

NUMERICAL SOLUTION OF NAVIER-STOKES EQUATIONS FOR INTERACTION OF SHOCK WAVE WITH LAMINAR BOUNDARY LAYER

M. Hanin, M. Wolfshtein and U. E. Landau
Department of Aeronautical Engineering
Technion - Israel Institute of Technology
Haifa, Israel

Abstract

The interaction of a laminar boundary layer with an impinging shock wave is studied by solving numerically the Navier-Stokes equations of compressible viscous two-dimensional flow. A time-dependent finite difference method of second order accuracy is used. The outer edge of the computational region is placed just outside the viscous layer, and the inviscid relations for oblique shock and for simple compression and expansion waves are taken as boundary conditions at that edge. The computed solutions show reasonable agreement with experimental data.

Introduction

The interaction of a boundary layer with an impinging shock wave has been the subject of numerous investigations for the last 25 years. Until recently, theoretical work on the interaction was based on the boundary-layer approximation, which assumes that streamwise gradients of velocity are much smaller than the gradients across the layer. Studies based on this approximation¹⁻⁴ were successful in elucidating the main features of the flow, in particular the thickening and separation of the boundary layer, the extent of upstream influence, and the pattern of the reflected waves. However, their quantitative results agree well with experiments for relatively low Mach numbers and weak shocks only. It should be expected that for stronger shock waves the boundary layer assumptions are no longer applicable, since a strong shock would produce large streamwise gradients of velocity in the impingement region. To take these effects into account, it is necessary to solve the full Navier-Stokes equations of flow.

Numerical solutions of the Navier-Stokes equations for a laminar boundary layer interacting with an oblique shock wave were obtained by Skoglund and Gay⁵ (1969) and by McCormack⁶ (1971), using modified Lax-Wendroff finite difference techniques. In both solutions the outer edge of the computational mesh was placed far away from the boundary layer so that the edge would not be intersected by reflected waves. As a consequence, a large part of the mesh was occupied by essentially inviscid flow, and relatively few mesh cells were left in the more important viscous-flow zone.

In the present study, the Navier-Stokes equations are solved numerically for the interaction of a laminar boundary layer with an impinging shock wave, using a scheme which allows to fit the computational mesh more closely to the region of viscous flow. The outer edge of the mesh is placed just outside the viscous layer, and the inviscid relations for oblique shock and for the reflected simple compression and expansion waves are taken as boundary conditions at that edge. Since the mesh does not contain much of the inviscid field,

it covers more densely the viscous layer in which the Navier-Stokes equations have to be solved accurately. This scheme is similar to those employed by Allen and Cheng⁷ and by Carter⁸ in solutions not involving shock waves. The present work indicates that the scheme is applicable also to the boundary layer - shock wave interaction.

Differential and Finite-Difference Equations

The differential equations of continuity, momentum (Navier-Stokes) and energy for compressible viscous flow in two dimensions may be stated in vector form as

$$\frac{\partial W}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = S \quad (1)$$

where

$$W = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ E \end{bmatrix}, \quad F = \begin{bmatrix} \rho u \\ p + \rho u^2 \\ \rho uv \\ u(E+p) \end{bmatrix}$$

$$G = \begin{bmatrix} \rho v \\ \rho uv \\ p + \rho v^2 \\ v(E+p) \end{bmatrix}$$

$$S = \frac{1}{Re} \frac{\partial}{\partial x} \begin{bmatrix} 0 \\ \sigma_x \\ \tau \\ \frac{\mu}{Pr} \frac{\partial T}{\partial x} + u\sigma_x + v\tau \end{bmatrix} + \frac{1}{Re} \frac{\partial}{\partial y} \begin{bmatrix} 0 \\ \tau \\ \sigma_y \\ \frac{\mu}{Pr} \frac{\partial T}{\partial y} + u\tau + v\sigma_y \end{bmatrix} \quad (2)$$

$$\begin{aligned} \tau &= \mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \\ \sigma_x &= \frac{2}{3} \mu \left(2 \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} \right) \\ \sigma_y &= \frac{2}{3} \mu \left(2 \frac{\partial v}{\partial y} - \frac{\partial u}{\partial x} \right) \\ E &= \rho \left(\frac{T}{\gamma} + \frac{u^2 + v^2}{2} \right) \end{aligned} \quad (3)$$

Here the variables are nondimensional. The velocity components u and v , the density ρ and the viscosity μ refer to the free-stream values u_0 , ρ_0 and μ_0 respectively, the pressure p is referred to $\rho_0 u_0^2$ and the temperature T to u_0^2/c_p . The coordinates x and y are referred to the shock impingement distance x_s from the leading edge. The Reynolds number is defined by $Re = \rho_0 u_0 x_s / \mu_0$, the Prandtl number is denoted Pr , and γ is the ratio of specific heats.

In addition to the differential equations (1), we have the equation of state

$$p = \frac{\gamma-1}{\gamma} \rho T \quad (4)$$

and Sutherland's law relating viscosity to the temperature.

To obtain a numerical solution, the differential equations (1) were replaced by finite-difference formulae. The method chosen was that of Brailovskaya⁹. In this method each time step is split into two consecutive stages, giving

$$\begin{aligned} \bar{w}_{m,n}^{t+1} &= w_{m,n}^t - \Delta t \left[\frac{1}{2\Delta x} (F_{m+1,n}^t - F_{m-1,n}^t) \right. \\ &\quad \left. + \frac{1}{2\Delta y} (G_{m,n+1}^t - G_{m,n-1}^t) - S_{m,n}^t \right] \\ \bar{w}_{m,n}^{t+1} &= w_{m,n}^t - \Delta t \left[\frac{1}{2\Delta x} (\bar{F}_{m+1,n}^{t+1} - \bar{F}_{m-1,n}^{t+1}) \right. \\ &\quad \left. + \frac{1}{2\Delta y} (\bar{G}_{m,n+1}^{t+1} - \bar{G}_{m,n-1}^{t+1}) - S_{m,n}^t \right] \end{aligned} \quad (5)$$

where the bar denotes intermediate values between the two stages. The terms $S_{m,n}^t$ are calculated by central differences.

The stability condition for this system of equations was found by Carter⁸ to be

$$\begin{aligned} \Delta t \leq \min \left\{ \left[\frac{|u|}{\Delta x} + \frac{|v|}{\Delta y} + \sqrt{\frac{a^2}{(\Delta x)^2} + \frac{a^2}{(\Delta y)^2}} \right]^{-1}, \right. \\ \left. \frac{Pr Re}{2\gamma} \frac{\rho}{\mu} \left[\frac{1}{(\Delta x)^2} + \frac{1}{(\Delta y)^2} \right]^{-1} \right\} \end{aligned} \quad (6)$$

where a is the local speed of sound (referred to u_0).

Boundary and Initial Conditions

The computational region is shown in Fig. 1.

The upstream edge is located well ahead of the shock impingement point so that the flow there is practically not affected by the interaction. The values of the variables at the upstream edge were taken from an approximate theory of boundary layer on a flat thermally-insulated plate in supersonic stream.

$$l_1 = 200 \delta_0$$

$$l_2 = 2 \delta_0$$

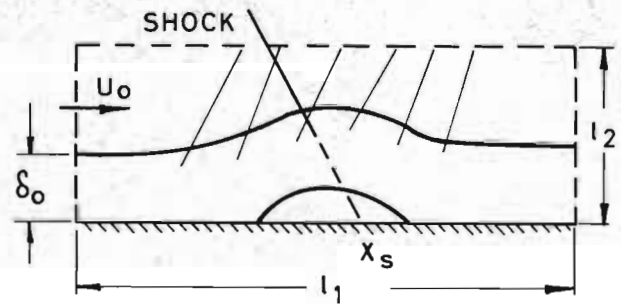


FIG. 1 - REGION OF SOLUTION

The outer edge of the computational region is placed in essentially inviscid flow. The edge is crossed by the shock wave and by simple compression and expansion waves which emerge from the interacting boundary layer. The simple waves are left-running characteristics, and their direction is related to the local Mach number M by

$$\delta = \text{artg} \left(\frac{v}{u} \right) + \text{artg} \left(\frac{1}{\sqrt{M^2-1}} \right) \quad (7)$$

In a simple wave flow all variables have zero gradients in the characteristic direction. This requirement was used as boundary condition at the outer edge. In applying the condition numerically an interpolation had to be made, since characteristics drawn from mesh points of the $(N-1)$ -th row do not in general pass through mesh points of the N -th row.

The impinging shock wave was specified by prescribing the pressure jump ratio of the shock at the outer boundary. The pressure ratio and the Mach number ahead of the shock determine, by the Rankine-Hugoniot relations, the jump ratios of the remaining variables. To reduce numerical fluctuations in the solution, the pressure at two neighbouring points ahead of the shock was equalized, and the pressure jump was spread along two cells.

On the adiabatic wall, the boundary conditions are

$$u = 0, \quad v = 0, \quad \frac{\partial T}{\partial y} = 0, \quad y = 0 \quad (8)$$

The values of temperature and pressure at the wall were extrapolated from inner field points using second-degree polynomials. In the polynomial for the temperature, the adiabatic wall condition was taken into account. In extrapolating the pressure it was found expedient to assume a zero normal

gradient of pressure at the wall, since a better numerical stability was then obtained.

The downstream edge of the computational mesh is placed far enough from the interaction region so that the pressure is practically constant there and the streamwise gradients of flow variables are very small. The boundary conditions imposed at that edge require that the streamwise gradients vanish there.

The initial values were chosen by substituting the undisturbed upstream values over the whole field, except of a region below the shock where a separation bubble was introduced in order that the initial state would have a qualitative resemblance to the final steady flow.

Some Numerical Details

A mesh of 20×50 cells was used in the numerical solution. The width to length ratio of each cell was $\Delta y/\Delta x = 1/40$. The time step intervals Δt were $\frac{1}{2}$ of the maximum values indicated by the stability condition (6).

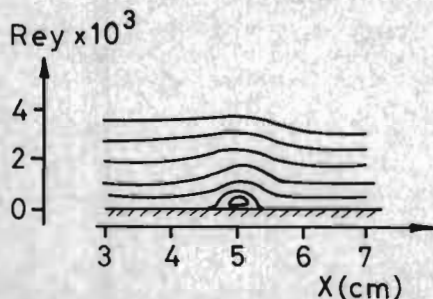
During the first 300 time steps an artificial diffusion term ⁵ was added to the viscous stress terms $S_{m,n}^L$ in order to stabilize the transition from the m,n initial state. In the subsequent steps (from 301-th on) artificial diffusion was not used.

The final steady-state solution was assumed to be reached when the root-mean-square time derivative of all the variables over the entire field decreased to the order of magnitude of the truncation errors as estimated by a Taylor-series analysis of the difference equations. It took about 500 time steps to reach the final solution.

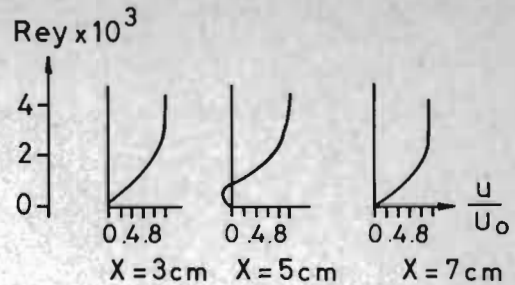
The calculations were performed on the IBM 370/165 digital computer at the Technion. The computer time for each time step was about 0.5 sec.

Results and Discussion

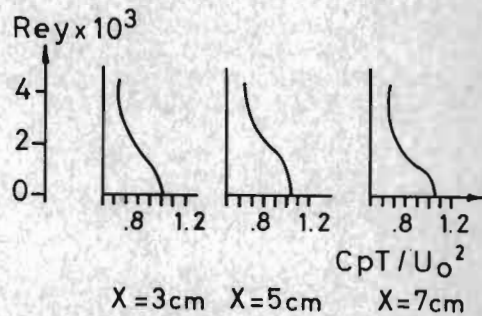
Numerical solutions for the boundary layer - shock wave interaction were computed by the method



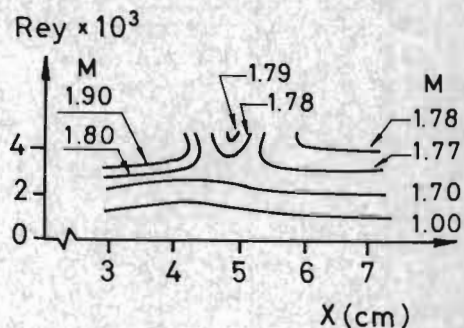
2a. STREAMLINES



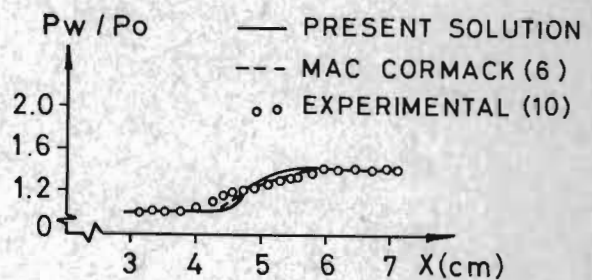
2 b. VELOCITY PROFILES



2c. TEMPERATURE PROFILES



2d. MACH NUMBER CONTOURS

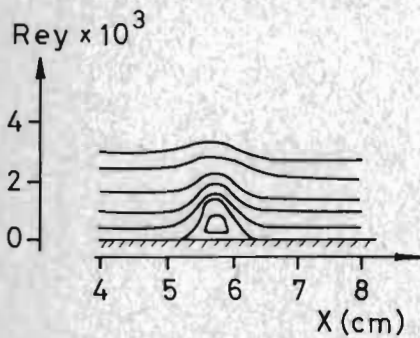


2e. PRESSURE ON WALL

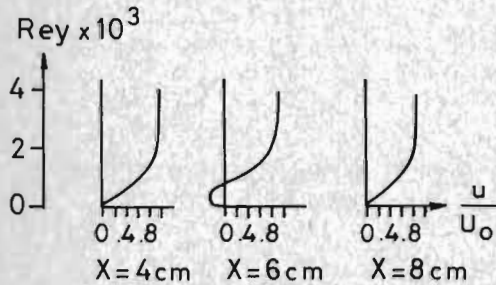
FIG 2. RESULTS FOR $P_\infty/P_0 = 1.40$, $M_0 = 2.0$, $Re = 2.96 \times 10^5$, $X_s = 4.95$ cm

described above for the following two cases: Mach number $M_0 = 2.0$, pressure ratio $p_\infty/p_0 = 1.40$ and 1.91, impingement Reynolds number $Re = u_0 \rho_0 x_s / \mu_0 = 2.96 \times 10^5$ and 3.29×10^5 respectively. Here p_∞ is the pressure downstream of the reflected waves, and the subscript 0 denotes values in the undisturbed stream. The two cases were chosen in order to compare the results with experimental laminar-flow data reported by Hakkinen et al.¹⁰ and with a numerical solution obtained for $p_\infty/p_0 = 1.4$ by MacCormack⁶. The resulting streamlines, velocity profiles, temperature profiles, lines of constant Mach number, and wall pressure distributions are shown in Figs. 2 and 3.

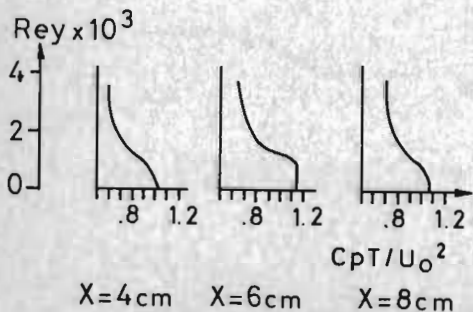
For both values of the pressure ratio, the boundary layer is seen to separate and reattach, in agreement with experimental observations. The extent of the recirculation bubble and the magnitude of the reverse velocity are larger for the



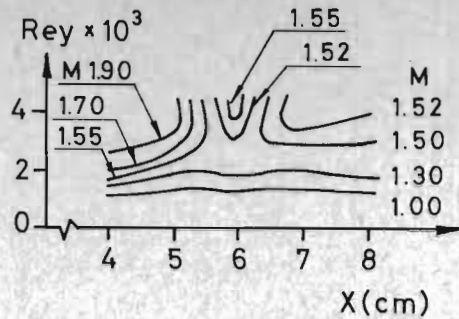
3a. STREAMLINES



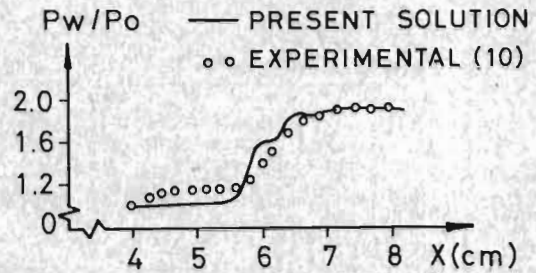
3b. VELOCITY PROFILES



3c. TEMPERATURE PROFILES



3d. MACH NUMBER CONTOURS



3e. PRESSURE ON WALL

FIG. 3. RESULTS FOR $p_\infty/p_0 = 1.91$, $M_0 = 2.0$, $Re = 3.29 \times 10^5$, $X_s = 5.85$ cm

stronger shock. The temperature profiles have a plateau in the recirculating region, as should be expected.

The calculated pressure on the wall rises more steeply than the experimental values. Further work is being done to clarify the cause of this behaviour.

On the whole, the results are realistic and agree reasonably with experimental data. It may be concluded that the Brailovskaya finite difference scheme used in conjunction with the simple-wave boundary condition gives a suitable method for computing the shock wave - laminar boundary layer interaction.

References

1. Lighthill, M.J., "On Boundary Layers and Upstream Influence", Proc. Roy. Soc. (London) Ser. A, Vol. 217, pp. 478-507 (1953).

2. Lees, L. and Reeves, B.L., "Supersonic Separated and Reattaching Laminar Flows, Pt. 1: General Theory and Application to Adiabatic Boundary Layer - Shock Wave Interaction", AIAA J., Vol. 2, No. 11, pp. 1907-1920 (1964).
3. Reyhner, T.A. and Flugge-Lotz, I., "The Interaction of a Shock Wave with a Laminar Boundary Layer", Int. J. of Non-Linear Mech., Vol. 3, No. 2, pp. 173-199 (1968).
4. Murphy, J.D., "A Critical Evaluation of Analytic Methods for Predicting Laminar-Boundary-Layer Shock Wave Interactions", NASA Symp. on Analytical Methods in Aircraft Aerodynamics, NASA SP-228, pp. 515-531 (1969).
5. Skoglund, V.J. and Gay, B.D., "Improved Numerical Techniques and Solution of a Separated Interaction of an Oblique Shock Wave and a Laminar Boundary Layer", New-Mexico Univ. Rep. ME-41 (69) S-068 (1969).
6. MacCormack, R.W., "Numerical Solution of the Interaction of a Shock Wave with a Laminar Boundary Layer", Proc. 2-nd Internat. Conf. on Numerical Methods, Springer-Verlag, pp. 151-163 (1971).
7. Allen, J.S. and Cheng, S.I., "Numerical Solutions of the Compressible Navier-Stokes Equations for the Laminar Near Wake", Phys. of Fluids, Vol. 13, No. 1, pp. 37-52 (1970).
8. Carter, J.E., "Numerical Solution of the Navier-Stokes Equations for the Supersonic Laminar Flow over a Two-Dimensional Compression Corner", NASA TR-R 385 (1972).
9. Brailovskaya, I.Y., "A Difference Scheme for Numerical Solution of the Two-Dimensional Non-steady Navier-Stokes Equations for a Compressible Fluid", Sov. Phys. Doklady, Vol. 10, No. 2, pp. 107-110 (1965).
10. Hakkinen, R.J., Greber, I., Trilling, L., and Abarbanel, S., "The Interaction of an Oblique Shock Wave with a Laminar Boundary Layer", NASA Memo 2-18-59W (1959).

D I S C U S S I O N

B.G. Newman (McGill University, Montreal, Canada):
Is it necessary to develop a time-dependent solution?

M. Hanin: Several steady-state iterative solutions of Navier-Stokes equations were developed for incompressible flows. However, the steady-state methods of solution were either equivalent to time-dependent methods or had worse stability and convergence properties (1). Probably this would also be the case regarding compressible flows.

Reference:

(1) P.J. Roache, Computational Fluid Dynamics, Hermosa Publishers, Albuquerque, N.M., 1972.