by separation portion, (2) expansion fans caused by the turning of flow over the bump in and around the bump maximum point and (3) re-compression waves to make the flow streams parallel to the wall after the stream passes the bump’s maximum point. The study of flow around this separation bubble is the focus of this section.

As shown in Fig. 9 the compression and re-compression waves approach each other, converging to each other and make a stronger wave. On the other hand the expansion fans diverge as depicted in Fig. 9. This portrait agrees with the physics of traveling waves, as described here. Consider a simplified case of uni-dimensional flow containing a series of compression- or expansion-waves. For a sequence of compression waves, the front waves are always followed by the waves at higher temperature, which contain higher wave speeds. So compression waves overtake each other (converge on each other) making a stronger wave which could grow to a shock wave. On the other hand, in a sequence of expansion waves the front waves are followed by waves at lower temperature containing lower wave speed. So expansion waves cannot catch up to those preceding them so they diverge forming expansion fans.

Away from the separation bubble and around the leading edge of the plate, a weak shock is generated from the boundary layer development. This shock is generated by the initial growth of the boundary layer.

The model problem corresponds to the experiments of Hakkinen et. al. (1959), Ref. [4]. This experiment was performed in an $8 \times 8$-inch supersonic wind tunnel in the Gas Turbine Laboratory of the Massachusetts Institute of Technology. The tunnel was able to deliver a flow with a fixed $Mach = 2$. The Reynolds number range, based on the distance from the leading edge of the plate to the shock impingement point, were $1 \times 6.0 \times 10^5$.

Numerically simulating this determining test has been a challenge for the accuracy assessment of numerical schemes. Several people have assessed their numerical schemes and flow solvers against this case. To name a few: Beam and Warming (1978), Ref. [3], MacCormack (1975), Ref. [13], Thomas and Walters (1987), Ref. [18], Liou and Steffen (1993), Ref. [11] and Amaladas (1995), Ref. [1]. All of
these researchers have studied a case corresponding to Reynolds number equal to $2.96 \times 10^5$ with the shock angle $\beta = 32.6^\circ$ and inflow Mach = 2.00.

**Computational Domain and Flow Description**

The flat plate has a length of 9 cm. The leading edge of the plate is located at $x = 0.0$. The computational domain is extended to some distance upstream of the leading edge of the plate, 6 mm. This distance allows the leading edge shock wave to form properly. The geometric configuration of the flow past the plate is shown in Fig. 10. As shown in this figure, $L$ is the location that the incident shock impinges on the plate if the flow were assumed inviscid.

Two grid resolutions are used here. (1) A very fine grid, $201 \times 201$, in order to completely capture the physical detail of the flow, and (2) some coarser grids for the grid independency tests, e.g. $85 \times 99$ and $41 \times 49$.

Free stream conditions for this test case are taken as follows: $M_{\infty}=2.00$, $p_{\infty}=13$ kpa and $T=288^\circ K$. The Reynolds number per unit length is: $5.98 \times 10^6$ m$^{-1}$, or $Re_L = 5.98 \times 10^6 \times L$. To compare the result of this study with the experiments of Hakkinen (1959) in which $Re_L = 2.96 \times 10^5$, we require $L=4.95$ cm, where $L$ is the shock impingement point if the flow were inviscid.

**Initial and Boundary Conditions**

A uniform initial condition of $p=13$ kpa, $T = 288^\circ K$, with $M=2.00$ and flow angle $\phi = 0.0$ w.r.t. the x-axis are specified as the initial conditions for this problem. The boundary conditions for this problem are as follows. At the supersonic inflow, all primitive variables were specified. The inflow plane is split into two parts. (1) Below the incident shock location, where free stream conditions: $p=13$ kpa, $T = 288^\circ K$, with $M=2.00$ and flow angle $\phi = 0.0$ are specified and (2) above the incident shock entrance location, where primitive variables corresponding to the post shock conditions are specified. These post shock conditions are determined from Rankine-Hugoniot relations by selecting the shock angle $\beta = 32.6^\circ$ and the upstream Mach number equal to 2.00. That gives: $p_2/p_1=1.188$, $T_2/T_1=1.051$, $M_2=1.889$ and $\phi =3.108$. Therefore, the post shock conditions (conditions above the incident shock location at the inlet plane) are determined as follows: $p=15.444$ kpa, $T=302.688^\circ K$, $u=657.802 \frac{m}{s}$ and $v=-35.717 \frac{m}{s}$.

The exit plane is positioned far enough from the separation region so that all the gradients in the flow direction are set to zero. Although this condition is not exact, it can be justified as follows. The boundary layer equations close to the bottom wall are parabolic in type, which allows the exit plane condition to be safely determined from its history at the upstream neighborhood of the exit plane. The remainder of the flow in the exit plane is supersonic and its governing equations are hyperbolic. Therefore, setting the gradients of primitive variables along the plate at the exit plane equal to zero will not introduce significant error in the region of shock boundary layer interaction.

A no slip adiabatic wall condition was imposed on the solid wall at the bottom of the computational domain. Also, the symmetry condition is enforced ahead of the plate at the bottom of the computational domain. The top of the computational domain is assumed to be far enough from the computational domain so the reflected shocks and the leading edge shock totally remain inside the computational domain. This allows the same condition at the inlet plane as referred to post shock conditions, i.e. above the incident shock, to be enforced at the top boundary of the computational grid.

**Results and Comparisons**

A qualitative study of the shock boundary layer interaction is performed first. To completely capture the physical detail of the flow, a very dense grid, i.e. $201 \times 201$, is taken. This provides enough nodes inside the separation bubble, $\approx 30$, in order to capture the profile of recirculating flow.

Fig. 11 shows the density contours obtained from the current study. As shown in this figure, the incident shock is strong enough to cause the boundary layer to separate. This boundary layer reattaches somewhere downstream of the impingement point.
To study the flow condition in and around the separation bubble, the region is split into two regions. (1) Outside the separation bubble, i.e. away from the circulating region and in the inviscid core, and (2) inside the separation bubble, as follows:

(1) Outside the Recirculating Region

Fig. 12 shows the velocity vector and streamline patterns in and around the separation bubble. The separation bubble, earlier called an imaginary bump, causes a sequence of compression waves, expansion fans and re-compression waves to be generated, as depicted in Fig. 9. The influence and presence of the imaginary bump in the region away from the separation bubble (above the separation bubble) is studied here. The compression and expansion effects of these waves can be better seen if the pressure distribution is illustrated along a constant \( y \) line above the separation bubble. As shown in Fig. 12, the location of the center of the separation bubble is well below \( y \approx 1 \) \( mm \). Hence, \( y=1 \) \( cm \) is well above and outside the separation bubble. Fig. 13 shows the pressure distribution along \( y = 1 \) \( cm \). In this figure, the pressure jump due to the leading edge shock, incident shock, compression wave, expansion fans and re-compression waves are shown. The pressure rise in the compression wave, pressure drop in the expansion fan followed by the pressure rise in the re-compression waves are clearly shown in Fig. 13.

Referring to Fig. 11, the approach (or convergence) of compression waves and divergence of expansion fans as schematically sketched in Fig. 9 has also been correctly predicted in the current computation. This experience agrees well with the physical interpretation of the uni-directional flow for the compression waves (convergence of compression waves) and expansion fans (divergence of expansion waves) as explained earlier.

The weak shock generated by the initial growth of the boundary layer at the leading edge of the plate has also been captured in the current computation, as shown in Figs. 11 and 13.

(2) Inside the Recirculating Region

Near the point where the shock wave approaches the wall, i.e. in and around the recirculating zone, the rate of change of \( \partial u/\partial x \) and \( \partial v/\partial x \) become the same order of magnitude and the pressure gradient in the transverse direction, \( \partial p/\partial y \), can no longer be ignored. That is, the boundary layer approximation is not an appropriate assumption in this region anymore. The physics of the flow inside the separation bubble is studied here. Fig. 14 shows the iso-bar lines near the place that the incident shock reaches the wall. The iso-bar lines are bent backward in this region. This phenomena is explained here by a close look at the pressure distribution in the vicinity of the wall.

Due to the no slip conditions at the solid boundary, the particles near the wall can only move with subsonic velocities, as opposed to the flow away from the wall, which can move supersonically. Therefore, the incident shock wave, which originated in the external stream, cannot reach right down to the wall. Hence, the iso-bar lines which are originally as shock wave in the supersonic region, i.e. away from the wall, diffuse to compression waves near the wall in the subsonic region. This phenomena is shown in Fig. 14 and is explained as follows.

Upstream of the recirculating zone, the streamlines inside the boundary layer are parallel to that of the external flow just outside the boundary layer. Therefore, pressure inside the boundary layer is the same as that at the edge of the boundary layer. That is \( \partial p/\partial y \) vanishes in this region. In other words, the boundary layer theory applies there. The same idea is true downstream of the recirculating region, where streamlines become parallel again to the contour of the body, after the flow re-attachments, and boundary layer concept is applicable again. That is pressure inside the boundary layer match the value just outside the boundary layer, i.e. \( \partial p/\partial y \approx 0 \) likewise upstream of the recirculating zone. The fact that boundary layer pressure is the same as that just outside the boundary layer both upstream and downstream of the separation bubble, requires that the pressure rise inside the boundary layer match that of the external flow.

Outside the boundary layer, in which supersonic flow exists, the presence of shock waves are allowed, and large gradients of pressure (or any other primitive variable) are permitted. On the other hand
inside the subsonic region of the boundary layer, near the wall, shocks are not allowed. This makes the pressure rise near the wall to be more gradual than that of the external stream. This flattening of the pressure gradient in the subsonic region of the boundary layer is illustrated in Fig. 14, where the isobar lines as shock waves away from the wall are diffused and made compression waves near the wall and, therefore, they are bent backward near the place that the shock approaches the wall. This phenomena is quite different from the inviscid flow in which shock waves can totally reach the solid wall, as shown in Fig. 15. Therefore, no diffusion of pressure gradient is observed near the wall for inviscid flow. Comparison of wall pressure distribution on the wall for viscous and inviscid flows are shown in Fig. 16. Interestingly, the ultimate pressure rise in the viscous flow matches that of inviscid flow, as shown in Fig. 16, although pressure is significantly diffused for the viscous flow as opposed to the inviscid case. Also shown in Fig. 16, the pressure profile becomes almost level, flat, around the location that the shock wave approaches the wall. This is the plateau profile of pressure inside the recirculating region. More detail on the pressure distribution inside the separation bubble in the shock boundary layer interaction, is given by Hakkinen (1959), Ref. [4].

Comparisons with Experimental Values

The quantitative study of the shock boundary layer interaction is given here. Fig. 17 shows the comparison of pressure distribution over the plate obtained by the current computation with that of experimental values. The agreements are excellent. Fig. 18 shows the comparison of skin friction over the plate obtained by the current computation with that of experimental values. The agreements are fairly good.

The test cases are performed in three different grid densities, 201 × 201, 85 × 99 and 41 × 49, in order to obtain a grid independent solution. The results shown in Figs. 17 and 18, show that results are fairly grid independent even in the coarsest grid, 41 × 49. This is an interesting feature of the Roe scheme as a non-diffusive scheme, which allows that the solution become grid independent on a fairly coarse grid. This feature of the Roe scheme makes it suitable for the viscous flow computations. The same problem has been computed by van Leer scheme as reported in Ref. [18], in which 113 nodes were required in the transverse direction to achieve a grid independent solution.

Conclusion

The formulation for the numerical flux Roe in generalized coordinates as outlined in Part I of this publication are compared with those of analytical values or experimental results. Good agreement for all the test cases was obtained.

Acknowledgment

The financial support for the course of this study is provided by the Ministry of Culture and Higher Education of I.R. of Iran and Carleton University. They are gratefully acknowledged.

References


Figure 13: Pressure distribution along the line $y \approx 0.01$ meter; pressure rise and pressure drop due to the leading edge shock, incident shock, compression wave, expansion fans and re-compression waves are shown here, obtained by current study.

Figure 14: Pressure contour for the shock boundary layer interaction, obtained by current study on 201 $\times$ 201 grid.
Figure 15: Pressure contour for the shock impingement to the solid wall, obtained for inviscid flow by current study on $85 \times 91$ grid.

Figure 16: Comparison of pressure distribution on the wall for viscous and inviscid flow analysis, obtained by current study.

Figure 17: Comparison of computed pressure distribution over the wall with experimental values. The computation is performed in three grid densities.

Figure 18: Comparison of skin friction over the wall with experimental values. The computation is performed in three grid density.