Calculation Methods for Compressible Turbulent Boundary Layers—1976
Calculation Methods for Compressible Turbulent Boundary Layers—1976

Dennis M. Bushnell
Aubrey M. Cary, Jr.
and Julius E. Harris

Prepared by
NASA Langley Research Center

Scientific and Technical Information Office
National Aeronautics and Space Administration
1977
PREFACE

Calculation procedures for nonreacting compressible two- and three-dimensional turbulent boundary layers are reviewed. A summary of integral, transformation, and correlation methods, as well as finite-difference solutions of the complete boundary-layer equations is included. Alternative numerical solution procedures are examined, and both mean field and mean turbulence field closure models are considered. A discussion of physics and related calculation problems peculiar to compressible turbulent boundary layers is included. A listing of available solution procedures (finite-difference, finite-element, and weighted-residual methods) is provided. Detailed consideration is given to influence of compressibility, low Reynolds number, wall blowing, and pressure gradient upon mean field closure constants.

The information contained in this publication was presented at the 1976 Von Kármán Institute for Fluid Dynamics Lecture Series "Compressible Turbulent Boundary Layers," March 1-5, 1976, Rhode-St.-Genèse, Belgium.
# CONTENTS

**PREFACE** ........................................ iii

**INTRODUCTION** ..................................... 1

**SYMBOLS** ......................................... 1

**MAJOR GENERAL REFERENCES** ......................... 5

**EQUATIONS AND CLOSURE METHODS** ................. 6
- Governing Differential Equations ............... 6
- Closure Methods .................................. 7

**FLOW PHENOMENA PECULIAR TO CALCULATION OF COMPRESSIBLE TURBULENT BOUNDARY LAYERS** ........... 9
- Normal Pressure Gradient .......................... 9
- Influence of Compressibility and Density Fluctuation Terms ................. 10
- Comparison of Mean Static Temperature and Turbulent Kinetic Energy .......... 11
- Precursor Transition Effect ....................... 12
- Large $p'$ Levels .................................. 13
- Definition of Boundary-Layer Thickness .......... 13
- Energy Loss Via Acoustic Waves .................. 14

**TRADITIONAL PREDICTION METHODS** ............... 14
- Transformations .................................... 15
- Correlation Methods ............................... 17
- Integral Methods .................................. 23

**NUMERICAL SOLUTION PROCEDURES** ................. 27
- Solution Techniques .............................. 27
- Systems of Independent Variables ............... 29

**MEAN FIELD CLOSURE** ............................. 31
- Wall Damping Constant ............................ 33
- Prandtl Wall Constant ............................. 35
- Outer or Wake Constant ........................... 35
- Turbulent Prandtl Number ......................... 37
- Intermittency Normal to the Wall ............... 38

**APPLICATIONS OF MEAN FIELD CLOSURE METHODS** ........ 38
- The Case of the Nozzle Wall Turbulent Boundary Layer ................. 38
- Influence of Wall Blowing Upon Skin Friction ................. 40
- Transitional Flow Calculations ................. 40
- Additional "Nontypical" Applications of Mean Field Closure Methods ........ 41

**NONEQUILIBRIUM AND MEAN TURBULENT FIELD CLOSURE** ........ 44
- Physical Problem ................................ 44
- Equations and Modeling for Compressible Nonequilibrium Flows ................. 45
INTRODUCTION

The design of missiles, manned and unmanned entry vehicles, and transportation systems capable of speeds in the transonic and supersonic regimes requires detailed information on such boundary-layer quantities as skin friction, aerodynamic heating, and viscous displacement thickness and mass flow. Typical components requiring such design information include nacelles, control surfaces, turbomachinery blading, nozzles, airfoils and fuselage, inlets, and combustors. Detailed boundary-layer predictions are also needed for the design of facilities used in experimental investigations.

The various stages of design demand boundary-layer information in increasing detail. For this reason and also because flow complexity varies considerably (depending upon the design and component), a hierarchy of calculation procedures has been developed over the years, ranging from simple, "back of the envelope" methods to complex, numerical approaches which require use of the largest digital computers. In the present paper this entire gamut of methods is reviewed, with emphasis on the more complete procedures which solve numerically the partial-differential equations governing boundary-layer motion and on the influence of conditions usually encountered in practice (such as pressure gradient, mass injection, and low Reynolds number) upon the necessary closure "constants" used in representing the turbulent shear. Also included is a detailed discussion of some physics and resultant calculation problems which are peculiar to compressible turbulent boundary layers. The basic purpose of this volume is to provide a ready reference and introduction to the various procedures currently available for calculation of compressible turbulent boundary layers. Therefore, also included is a listing of available methods of the more complete type and some discussion of the various alternate numerical procedures which can be used for solving the nonlinear partial-differential equations governing fluid motion in compressible turbulent boundary layers. This review does not include detailed consideration of time-dependent boundary layers, relaminarization, and heterogeneous or chemically reacting flows.

**SYMBOLS**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Van Driest damping factor (eq. (41))</td>
</tr>
<tr>
<td>$A^*$</td>
<td>$= A u_T / v$</td>
</tr>
<tr>
<td>$a$</td>
<td>speed of sound</td>
</tr>
<tr>
<td>$a_1$</td>
<td>$= \overline{u'v'} / 2e$</td>
</tr>
<tr>
<td>$C_f$</td>
<td>skin friction coefficient, $\tau_w / \frac{1}{2} \rho_e u_e^2$</td>
</tr>
<tr>
<td>$C_{f, 0}$</td>
<td>skin friction coefficient without wall blowing</td>
</tr>
<tr>
<td>$c$</td>
<td>wing chord</td>
</tr>
</tbody>
</table>
\( c_p \) specific heat at constant pressure
\( d \) unit length
\( e \) turbulent kinetic energy, \( \frac{1}{2}(u'^2 + v'^2 + w'^2) \)
\( F \) mass flow parameter, \( \rho_w v_w / \rho_e u_e \)
\( F_G, F_{Rx}, F_{Ro} \) functions from Spalding and Chi (ref. 81)
\( G = u_2 / u_2, e \)
\( H \) total enthalpy, \( h + u'^2 \); also form factor, \( \delta^* / \theta \)
\( \tilde{H} = u_3 / u_1, e \) (eq. (95))
\( H_i = \delta_i / \theta_i \)
\( h \) static enthalpy
\( h_1, h_2, h_3 \) metric coefficients
\( K \) Prandtl constant
\( \hat{K} \) factor for variable grid spacing, \( \Delta n_{n+1} / \Delta n_n \)
\( L \) reference length; also dissipation length scale (fig. 48)
\( l \) mixing length
\( M \) Mach number
\( \dot{m} \) mass flow
\( N_{Nu} \) Nusselt number, \( (N_S t)(N_{Pr})(R_e, x) \)
\( N_{Pr} \) molecular Prandtl number
\( N_{Pr, T} \) total turbulent Prandtl number, \( \epsilon / \kappa_T \)
\( N_{Pr, t} \) static turbulent Prandtl number, \( \epsilon / \kappa_t \)
\( N_{St} \) Stanton number, \( \dot{q}_w / \rho_e u_e c_p(T_{aw} - T_w) \)
\( p \) pressure
\( p^+ \) pressure gradient parameter, \( (\nu u_e / u_t^3)(d u_e / dx) \)
\( \dot{q} \) wall heat transfer rate
R  Reynolds number, $\rho u/\mu$; also universal gas constant

$r_c$  transverse radius of curvature

$r_N$  nose radius

$T$  temperature

$u, v, w$  velocity components in x-, y-, and z-directions

$u_i$  general velocity notation ($i = 1, 2, 3$)

$u^+$  $= u/u_T$

$u^*$  Van Driest's generalized velocity

$u_T$  friction velocity, $(\tau_w/\rho_w)^{1/2}$

$V$  total velocity vector

$x, y, z$  Cartesian coordinates

$x_i$  curvilinear coordinates ($i = 1, 2, 3$)

$y^+$  $= yu_T/\nu$

$\alpha$  Clauser constant; also angle of attack

$\beta_K$  pressure gradient parameter, $((\delta_i/\tau_w)(dp/dx))$

$\Gamma$  intermittency function

$\Gamma_x$  streamwise intermittency function

$\Gamma_y$  normal intermittency function

$\gamma$  ratio of specific heats

$\delta$  boundary-layer thickness

$\delta^*$  displacement thickness, $\int_0^\delta \left(1 - \frac{\rho u}{\rho_e u_e}\right) dy$

$\delta^+$  $= \delta u_T/\nu_w$

$\epsilon$  dynamic eddy viscosity

$\zeta$  transformed normal coordinate, Crocco (eq. (91))

$\eta$  transformed normal coordinate, Levy-Lees (eq. (32)); also transformed transverse coordinate for Crocco variables (see fig. 53)
\[ \Theta = \frac{T}{T_e} \]

\[ \Theta = \text{momentum thickness, } \int_0^\delta \frac{\rho u}{\rho_e u_e} \left( 1 - \frac{u}{u_e} \right) dy \]

\[ \kappa_T = \frac{v'\eta'}{\partial H/\partial y} \]

\[ \kappa_t = \frac{v'\eta'}{\partial h/\partial y} \]

\[ \mu \] molecular viscosity

\[ \nu \] kinematic viscosity

\[ \xi \] transformed streamwise coordinate (eq. (31)); also \( x_1 \)-coordinate for Crocco variables

\[ \rho \] density

\[ \sigma \] kinematic eddy viscosity (eq. (40))

\[ \tau \] shear stress

\[ \phi \] shear function (eq. (92))

\[ \phi \] spreading angle (fig. 5)

\[ \varphi \] circumferential angle

\[ \psi \] stream function

\[ \omega \] vorticity

Subscripts:

\[ aw \] adiabatic wall

\[ bl \] boundary layer

\[ cr \] critical value

\[ e \] local edge of boundary layer

\[ eff \] effective value

\[ i \] incompressible

\[ i, j, k \] indices

\[ L \] reference length

4
max  maximum value
n  index integer
o  stagnation value
p  region of peak $v'^2$
 t  total
tr transition value
vo based on virtual origin location
x  based on distance coordinate $x_i$
$x_i$ coordinate direction ($i = 1, 2, 3$)
w  wall value
$\theta$ based on momentum thickness
$\infty$ free stream

A prime denotes a fluctuating quantity.
A bar indicates a mean quantity.

MAJOR GENERAL REFERENCES

The major reference books pertinent to the calculation of compressible turbulent boundary layers were mainly written in two distinct time frames: late fifties to early sixties (refs. 1 to 5) and late sixties to early seventies (refs. 6 to 12). Of these works, the ones most useful in the preparation of the present volume on calculation methods included references 2, 5, 8, 9, and 12. Several review and background articles are also available (e.g., refs. 13 to 25). All of these were quite valuable, especially the papers of Reynolds (refs. 19 and 20) and Bradshaw (ref. 22). Another category of general references is conference proceedings (e.g., refs. 26 to 30). These are excellent sources, particularly for comparisons between data and theory (especially refs. 28 and 30). Reference 26 contains many of the fundamental concepts, such as the Morkovin hypothesis, which are the foundation for the current generation of compressible calculation procedures. A final category of general references includes reviews of available data (e.g., refs. 11 and 31 to 37), which are especially important for evaluation of test cases suitable to "calibrate" the various turbulence closure constants.
EQUATIONS AND CLOSURE METHODS

Governing Differential Equations

The basic differential equations governing compressible turbulent boundary-layer flow are (1) a statement of the conservation of mass, (2) momentum equations obtained from the Navier-Stokes equations, and (3) an expression for the conservation of energy. Also needed for solution is an equation of state (ideal gas assumed herein) and equations for molecular transport properties.

Given this set of equations it is conceptually possible to integrate forward in time on a computer and, given sufficient grid resolution, obtain the turbulent motions "exactly" with very little empiricism. However, as noted in reference 12 and elsewhere, the computer capacity currently available falls several orders of magnitude short of the capability needed for such a solution. Therefore, one must resort to the usual Reynolds averaging, where the flow is represented by a time mean and an instantaneous fluctuation; for example,

\[ u = \bar{u} + u' \quad \quad v = \bar{v} + v' \]

where the following averaging rules apply:

\[ \bar{u} v' = 0 \quad \quad u'v' \neq 0 \]

To simplify the problem for ease of solution, with very little loss of accuracy (ref. 38), order of magnitude arguments are made (usually referred to as the boundary-layer assumptions); that is,

\[ x >> y \quad u >> v \quad \frac{\partial}{\partial x} << \frac{\partial}{\partial y} \]

The resultant set of governing equations in surface curvilinear coordinates for the steady, two-dimensional boundary-layer flow (\( \delta/r_c << 1 \)) of a compressible ideal gas (in the turbulent state) are

Conservation of mass:

\[ \frac{\partial (\bar{\rho} \bar{u})}{\partial x} + \frac{\partial (\bar{\rho} v)}{\partial y} = 0 \quad (1) \]

Longitudinal momentum:

\[ \bar{\rho} \frac{\partial \bar{u}}{\partial x} + \bar{\rho} v \frac{\partial \bar{u}}{\partial y} = - \frac{\partial \bar{P}}{\partial x} + \frac{\partial}{\partial y} \left[ \mu \frac{\partial \bar{u}}{\partial y} - \frac{(\bar{p} \bar{v})' u'}{\bar{v}'} \right] \quad (2) \]

Normal momentum:

\[ \frac{\partial \bar{P}}{\partial y} = - \frac{\partial}{\partial y} \left[ (\bar{p} \bar{v})' v' \right] \quad (3) \]
Conservation of energy:

\[
\bar{\rho} \frac{\partial \bar{H}}{\partial x} + \bar{\rho} \frac{\partial \bar{H}}{\partial y} = \frac{\partial}{\partial y} \left( \bar{\mu} \frac{\partial \bar{H}}{\partial y} + \left( N_{Pr} - 1 \right) \frac{\partial \bar{u}^2}{2 \partial y} \right) - \frac{\left( \bar{\rho} \bar{v} \right) \bar{v}'}{\partial y} - \bar{u} \left( \frac{\left( \bar{\rho} \bar{v} \right) \bar{u}'}{\partial y} \right)
\]  

(4)

Detailed discussion and derivation of these equations are available in references 2, 12, 39, 40, and 41. The equations are also given in references 42 and 43, in terms of mass-weighted dependent variables.

There are two points to be made concerning these basic equations (eqs. (1) to (4)). First of all, in the averaging process the usual new unknowns have appeared (shown underlined) which account for the turbulent fluxes of momentum and energy. The specification of these quantities, in terms of known parameters, comprises the major difficulty in the calculation of compressible turbulent boundary layers and is usually termed the "closure problem." The other point concerns the term \( \bar{\rho} \bar{v} \). Expanded out, this term is

\[
\bar{\rho} \bar{v} = \bar{\rho} \bar{v} + \bar{\rho} \bar{v'}
\]

(5)

Since \( \bar{\rho} \bar{v'} \) appears in the equations with normal or \( y \)-derivatives, the term is not negligible and must be accounted for. Fortunately, \( \bar{\rho} \bar{v'} \) always appears with \( \bar{\rho} \bar{v} \), and therefore, a new definition of \( \bar{\rho} \) can be used

\[
\bar{v} = \bar{v} + \frac{\bar{\rho} \bar{v'}}{\bar{\rho}}
\]

(6)

(which is actually a mass-weighted variable), and the influence of \( \bar{\rho} \bar{v'} \) can be included implicitly in the solution. This approach is satisfactory as long as the actual \( \bar{v} \) value is not required. If \( \bar{v} \) values must be computed, some model of \( \bar{\rho} \bar{v'} \) is obviously necessary. This inclusion of \( \bar{\rho} \bar{v'} \) is an important issue. If the \( \bar{\rho} \bar{v} \) term were not handled in the manner shown in equations (2) and (4), then two Reynolds stress terms appear: \( \bar{u} \bar{p} \bar{v'} \) and \( \bar{u} \bar{u} \bar{v'} \). Morkovin, in reference 44, shows that \( \bar{u} \bar{p} \bar{v'} / \bar{\rho} \bar{u} \bar{v'} \) (or the ratio of the compressibility term to the usual low-speed Reynolds stress term) can be of the order of 0.6 to 1.0 for a supersonic boundary layer, and therefore \( \bar{\rho} \bar{v'} \) would have to be accurately modeled (an extremely difficult task).

Closure Methods

In the present paper the usual breakdown of closure procedures (e.g., ref. 45) into (1) simple or zeroth-order methods, (2) first-order or mean field closure methods, and (3) second-order or mean turbulence field closure methods is followed.

The zeroth-order case consists of two major subcategories: integral approaches and empirical laws, such as \( C_f \) correlations. In the most general of these approaches, the integral methods, equations (1) to (4) are formally integrated in the normal or \( y \)-direction. As a result of this procedure the unknown turbulent flux terms disappear, but their influence is still present in that profiles must be supplied (assumed, obtained from data, etc.), and these profiles are influenced to a great extent by the turbulence induced fluxes. There-
fore, the simple or zeroth-order procedures are characterized by a requirement for substantial amounts of empiricism.

In the mean field closure approaches the partial-differential equations are solved directly (eqs. (1) to (4)) and the turbulent flux terms are related to mean flow quantities. This approach is nearly correct (and indeed quite exact) for slowly varying flows and over a wide variety of boundary conditions. At the present time this closure approach is tending to supplant the integral approaches in industrial application, primarily as a result of a greater reliability (accuracy of prediction) over a wide range of conditions and routine use of high-speed computers. This increased capability is purchased at the expense of considerably increased computer time (compared with the best of the integral methods).

In mean turbulent field closure, the differential equations (derived from the Navier-Stokes equations, as described in a later section) governing the turbulence flux terms are solved. These new equations involve additional unknowns, but the mean profiles generally are relatively insensitive to the precise details of the modeling in these second-order equations. This approach has actually had limited application to the compressible boundary-layer case (primarily as a result of the success of the mean field methods) and is really needed only in cases where the flow undergoes a sudden change in boundary condition or experiences a large gradient (a situation generally termed nonequilibrium).

The usual range of application of each of these three closure approaches is indicated schematically as follows:

<table>
<thead>
<tr>
<th>EQUILIBRIUM</th>
<th>NEAR EQUILIBRIUM</th>
<th>NONEQUILIBRIUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>• ZERO OR EQUILIBRIUM</td>
<td>• MODERATE DEPARTURES</td>
<td>• SUDDEN APPLICATION OR</td>
</tr>
<tr>
<td>GRADIENTS OF</td>
<td>FROM ZERO OR</td>
<td>REMOVAL OF LARGE</td>
</tr>
<tr>
<td>PRESSURE, WALL</td>
<td>EQUILIBRIUM GRADIENTS</td>
<td>GRADIENTS IN WALL</td>
</tr>
<tr>
<td>TEMPERATURE, AND</td>
<td></td>
<td>TEMPERATURE ,</td>
</tr>
<tr>
<td>WALL INJECTION</td>
<td></td>
<td>PRESSURE, WALL</td>
</tr>
<tr>
<td>• HIGH REYNOLDS NUMBER</td>
<td>• LOW REYNOLDS NUMBER</td>
<td>• TRANSITIONAL FLOWS</td>
</tr>
</tbody>
</table>

**ZEROTH ORDER METHODS**

**MEAN FIELD METHODS**

**MEAN TURBULENCE FIELD METHODS (WITH LENGTH SCALE EQUATION)**

In the present review an operational definition of equilibrium is used, taken from reference 46: [equilibrium refers] "to a layer that has completely settled down after a discontinuity and is developing with its new boundary conditions with no 'memory' of the discontinuity." Succeeding portions of the present report cover these three closure approaches in considerable detail, especially the mean field approaches.
FLOW PHENOMENA PECULIAR TO CALCULATION OF COMPRESSIBLE TURBULENT BOUNDARY LAYERS

Calculation procedures for compressible turbulent boundary layers are based, to a great extent, upon techniques, modeling constants, etc., developed originally for the low-speed case. This section discusses many of the significant differences and new or altered physics which occur in the compressible case, as compared with the low-speed situation. The discussion is intended to aid in evaluating the applicability of low-speed results to the compressible (particularly high Mach number) case and to indicate possible pitfalls and sources of inaccuracy in the calculation of compressible turbulent boundary layers.

Normal Pressure Gradient

Shown in figure 1 are typical static-pressure distributions measured across high Mach number turbulent boundary layers (two cases are shown, helium (ref. 47) and nitrogen (ref. 48)). In both cases the boundary layer involved was on a nozzle wall with only a small local longitudinal pressure gradient, although there are significant free-stream static-pressure variations due to uncanceled Mach waves. (See ref. 48.) Two points are obvious from this figure: the static pressure is not constant across these high Mach number turbulent boundary layers (\( \partial p/\partial y = 0 \) for \( M = 0 \)), and the wall pressure value is greater than the edge value by approximately 50 percent for these cases (\( M_\infty \approx 20 \)).

A possible origin of at least a portion of this nonconstant \( p(y) \) can be readily seen from a simple examination of equation (3) (normal momentum equation)

\[
\frac{\partial \bar{p}}{\partial y} = -\frac{3}{2} \frac{\partial \left( \bar{\rho} v'^2 \right)}{\partial y}
\]

or

\[
\bar{p} + \bar{p} v'^2 \approx \text{Constant}
\]

Evaluating this expression between the region of peak \( v'^2 \) (\( y/\delta = 0(0.1) \)) and \( \bar{p}_w \approx \bar{p}_e \) from fig. 2) and the wall, one obtains

\[
\bar{p}_w = \bar{p}_e + f \rho_e u_e^2 \frac{v'_2}{u_e^2}
\]

where \( \bar{p}_p \approx f \rho_e \) or

\[
\frac{\bar{p}_w}{\bar{p}_e} \approx 1 + f \gamma \frac{M_e^2 v'_2}{u_e^2}
\]

Limited data (e.g., ref. 49) indicate that the nondimensional velocity fluctuation levels in boundary layers are not significantly influenced by Mach number,
and therefore a reasonable value for $\sqrt{\frac{v'^2}{u'_e^2}}$ of 0.06 was used (from low-speed data). Using $f = 0.2$ and $\gamma = 1.4$, equation (10) becomes (to correct order of magnitude)

$$\frac{\rho_w}{\rho_e} = 1 + (10^{-3}) M_e^2$$

(11)

and indicates a possible dominant influence of Mach number upon the nonconstant $p(y)$ phenomenon, although the influence may not be as strong as $M^2$ due to $f$ being a function of $M$. Shown in figure 2 (from ref. 50) is a comparison between this simple prediction (eq. (11)) and most of the available data. The relatively good agreement between the present simple expression (eq. (11)) and the available data (fig. 2) may indicate that at least some of the nonconstant $p(y)$ effect is caused by the turbulence field. This is probably aggravated at high Mach number by the fact that the dynamic pressure associated with the turbulent fluctuations becomes a significant fraction of the static pressure level at high Mach number. Research by Finley (ref. 51) indicates that most of the $P_w > P_e$ problem is due primarily to inviscid disturbances whose detailed influence is modified by the turbulence effect just described.

Calculation experience indicates that the large $\rho$ changes associated with static temperature variations for high Mach number boundary layers (a change in density by a factor of approximately 100 for $M \approx 20$, $T_w + T_t$, and $\gamma = 5/3$) greatly override the $P_w/P_e > 1$ effect and therefore, while interesting, the $P_w/P_e > 1$ effect is not currently considered first order for calculating high Mach number boundary-layer flows.

Influence of Compressibility and Density Fluctuation Terms

There are no definitive, detailed measurements of the complete second-order correlations in a highly compressible flow (including $p'$ and $\rho'$ terms) with which to assess, in a straightforward manner, the influence of compressibility and density fluctuation terms upon closure models used in the calculation of compressible turbulent boundary layers. There is evidently a true Mach number effect on turbulence structure for free flows (ref. 52), at least for the free shear layer case at high Reynolds number with a sustained Mach number difference across the shear layer. However, no important compressibility ($p'$, $\rho'$) influence has ever been isolated for the boundary-layer case (except for $\rho'v'$; see discussion following eq. (6)). Detailed consideration of this area is beyond the scope of the present paper; the purpose here is merely to warn the reader that the following arguments concerning the relative absence of noticeable compressibility effects are deductive. There does not yet exist a definitive set of measurements to completely lay to rest the question of possible compressibility effects upon turbulence structure, although Morkovin's arguments (ref. 26, pp. 367-380), which were based upon hot-wire data up to $M \approx 5$ where $p'$ effects were neglected, have proved thus far to be correct.

**Fluctuating Mach number.**- One method of evaluating the possible influence of compressibility upon turbulence structure is to examine the magnitude of the fluctuating Mach number $M'$; that is,
\[ M' \approx \frac{u'}{\bar{a}} = \frac{M_a'}{\bar{a}} \]

For the case of maximum \( \rho' \) (i.e., \( T_w + T_{t,e} \) and \( T_t(y) = \text{Constant} \)) near the edge of the sublayer

\[ \frac{u'}{\bar{a}} = \frac{u'}{u_e} M_e \sqrt{\frac{T_e}{T}} \]

and for \( \gamma = 1.4 \)

\[ \frac{u'}{\bar{a}} \approx \frac{u'}{u_e} \frac{M_e}{\sqrt{1 + \frac{M_e^2}{5}}} \]

which is only of the order of 0.2 for high \( M_e \) and \( u'/u_e \approx 0.1 \). Therefore the main term in equation (12) is the second one \( \langle \rho' \rangle \frac{u'}{\bar{a}} \), which, for \( \alpha'/\bar{a} = 0(0.1) \), can be of the order of 1 or greater for high \( M_e \). Therefore, for the high hypersonic case the fluctuating Mach number can be of the order of 1 and compressibility effects may become important for accurate turbulence modeling.

Presence of \( \rho' \) terms in Reynolds stress expression.- In equation (2) the complete Reynolds stress term is \( \langle \rho \nu \rangle \frac{u'}{\bar{a}} \). Expanding this term (ref. 13) one obtains

\[ \langle \rho \nu \rangle \frac{u'}{\bar{a}} = \bar{\nu} \frac{u'}{\bar{a}} \nu' + \bar{\nu} \rho' \frac{u'}{\bar{a}} \nu' + \rho' \frac{u' \nu'}{\bar{a}} \]

Using the usual order of magnitude arguments, \( \nu' < \bar{\nu} \) and \( \rho' < \bar{\rho} \),

\[ \langle \rho \nu \rangle \frac{u'}{\bar{a}} \approx \bar{\nu} \frac{u'}{\bar{a}} \nu' \]

within an accuracy of approximately 20 percent. Calculation experience (e.g., ref. 53) indicates that the \( \rho' \) terms in the Reynolds stress equation (eq. (15)) are only important for the case where \( \varepsilon/\mu \) is relatively small \( (\varepsilon/\mu < 100) \). Therefore, for the calculation of compressible turbulent boundary layers the Reynolds stress term generally assumes the same form as the low-speed case \( \langle \bar{\rho} \frac{u' \nu'}{\bar{a}} \rangle \). This does not imply, however, that \( \frac{u' \nu'}{\bar{a}} \) can be modeled in the same manner (or has similar values) as in the low-speed case.

Comparison of Mean Static Temperature and Turbulent Kinetic Energy

As will now be shown, there is a possibility that, for high Mach numbers, the static temperature can be of the same order of magnitude as the temperature equivalence of the energy associated with the turbulent velocity fluctuations. The basic problem is most easily seen by beginning with the expression for instantaneous total enthalpy

\[ H = c_p T_t = c_p T + \frac{u^2}{2} + \frac{v^2}{2} + \frac{w^2}{2} \]
The use of Reynolds time averaging (and recognizing that $\bar{w} = 0$ for two-dimensional boundary layers) yields

\[ c_p \bar{\tau} = c_p \bar{\tau} + \frac{\bar{u}^2}{2} + \frac{\bar{v}^2}{2} + \frac{\bar{w}^2}{2} \]

or

\[ \bar{\tau} = \bar{\tau} - \frac{\bar{u}^2}{2c_p} - \frac{\bar{v}^2}{2c_p} - \frac{\bar{w}^2}{2c_p} \]

In order to determine the static temperature from a boundary-layer solution, some form of equation (19) is generally employed, using terms I and II only. Term III has been almost universally neglected. For $M_e = 22$, $T_w + T_{e}$, and $M_{local} = 10$ (i.e., at $y/\delta \approx 0.5$) in air, the usual terms (I and II) yield $T/T_t \approx 0.0476$. Assuming $U'/U_e = 0.05$, the additional (high $M$) term (III) yields a value of approximately 0.0075, or approximately 16 percent of the value of terms I and II combined. Similar comparisons over the Mach number range are shown in figure 3.

What this discussion indicates is that for very high Mach numbers, $\bar{\tau} \approx \bar{u}^2/2c_p$ and $T_{static}$ can become of the same order as the temperature equivalence of the energy associated with the turbulent motions. Since the intensity of the turbulence is not generally known to within an accuracy of approximately 20 to 30 percent and $T_{static}$ can depend upon the square of the intensity (eq. (19)), the accuracy of the computed mean density field for extremely high Mach number turbulent boundary layers could be quite poor.

Precursor Transition Effect

The precursor transition effect is characterized by the existence of large-scale disturbances (and subsequent breakdown into turbulence) in the outer region of compressible boundary layers far upstream of the nominal wall transition point. (See fig. 4 for typical schlieren photographs of this behavior.) This phenomenon is quite commonly observed (e.g., refs. 54 to 56). The spreading rates and possible structure of these disturbances were examined in reference 57 (see fig. 5, taken from ref. 57), while an attempt to model their influence in a mean field closure procedure is reported in reference 58. The major impact of this phenomenon upon the calculation of turbulent boundary layers is that at the nominal wall transition location, the outer portion of the profile is already transitional and turbulent in nature (see fig. 6, taken from ref. 58), and therefore, the usual procedure of starting the calculation at the beginning of the wall transition location with a laminar profile cannot be followed.

A simple method of including this phenomenon is to start the calculation upstream of the wall transition location (with a laminar-like profile) where the transition bursts first initiate. (See ref. 59.) Figure 7 shows a possible further manifestation of this effect - an increase in surface heating upstream of the nominal wall transition point as usually defined (taken from ref. 60). As a
result of the outward movement of the critical layer with Mach number (fig. 8 (from ref. 61) for near adiabatic conditions), this precursor effect occurs further upstream and becomes more important as Mach number increases. Unless a measured starting profile is available, accurate compressible-boundary-layer calculations, at least for the low to moderate Reynolds number range, should include some consideration of this precursor influence. (See also ref. 32.) Experimental evidence indicates that the magnitude of the precursor transition effect may be influenced by $T_w/T_e$, the effect perhaps becoming less pronounced at lower values of $T_w/T_e$.

Large $p'$ Levels

The usual assumption in second-order closure approaches is that most pressure fluctuation terms, especially the so-called pressure-dilation terms (ref. 52) can be neglected. The reasons for this assumption are threefold: (1) The limited data for low speeds indicate that these terms are indeed small; (2) since $M'$ is usually small, the influence of compressibility (which is primarily represented in the second-order approaches by the $p'$ terms) must be small; (3) there is an absence of data concerning the actual magnitude of these terms. Although no data for the terms themselves exist, there is limited data for $p'$ itself, which will now be examined.

Figure 9 indicates the level of rms wall-pressure fluctuations (normalized by the external mean static pressure) as a function of Mach number (from ref. 47). These data indicate that for moderate hypersonic Mach numbers ($M \approx 8$ to 12) the wall-pressure fluctuation levels approach 10 percent, which is a fluctuation level typical of the longitudinal velocity field. In fact, data from reference 62 at $M_e = 9.4$ indicate that the turbulence is dominated by high-frequency pressure fluctuations.

The distribution of $p'$ with distance away from the wall is shown in figure 10 (from ref. 62). These data indicate that $p'$ levels are large, not only at the wall, but across the sensibly turbulent portion of the boundary layer. The $p'$ intensity diminishes only in the outer (intermittent) portion of the flow. Therefore, the $p'$ levels for hypersonic turbulent boundary layers can evidently be large; this calls for a close, careful examination of second-order closure schemes for the high $M$ case (and also for separated flows, where $p'$ levels are large even at low Mach number). Of especial importance are $p'$ terms such as $p'(\partial u_i/\partial x_i)$. (See also ref. 63.)

Definition of Boundary-Layer Thickness

Difficulty in defining the boundary-layer thickness occurs because of large differences between $\delta_{\text{pitot}}$ and $\delta_{\text{velocity}}$ for high $M$ boundary layers. A typical result for $M \approx 20$ (from ref. 47) is shown in figure 11, where the nominal $\delta_{\text{velocity}}$ is approximately 50 percent of $\delta_{\text{density}}$ (or $\delta_{\text{pitot}}$). The pitot edge is easier to measure but there are two fundamental questions: with a relative absence of mean shear above $\delta_{\text{velocity}}$, what is the turbulence shear stress (and turbulence eddy) content in this region, and what thickness does one use to scale conventional mean field closure models (such as mixing length) for
the high M case? These questions also involve the fact, as previously dis-
cussed, that the density becomes difficult to determine accurately for high M
due to the small difference between two large numbers. These questions are
still open, but limited information (for instance, ref. 64) indicates that
$\delta_{velocity}$ should be used as the scaling function.

Energy Loss Via Acoustic Waves

This subject has received little attention since the original work of
Laufer (ref. 26, pp. 381-393). The physical problem arises because of the
increasing intensity of turbulent boundary-layer sound radiation with increasing
Mach number. For large enough radiated sound levels the amount of energy car-
ried by these waves can be appreciable and represents a new energy dissipation
mechanism in high Mach number compressible flows.

In reference 26 Laufer computed that for $M_e \approx 5$, the energy radiated away
is approximately 1 percent of the work done by turbulent shear and therefore is
negligible. However, if Laufer's equations hold for the case with $M_e = 20,
T_w/T_t \rightarrow 1$, and $\gamma = 5/3$, the same calculation yields radiated energy of 25 per-
cent which is no longer negligible. For the nozzle wall boundary layers or for
boundary layers measured on wind-tunnel models which have turbulent boundary
layers, the flow probably reaches an equilibrium state involving a balance
between the absorption of acoustic energy radiated by the turbulent flows which
surround the local flow and the acoustic energy radiated by the local flow
itself, that is, a balance between gain and loss as far as acoustic energy is
concerned. This balance probably results in a lower net loss and thus may tend
to obscure (in ground facilities) the true importance of this energy radiation
effect. Only measurements in a "quiet tunnel" (ref. 65) or on a flight vehicle
can clarify the true influence of energy loss via acoustic waves.

There are other problems peculiar to the calculation of compressible turbu-
lent boundary layers such as (1) persistence and importance of wall temperature
history effects, (2) increasing predominance of low Reynolds number amplifica-
tion, and (3) variable edge entropy, but these topics are more conveniently dis-
cussed in connection with the mean field closure methods. It should be noted
that the nondimensional burst period for compressible flows is approximately the
same as that for low-speed flows (ref. 66).

TRADITIONAL PREDICTION METHODS

Methods for calculating turbulent, compressible boundary-layer development
have been a topic of extensive research for many years. Until the advent of the
high-speed computer, most of this research effort was concerned with prediction
methods which could be applied by using only hand calculations or very simple
machine computations. Even now, work is continuing on these easily calculable
methods because of their inherent accuracy (due to extensive empirical valida-
tion) and because of the ease and speed with which these methods can be applied
to engineering problems. Traditional methods can be classified into two general
categories: integral methods and correlation methods. Numerous survey papers
are available (e.g., ref. 18 and pp. 181-229 of ref. 28) which attempt to evalu-
ate the accuracy and limits of applicability of various traditional methods for compressible two-dimensional turbulent flows.

The objectives of this section are to describe and discuss the merits of each category of traditional methods. In addition, several illustrative examples of each category will be discussed and compared with experimental data. Comments concerning the accuracy and applicability of these prediction methods will be made. A discussion of compressibility transformations is first presented since transformations are frequently used in both correlation methods and integral methods; next, a discussion of correlation methods including compressible law-of-the-wall/law-of-the-wake formulations is given; finally, integral methods are discussed. Integral methods may use a transformation technique and/or a correlation method in the complete solution procedure.

Transformations

One of the earliest approaches to predicting compressible turbulent flows was to seek a transformation which, when applied to the governing equations for compressible flow, yields identically the incompressible equations. This technique has been used with success for laminar boundary-layer flows which satisfy certain requirements on the relationship between kinematic viscosity and temperature. (Reynolds stress terms do not appear in the laminar boundary-layer equations.) The obvious advantage of this approach for turbulent flows is that, if successful, the more extensive knowledge of the mechanism of turbulent momentum transfer for incompressible flows can be used to predict compressible flows. The transformation concept assumes that the companion incompressible flow resulting from the transformation is physically observable; in addition, it is usually necessary to assume an invariance hypothesis concerning some transformation scale. Beckwith (ref. 18) points out several examples where the physical concepts of a compressibility transformation are violated because of a lack of correspondence of the transformed incompressible flow. These include the following: (1) Normal pressure gradients which may be important in high-speed flows do not exist in incompressible flow and, therefore, are not transformed; (2) dissipation effects in heating calculations are not properly transformed; (3) large normal temperature gradients in the transformed incompressible flow result from the transformation and, according to the equation of state, cannot exist in the constant-density flow; (4) fluctuating density terms in the compressible formulation have no counterpart in the corresponding constant-density flow.

The earliest transformation for turbulent flow was presented by Dorodnitsyn (ref. 67) who considered only the Von Kármán momentum integral equation. Mager (ref. 68) was the first to attempt to transform the partial-differential equations for turbulent boundary-layer flow. Examples of other attempts to define a transformation in the same time frame were Culick and Hill (ref. 69), Burggraf (ref. 70), and various reference temperature or enthalpy methods for zero-pressure-gradient flows (e.g., refs. 71, 72, and 73).

Some years later, Coles (ref. 15), criticizing the assumption made in references 68 and 70 of invariant turbulent shear and stream function under the transformations, proposed a more physically acceptable transformation in which the adiabatic, compressible, and the constant-density flows are assumed to be
related by three scaling parameters \( \sigma(x) \), \( \eta(x) \), and \( \zeta(x) \). The first parameter relates the stream functions of the two flows, the second is a multiplicative factor of the Dorodnitsyn-Howarth scaling of the normal coordinate, and the third relates the streamwise coordinates of the two flows. An additional assumption pertaining to the invariance of a Reynolds number characterizing the law-of-the-wall region of the boundary layer is necessary to complete the transformation. This assumption, which Coles called the substructure hypothesis, provides a substitute for a reference state utilized with many theoretical approaches. Coles' transformation was then modified or extended as follows: (1) Crocco (ref. 74) modified Coles' transformation to include effects of heat transfer and pressure gradient; (2) Baronti and Libby (ref. 75) replaced Coles' substructure hypothesis with a sublayer hypothesis based on experimental observation; (3) Jeromin (ref. 76) extended Coles' transformation to include effects of mass transfer; (4) Lewis, Kubota, and Webb (ref. 77) defined new coordinates consistent with Coles' transformation and eliminated the need for a substructure or sublayer hypothesis (they found, however, that dissipation effects for compressible flows invalidate the transformation for high wall temperatures); (5) Economos and Boccio (ref. 78) empirically modified Coles' law of the wall/law of the wake and introduced two compatibility equations which provide the closure conditions. Coles' approach and the subsequent companion work represent the primary advances in transformation theory in recent years.

Comparisons with experimental data have shown that transformations yield good results only for moderate Mach numbers \( (M \leq 6) \) and for moderate wall heating \( (T_W/T_t \geq 0.5) \) with zero or mild pressure gradients (refs. 75, 77, 79, and 80). (In ref. 79, it was observed that while compressibility had little effect on mixing length for flat-plate-type turbulent boundary layers, Mager's transformation predicts a large effect.) By empirically modifying Coles' transformation, Economos and Boccio (ref. 78) were able to extend the range of agreement with experimental data beyond any other method. While this empirical modification was not physically appealing (transformation of impermeable wall case yields mass transfer in transformed plane), agreement with experiment was substantially improved. This is illustrated in figure 12 where the methods of Baronti and Libby (ref. 75) and Economos and Boccio are compared with experiment; here, local skin friction values were obtained from experimental velocity profiles according to each transformation approach (by fitting law-of-the-wall profiles to experimental profiles) and normalized by a reference value. (See ref. 80 for details.) This reference value is equivalent to the measured skin friction as illustrated in figure 12. As the ratio of wall temperature to total temperature is decreased, a systematic error appears in the skin friction obtained using Coles' transformation with the sublayer hypothesis as recommended by Baronti and Libby. The empirical modification of Economos and Boccio apparently eliminates this disagreement.

The ability of Coles' original transformation to predict flat plate skin friction and heating for Mach 4 to 13 and \( 0.14 \leq T_W/T_t \leq 0.7 \) is illustrated in figure 13. The experimental data are transformed according to Coles' method and compared with a good incompressible prediction (ref. 81). The prediction of skin friction is generally poor; this poor agreement results from an effect of \( T_W/T_t \), not accounted for in the transformation. (See ref. 82.) The improved heating prediction in figure 13 reflects the use of a Reynolds analogy factor \( (2N_{St}/C_f) \) to adjust the level of the data.
Both references 77 and 78 applied their versions of Coles' transformation with an incompressible integral method, and examples of the results are shown in figure 14. Here, the skin friction and momentum thickness for the waisted body of revolution from reference 83 are compared with predicted values from each method. These comparisons properly belong in a later section on integral methods but are shown here to illustrate the accuracy of these transformation methods. The agreement of the method of reference 78 (fig. 14(a)) with experiment is good and rivals some of the more advanced numerical methods, while the predictions of reference 77 (fig. 14(b)) are obviously poor and unacceptable for engineering purposes. (However, curvature and laminarization effects may be present in some of the data of ref. 83 and thus could influence these comparisons.)

These comparisons of the results from transformation methods with experimental data and other comparisons in the cited references lead to the general conclusion that while some success for adiabatic flows is evident, transformation methods are not presently desirable for general applications, a conclusion also reached in references 18 and 28 (pp. 181–229).

Correlation Methods

Correlation methods are perhaps the most popular of the traditional methods because they are generally simple to apply and some are quite accurate. Even though these methods are strictly applicable only for two-dimensional or axisymmetric flows with zero pressure and wall temperature gradients, they are widely used for parametric design studies and preliminary estimates. The empirical input to some of these methods enhances the accuracy and allows easy modification of the method to include new empiricism. Some of these methods provide predictions for wall shear stress only, while others also allow specification of the boundary-layer profiles. Additional assumptions for the Reynolds analogy factor $2\tau_{st}/C_f$ and the recovery factor are required in order to calculate wall heat transfer for those methods which only predict wall shear.

Spalding and Chi (ref. 81) have presented an excellent summary of available correlation methods for compressible turbulent flow up to 1964. Since 1964 the only correlation method to arise is that of White and Christoph (ref. 84). According to Spalding and Chi, correlation methods may be classified as follows:

(1) Methods using Prandtl or Von Kármán differential equations (i.e., Prandtl or Von Kármán mixing length concepts)

(2) Theories based upon other differential equations

(3) Theories based upon a fixed velocity profile

(4) Theories based upon incompressible formulae with reference properties

In categories 1 and 2 the shear stress is assumed to be constant through the boundary layer and equal to its wall value; in category 3 the velocity profile is assumed independent of compressibility; in category 4 the incompressible relations are assumed to apply for compressible flows if the gas properties are evaluated at a reference temperature or enthalpy where the reference temperature
is a function of Mach number, ratio of wall to edge temperature, and recovery factor. Some examples of the more popular methods in each category are as follows:

Category 1 - Van Driest I (ref. 13) and Kutateladze and Leont'ev (ref. 85), which use a Prandtl mixing length, and Wilson (ref. 86), Van Driest II (ref. 87), Harkness (ref. 88), Deissler and Loeffler (ref. 39), and Moore (ref. 89), which use a Von Karman mixing length.

Category 2 - Li and Nagamatsu (ref. 90) and Kosterin and Kashmarov (ref. 91).

Category 3 - Cope (ref. 92) and Monaghan (ref. 93).

Category 4 - Summer and Short (ref. 73) and Eckert (ref. 72).

In addition to these methods, Spalding and Chi (ref. 81) present a method which uses a Van Driest type analysis to determine a function $F_c$ and empirical data to determine a function $F_R$. In their method the assumption is made that $F_cF_f$ varies uniquely with $F_R$, according to an incompressible law ($R$ is a Reynolds number) and that $F_c$ and $F_R$ are functions only of Mach number, ratio of wall temperature to edge temperature, and recovery factor.

Numerous survey papers (refs. 16, 80 to 82, 84, and 94 to 98) have attempted to determine which of the correlation techniques best predicts available experimental data for compressible turbulent boundary-layer flow. The method which is found to be the "best" in each of these survey papers is usually either Van Driest II (ref. 87), Spalding and Chi (ref. 81), or a reference temperature approach (e.g., Eckert, ref. 72). The choice of a "best" method is influenced in each of the survey papers by particular selections of virtual origin, Reynolds analogy factor for heating calculations, and recovery factor. (See ref. 82.)

Comparisons of experimental skin friction and heat transfer data with predicted values obtained from the methods of Van Driest (ref. 87), Spalding and Chi (ref. 81), and Eckert (ref. 72) for Mach numbers 4 to 10 and ratios of wall temperature to total temperature from 0.14 to 0.7 are shown in figures 15 to 17. The data were obtained in wind tunnels on flat-plate models with $d
\frac{p}{d}x = 0$ and $dT_w/dx = 0$; skin friction was measured with balances and wall heating was measured by transient techniques. The comparisons are presented in the form $F_cF_f$ against $F_{Rg}R_g$ or $F_{Rx}R_{x}v_0$, where $F_cF_f$ is equivalent to the incompressible value of skin friction and $F_{Rg}R_g$ or $F_{Rx}R_{x}v_0$ is equivalent to the corresponding incompressible value of Reynolds number. (See ref. 82 for further explanation.) In this manner, all the transformed experimental data can be compared directly with values from a good incompressible skin friction law (law from ref. 81 used herein) to judge the efficacy of each method.

The comparison of the data with predicted values from the Van Driest II method (ref. 87) is shown in figure 15. For this comparison, a momentum thickness Reynolds number is used, and a Reynolds analogy factor of 1.0 is assumed, as recommended for best results with the Van Driest II method by Hopkins and Inouye in reference 98. The comparison is favorable for heat transfer, but the
transformed skin friction data fall generally below the prediction. As reported in reference 82, the prediction error also appears to be a function of $T_w/T_t$. A similar comparison for the method of Spalding and Chi (ref. 81) is shown in figure 16. Here, the virtual origin location was assumed to be near the end of boundary-layer transition, and Von Kármán's Reynolds analogy factor (used as suggested in the appendix of ref. 95) was applied as recommended for the Spalding and Chi method in reference 82. The prediction of both the heating and skin friction data in figure 16 is quite good. The results from Eckert's method (ref. 71) are shown in figure 17 using the same virtual origin and Reynolds analogy factor as previously used with the Spalding and Chi method. The comparison between data and prediction is again very good in figure 17, but there appears to be a discernible variation of prediction errors with $T_w/T_t$. (See ref. 82.) Considering the wide range of flow conditions of the data, each of the three approaches provides a credible prediction method for zero-pressure-gradient turbulent-boundary-layer flows.

Comparisons of predictions from Eckert's method (ref. 72) with data from flight are shown in figures 18 and 19. Data obtained on conventional aircraft such as the A-5A, the Mirage IV, and the XB-70-1 (refs. 99 to 101) are shown in figure 18. The agreement with Eckert's method is good in each case. Heating data from sharp and blunt cones in flight are shown in figure 19 (from ref. 96) where Colburn's Reynolds analogy (ref. 102) was used for the calculation and the virtual origin was assumed to occur at the stagnation point or cone tip. A cone/flat plate transformation after Van Driest (ref. 103) was applied. The agreement between prediction and data is surprisingly good over the wide range of data included (3 $\leq M \leq$ 13, 0.2 $\leq T_w/T_e \leq$ 2.3). It appears therefore that these correlation methods provide accurate predictions of turbulent skin friction and heat transfer for a wide range of flow conditions (for $dp/dx \approx 0$, $dT_w/dx \approx 0$) in flight as well as in wind tunnels.

Compressible law of the wall.- The law-of-the-wall velocity profile has been useful in arriving at quantitative analytical results for incompressible flows. This incompressible law is empirical in nature but can be derived directly from mixing length concepts; that is,

$$\tau = \bar{p} l^2 \left( \frac{\partial u}{\partial y} \right)^2$$

and

$$l = Ky$$

(Prandtl)

so

$$\tau = \bar{p} K^2 y^2 \left( \frac{\partial u}{\partial y} \right)^2$$

Integration yields ($\rho =$ Constant and $\tau = \tau_w =$ Constant)

$$\frac{\bar{u}}{u_T} = u^+ = \frac{1}{K} \ln y^+ + C$$

(21)
which is the incompressible law of the wall and is valid only very near the wall but not in the viscous sublayer. Coles (ref. 104) extended equation (21) to include the wake portion of the boundary layer, and the resulting equation is

\[ u^+ = \frac{1}{K} \ln y^+ + c + \frac{\pi}{K} w \]  

(22)

where \((\pi/K)w\) is a wake function which accounts for the wake-like behavior of the outer portion of the turbulent boundary layer. Alternately, by evaluating equation (22) at the edge of the boundary layer and subtracting the resulting equation from equation (22), the velocity defect equation is obtained

\[ \frac{u_e - u}{u_T} = -\frac{1}{K} \ln \frac{\delta}{K} + \frac{\pi}{K} (2 - w) \]  

(23)

An expression for the incompressible profile from the wall to the outer edge of the buffer layer (beginning of fully turbulent region) was presented by Spalding (ref. 105) and Kleinstein (ref. 106). Modifications to the law of the wall for surface roughness (e.g., ref. 107) and wall injection (e.g., ref. 108) are available. Equation (23), the velocity defect distribution, is independent of surface roughness. The law-of-the-wall/law-of-the-wake expressions have been used extensively and with great success in studies of turbulent boundary layers either to obtain such parameters as \(C_f\) and \(\delta\) from experimental velocity profiles or as an auxiliary equation in integral or numerical prediction methods.

The successful application of the law-of-the-wall/law-of-the-wake velocity profile for incompressible flows has naturally led to numerous attempts to apply similar concepts to compressible flows. A brief description of some of the more prominent of these studies follows.

Van Driest (ref. 13) in 1951 derived a compressible law of the wall for zero-pressure-gradient flows by solving the turbulent-boundary-layer equations with Prandtl's mixing length formulation and a laminar and turbulent Prandtl number of unity. By comparing the resulting law of the wall from Van Driest's analysis with the incompressible law of the wall, it is clear that the effects of compressibility can be accounted for by defining a generalized velocity \(u^*\) in place of \(u\). (See Maise and McDonald (ref. 79).) This generalized velocity is defined as

\[ u^* = u_\infty \frac{1}{A} \sin^{-1} \left[ \frac{2A^2 (\bar{u}/u_\infty) - B}{(B^2 + A^2)^{1/2}} \right] = \int \left( \frac{\rho}{\rho_w} \right)^{1/2} du \]

where \(\rho\) is from Crocco's relation and

\[ A^2 = \frac{(Y - 1)/2 M_\infty^2}{T_w/T_\infty} \]
More recently Maise and McDonald (ref. 79) applied these concepts by assuming that Coles' law of the wall was valid for compressible flow if the velocities are defined as the generalized velocity. The resulting equation for the velocity defect was

\[
\frac{u_*^* - u_*^*}{u_T^*} = \frac{1}{K} \ln \frac{y}{\delta} + C(2 - w)
\]

where \( w \) is Coles' tabulated wake function (ref. 104). Using Clauser's results (ref. 1) for incompressible flow to define the constants, the final expression is

\[
\frac{u_*^* - u_*^*}{u_T^*} = -2.5 \ln \frac{y}{\delta} + 1.25(2 - w)
\]  

(24)

Maise and McDonald found that experimental velocity profiles for adiabatic, compressible turbulent boundary layers from Mach 1.5 to 5 through a wide range of Reynolds numbers were well correlated by equation (24). However, the velocity profiles for nonadiabatic flows in the same range of Mach number were poorly correlated.

Fernholz (ref. 109) used a similar analysis with generalized velocities but did not use Clauser's constants. Instead, experimental data were used to correlate the constant as a function of the ratio of wall temperature to adiabatic wall temperature and Reynolds number. The formulation was

\[
\frac{u_*^*}{u_T^*} = \frac{1}{K} \ln \frac{y u_T^*}{v_w} + F_1
\]  

(25)

where

\[
F_1 = f\left(\frac{T_{aw} - T_w}{T_{aw}}, \frac{\rho_e u_e \theta}{\mu_w}\right)
\]

Fernholz also included the velocity defect law but redefined the \( \delta \)-coordinate as was done for incompressible flows. The resulting equation was

\[
\frac{u_*^* - u_*^*}{u_T^*} = -\dot{M} \ln \frac{y}{\Delta^*} - \dot{N}
\]  

(26)

where

\[
\Delta^* = \dot{\delta} \int_0^1 \frac{u_e^* - u_*^*}{u_T^*} \, d \frac{y}{\delta}
\]

\( \dot{M} = 4.7 \) and \( \dot{N} = 6.8 \) for adiabatic walls. Equation (25) provided a good correlation of experimental velocity profiles for Mach 5 to 8 with moderate heat transfer. Equation (26) also correlated the defect region of these profiles for \( R_\theta = \rho_e u_e \theta / \mu_w \) greater than 1000 to 1500.
Mathews, Childs, and Paynter (ref. 110) using generalized velocity concepts fitted the velocity defect equation (eq. (24)) to experimental velocity profiles (by the method of least squares) and thereby obtained the skin friction and boundary-layer thickness which allowed the best profile fit. The test cases involved both normal and conical shock interactions with turbulent adiabatic boundary layers at supersonic speeds as well as a flat-plate supersonic flow. The velocity profile fits were good, and the resulting skin friction and boundary-layer thickness values compared favorably with experiment and other predictions.

Sun and Childs (ref. 111) modified the analysis of reference 110 by using a more realistic shear stress distribution through the boundary layer. Instead of assuming \( \tau = \tau_W = \text{Constant} \), the expression

\[
\frac{\tau}{\tau_W} = 1 - \left(\frac{y}{\delta}\right)^b
\]

was used. Equation (27) with \( b = 1 \) is a reasonable fit to experimental data for both subsonic and supersonic flows (ref. 25). This modification satisfies the physical condition that the velocity gradient be zero at \( y = \delta \) and provides more accurate values of boundary-layer thickness. The profile fit and the resulting skin friction were little changed from the analysis of reference 109.

Kane (refs. 112 and 113) used the compressible counterpart of equation (1) which is

\[
u^+ = \frac{1}{K} \ln y^+ + C
\]

where

\[
u^+ = \int_0^y \left( \frac{\rho}{\rho_W} \right)^{1/2} \frac{\tau}{\tau_W} \, du^+
\]

Equation (29) was evaluated by using the Crocco temperature profile and Squire's definition of \( \tau/\tau_W \) (ref. 114) which includes the effects of wall injection. Inclusion of an empirical extension of Kleinstein's buffer layer profile (ref. 106) to compressible flow as well as Coles' wake parameter completed the formulation of the compressible law of the wall/law of the wake. Kane finds that three profile functions remain undefined in his compressible law of the wall/law of the wake which must be obtained from a fit of experimental velocity profile data. A multiple regression analysis is then used to define the best functional variation of these three functions with appropriate fluid variables.

Squire (ref. 114) used an approach similar to Kane's method to extend the compressible law of the wall to flows with wall injection. By fitting experimental velocity profile data, the integration constant in the law of the wall was determined as a function of \( v_w/u_T \), and Mach number. The slope of the log region \( K \) was found to be independent of mass injection. Chen (ref. 115) extended the generalized velocity concepts used by Maise and McDonald (ref. 79) to flows with rough wall, heat transfer, and pressure gradient. White and
Christoph (ref. 116) included effects of longitudinal pressure gradient on the shear stress distribution and then, using a method similar to Squire (ref. 114) with \( v_w = 0 \), obtained a closed form solution for the law-of-the-wall velocity profile.

Danberg (ref. 115) used Coles' law-of-the-wall/law-of-the-wake profile and a reduced velocity

\[
u' = \int \left( \frac{\rho}{\rho_w} \right)^{1/2} \, du
\]

to account for compressibility effects. Four profile parameters were determined from least square fits of Danberg's compressible law of the wall to experimental velocity profile data from Mach 2 to 6 for adiabatic wall conditions. Danberg attempted a similar definition of a law of the wall for the temperature profile, but a paucity of experimental data prevented definitive results.

This discussion of the status and development of a compressible law of the wall indicates that reliable expressions for a compressible law of the wall exist for adiabatic wall conditions and Mach numbers less than 6. However, for flows with significant heat transfer, the definition of a compressible law of the wall is still unclear. The unresolved question of the effect of heat transfer on the applicability of a compressible law of the wall is clearly illustrated when the results of Maise and McDonald (ref. 79), which show poor correlation of compressible velocity profiles with heat transfer, are compared with the results of Gran, Lewis, and Kubota (ref. 118) which show good correlation of compressible velocity profiles with pressure gradient and heat transfer, both studies using nearly the same compressible law-of-the-wall formulation; the difference between the two formulations was that \( A^* \) (see eq. (26)) was used instead of \( \delta \) in the velocity defect law in reference 117. It thus appears that further progress in defining a general law-of-the-wall velocity profile for compressible turbulent boundary layers will be paced by progress in obtaining detailed and accurate experimental data through a wide range of flow variables.

Integral Methods

All integral methods solve the (Von Kármán) integral momentum equation along with various auxiliary relations. The two-dimensional Von Kármán momentum integral equation is obtained by integrating the x-momentum equation. When normal stress terms are neglected, the following equation is obtained:

\[
\frac{d\theta}{dx} + \frac{\theta}{u_e} \frac{d u_e}{dx} \left( H + 2 + \frac{u_e}{\rho_e} \frac{d p_e}{d u_e} \right) + \frac{1}{\rho_e u_e^2} \frac{d}{dx} \left( p_e \delta - \int_0^{\delta} p \, dy \right) = \frac{C_f}{2}
\]  

(30)

where

\[
\theta = \int_0^{\delta} \frac{\rho u}{\rho_e u_e} \left( 1 - \frac{u}{u_e} \right) \, dy
\]
If \( p \) is independent of \( y \) and the free-stream flow is adiabatic, then

\[
\frac{\partial u_e}{\partial y} \frac{\rho_e}{\partial u_e} = -M_e^2
\]

and equation (30) becomes

\[
\frac{d\theta}{dx} + \frac{\theta}{u_e} \frac{dH}{dx} + (H + 2 - M_e^2) = \frac{C_f}{2}
\]

There are two basic types of integral methods - those which solve the equations in the physical plane and those which use a compressibility transformation and then solve the equations in the incompressible plane. Most of the earlier methods used a compressibility transformation, while more recent methods favor solving the equations in the physical plane. Examples of auxiliary relations used in the numerous methods available in the literature are as follows:

1. Moments of the integral momentum equation, which are obtained by multiplying the momentum equation by \( y \) or \( u \) and integrating across the boundary layer (these moment equations contain unknowns involving turbulent shear stress or dissipation integrals which must be defined in terms of known quantities)

2. Entrainment equation and/or lag equation

3. Specification of velocity, temperature, and/or shear stress profiles

4. Specification of form factor, wall shear, and/or wall heating (Reynolds analogy)

By using selected auxiliary relations, the problem is reduced to solving a set of quasilinear, coupled ordinary differential equations by an available solution technique.

Integral methods have advantages which have assured their continued use through the years. Primary advantages are the following: (1) Solution procedures are fast and easily programed; (2) starting procedure is simple; (3) less detailed information on turbulence is necessary; (4) integration process eases restriction on accuracy of profile shapes. The disadvantages of such methods are that considerable empiricism is necessary to close the equation set (i.e., relying upon empirical input restricts accuracy and range of application) and that nonequilibrium effects are difficult to include. The latter restriction becomes particularly important for high Mach number flows. Of course, if a compressibility transformation is used, the integral method is subject to all the uncertainties inherent in the transformation.
Most earlier integral methods utilized a compressibility transformation in the solution procedure. Examples of these are the methods Reshotko and Tucker (ref. 119), Englert (ref. 120), Mager (ref. 68), Culick and Hill (ref. 69), and Spence (ref. 121). More contemporary methods using transformations are those of Sasman and Cresci (ref. 122), Camarata and McDonald (ref. 123), Flaherty (ref. 124), Zwarts (ref. 125), and Küster (ref. 30, pp. 19-1 - 19-11); extensions of Head's entrainment method (ref. 126) to compressible flows by transformations were given by Standen (ref. 127), So (ref. 128), and Green (ref. 129).

In recent years, integral methods which do not use a compressibility transformation but rather use auxiliary relations which are valid in the compressible plane have found considerable favor. A representative list of such methods includes those of Miller (ref. 130), Michel, Quémard, and Cousteix (ref. 131), Johnson and Boney (ref. 132), Pinckney (ref. 133), Reeves (ref. 30, pp. 6-1 - 6-A2-2), and White and Christoph (ref. 84). In addition, Green (refs. 129 and 134) and Green, Weeks, and Brooman (ref. 135) extended Head's entrainment method (ref. 126) to compressible flows also using physical variables. Obviously, discussion of all these methods is not feasible in the present paper; therefore, a representative example of each type of method is discussed. The particular methods chosen were a method using a transformation (Flaherty, ref. 124), a method using entrainment concepts in physical variables (Green, Weeks, and Brooman, ref. 135), a method specially configured for adverse gradients (generated by concave surfaces) also in physical variables (Pinckney, ref. 133), and a method in physical variables but using a compressibility transformation for a portion of the solution (Reeves, ref. 30, pp. 6-1 - 6-A2-2). The structural highlights of each of these four calculation procedures will be discussed along with comparisons with typical experimental data.

Method of Flaherty.- This method (ref. 124) is basically a modification of the procedure of Reshotko and Tucker (ref. 119) and solves the momentum integral and moment-of-momentum equations in the transformed plane defined by using Stewartson's transformation (ref. 136). The skin friction coefficient is obtained from the Ludwieg and Tillman incompressible equation (ref. 137) using reference temperature concepts (ref. 72). An empirical expression is used by Flaherty for the shear stress integral through the boundary layer and is the main improvement over the earlier Reshotko and Tucker method which used a constant shear stress. A provision for calculating wall heat transfer based upon the energy deficit in the boundary layer is also included.

Comparisons of predictions from Flaherty's method with two sets of experimental data are shown in figure 20. These example data were chosen to illustrate the ability of the method to predict boundary-layer growth along flat or curved surfaces with favorable pressure gradient. Good prediction of the boundary-layer thickness data from reference 138 at Mach 1.5 (fig. 20(a)) is achieved over the entire length of the test plate. Reasonable agreement with the data from reference 83 at Mach 2 (fig. 20(b)) on the waisted body of revolution is also obtained, but the agreement of prediction with both momentum thickness and skin friction data deteriorates in the region of adverse pressure gradient (x/L > 0.7). Comparisons of this method with other available data in references 30 (pp. 181-229) and 124 indicate generally good predictions of boundary-layer integral properties up to Mach 6 with moderately cooled walls and moderate pressure gradient.
Method of Green, Weeks, and Brooman.- This integral method (ref. 135) is essentially an extension of Head's method (ref. 126) to compressible flow and attempts to account for the influence of upstream flow history on the turbulent stresses. The procedure consists of solving in the physical plane the integral momentum equation, the entrainment equation, and a rate equation for the entrainment coefficient. This rate equation or lag equation is developed from an equation for shear stress derived from the turbulent kinetic energy equation. In addition, an empirical factor is introduced to account for the variation of integral parameters with Mach number in equilibrium flows. The method as formulated is restricted to adiabatic flows.

Two comparisons of predictions from this method with experimental data are shown in figure 21. The first is a comparison with the data of reference 83 (fig. 21(a)). The prediction is in good agreement with the experimental surface shear and momentum thickness, even at the rear (adverse pressure gradient) of the waisted body. Good agreement is also obtained with the data of Lewis, Gran, and Kubota (ref. 139), as shown in figure 21(b). Here the pressure gradient is zero up to \( x \approx 35.6 \text{ cm} \), adverse from \( x \approx 35.6 \text{ cm} \) to \( 45.7 \text{ cm} \), and then favorable thereafter. The good agreement seen in figure 21 is also evident in other comparisons shown in reference 135 for adiabatic conditions up to Mach 4.

Method of Pinckney.- This method (ref. 133) was especially tailored to predict turbulent boundary-layer development on compression (concave) surfaces common to hypersonic air breathing engines. The set of equations which is solved consists of the momentum integral equation, the moment-of-momentum equation, and the integral energy equation. Auxiliary relations consist of (1) assuming a Crocco-like temperature profile which satisfies the total energy deficit across the boundary layer as determined from the net heating along the surface (history effect), (2) assuming an empirical shear stress distribution across the boundary layer, and (3) assuming the Spalding and Chi (ref. 81) skin friction prediction (suitably shifted) is valid in pressure gradient flows (as well as the resulting heat transfer coefficient obtained from Spalding and Chi skin friction with Von Kármán's Reynolds analogy factor (ref. 140)). Provisions are made to allow calculations to proceed through transition and into fully turbulent boundary-layer flow.

Comparisons of Pinckney's method with experimental data from reference 141 are shown in figure 22. These integral boundary-layer-thickness data were obtained on an axisymmetric compression surface for both Mach 5 adiabatic flow and Mach 8 cold-wall conditions. Predictions from the integral method are generally in good agreement with the integral thicknesses for both experimental conditions shown. Comparisons with other data in reference 133 are also favorable and confirm reasonable accuracy for this prediction method at least up to Mach 8 with moderate wall cooling.

Method of Reeves.- This method (ref. 30, pp. 6-1 - 6-A2-2) solves the compressible turbulent boundary-layer equations including mass transfer by assuming a two-layer boundary-layer model. Mixing length concepts are assumed to apply in the inner layer, and this results in a compressible law-of-the-wall expression; this expression is inserted into the boundary-layer conservation equations which are integrated away from the wall to a matching location. The outer (or wake) layer solution uses an integral momentum method, the results from which
are matched to the inner layer solution at a prescribed match point. The outer layer integral solution employs a compressibility transformation. An empirical expression for the shear stress integral for the outer layer is specified. Apparently the dynamics of the boundary-layer solutions from this method depend on coupling and interaction between the inner and outer solutions.

Two comparisons of predictions from Reeves' method with experimental data are shown in figure 23. The predictions of the integral thicknesses for the data of reference 138 (fig. 23(a)) at Mach 1.5 are excellent. While the prediction of momentum thickness for the data of reference 83 (fig. 23(b)) at Mach 2.4 is good, the skin friction data are not well predicted. Reeves found good agreement between his predictions and experimental data up to Mach 10 for a wide range of wall temperature ratios with and without wall mass transfer. Some computational stability problems occurred for negative pressure gradients. The method was computationally quite rapid.

In summary, this brief review of various available integral solution procedures for two-dimensional compressible turbulent boundary-layer flow indicates that reasonably accurate predictions are possible using integral techniques for a wide range of flow conditions. Caution must be exercised in applying integral methods to flows with severe pressure gradients as well as other nonequilibrium effects and to flows where the empirical correlations used in the method do not apply. While integral methods provide fast and inexpensive calculations necessary for design and analysis of fluid systems, the limitations of any method must be clearly understood to prevent erroneous conclusions. As discussed earlier, transformation theory has not, thus far, developed into a tool for general application, and therefore, integral methods using a "complete" compressibility transformation should probably be avoided at present.

NUMERICAL SOLUTION PROCEDURES

This section considers the various alternative numerical procedures currently being used to solve the nonlinear partial-differential equations describing compressible turbulent boundary layers (which are parabolic in the marching or longitudinal direction). There are several numerical difficulties peculiar to the calculation of turbulent (as opposed to laminar) compressible boundary layers. These problem areas include (1) presence of a thin sublayer, which requires either a separate wall treatment or variable grid (or coordinate transformation), (2) rapid growth of the boundary layer with longitudinal distance, which requires a transformation or streamline mapping procedure, and (3) algebraic terms in the turbulence modeling expression which can alter the stability of the numerical calculation procedure.

Solution Techniques

The numerical solution procedures used to solve the compressible turbulent boundary-layer problem can be conveniently categorized in the following manner:
The conventional finite-difference methods have been the most popular. They usually employ Crank-Nicolson differencing and the Thomas algorithm for inversion of the coefficient matrix. The advantages of these procedures include ease of coding, numerical stability, and overall simplicity. Their disadvantages are mainly due to their essentially "brute force" approach, in that 100 nodes in the vertical direction are usually needed to adequately represent a turbulent profile (even with nonuniform mesh spacing). Therefore, the procedures generally require relatively large storage and long machine time, especially when chemical reaction effects are included.

The shooting procedure uses finite differences in the longitudinal or x-direction but solves an ordinary differential equation (two-point boundary-value problem) in the transverse or y-direction. Several users of this approach have encountered serious stability problems, especially with wall heating or cooling and pressure gradient cases. The solutions seem to be sensitive to guesses of the inner boundary conditions. Only a few codes presently use the shooting method, and it is probably not one of the best procedures available.

The box method is quite efficient for boundary layers and is of more recent vintage than the conventional finite-difference and shooting procedures. By a change of dependent variables the equations are reduced to a first-order nonlinear system, which is solved by Newton iteration and two-point differencing. This procedure has several advantages, which include high order spatial accuracy, even with a rapidly varying nonuniform grid, and the small number of nodes required for solution (obtained by Richardson extrapolation). As a result of these advantages, this procedure produces solutions in 7 times less machine time and with much less storage than conventional finite-difference methods.

In the micro-integral method, developed at Imperial College of Science and Technology, the boundary-layer growth is accounted for by use of a stream function as a transverse variable, thus reducing the number of nodes required. Most versions of this procedure also use a Couette flow analysis near the wall, which significantly reduces the required calculation time, as the use of small nodal spacing near the wall to resolve the sublayer is no longer necessary. Also, in
this method the convective terms are integrated between grid nodes before differentiation. This solution procedure is one of the more efficient ones available, but the Couette flow analysis near the wall must change with different boundary conditions, and, since the usual formula for grid control (stream-function boundary condition) is explicit, there may be problems for large longitudinal increments.

The Wortman approach is fairly recent and bears some faint resemblance to the shooting method in that finite differences are taken in the longitudinal direction only. An integrating factor is used to reduce the order of the equations and the procedure is referred to as an iterative operator method. The method appears to be quite fast and has a high order of spatial accuracy.

BLIMP (boundary-layer integral matrix procedure) has undergone extensive development, especially for application to flow chemistry problems. The procedure is fairly involved, and the method was specifically developed to minimize the number of grid nodes (especially important for the large equation set which results when chemical effects are included). The method is characterized by finite differences in the longitudinal direction, and strip integrals in the transverse direction with cubic spline fits and Newton-Raphson iteration.

The method of weighted residuals (or method of integral relations) is somewhat similar to strip integral procedures. The method uses weighting and approximating functions which must be assumed. The advantage to the procedure is again the small number of nodes involved, but the coding is fairly complex and considerable insight is often needed to select reasonable functional forms. For turbulent flows the computation time can be of the same order as for the conventional finite difference and the method (and the several variants thereof; including finite element which is a global as well as a local method of weighted residuals) has not been very popular.

In the method of characteristics a solution is obtained in the outer region of the boundary layer only and matched to a law of the wall. For this procedure the viscous terms in the boundary-layer equations must be neglected.

Systems of Independent Variables

The simplest set of independent variables to use would be the actual nondimensional physical quantities x,y (ref. 167). An advantage of this variable system, when compared with the more usual transformations, is the decreased labor involved when changes are made in the physical specification of the problem (i.e., one does not have to keep untransforming and retransforming for quantities or boundary conditions which are specified functions of physical variables). Therefore, these variables are particularly useful for inclusion of alternate turbulence models and to determine the influence of a specified variation of pressure in the transverse direction. However, this system of variables does not account for boundary-layer growth with longitudinal distance without periodic nodal point redistribution and is therefore seldom used in production codes which are expected to apply over changes in Reynolds number of several orders of magnitude.
One method of solving this boundary-layer growth problem is to use a normalizing factor on \( y \) which is a function of \( x \), such as \( \delta, \delta^* \), etc. This will keep the boundary layer within the computational grid, but longitudinal derivatives of the normalizing factors are required; if \( \delta^* \) is used, this quantity can become negative for cold walls.

More generally, solutions are obtained using some variant of the Levy-Lees transformation (e.g., ref. 11)

\[
\xi = \int \rho_{e} u_{e} \, dx
\]

\[
\eta = \frac{1}{\xi^{\eta}} \int \rho \, dy
\]

originally developed for laminar boundary layers. These variables reduce the boundary-layer growth and the influence of variable density in the computational domain. As a rule of thumb, \( \eta \approx 0.5 \) for laminar flows and \( \eta \approx 0.8 \) for turbulent flows. However, in general \( \eta = f(x) \) and the boundary layer may still grow out of (or into) the mesh (i.e., \( \eta_{e} \neq \text{Constant} \) for turbulent flows without solving for \( \eta(x) \)).

Another alternate set of independent variables involves the use of a stream function \( \Psi \) as the normal or transverse coordinates (ref. 9). This approach automatically tracks the boundary-layer growth, but the boundary conditions must be specified or determined. One could also use the Crocco or velocity variables (ref. 168) where the normal coordinate is a function of \( u/u_{e} \). This allows an accurate solution with a nearly constant \( \Delta u/u_{e} \) step size, which corresponds to a highly nonuniform \( \Delta y \); that is, the number of necessary nodes is very effectively minimized. However, this variable set is not suitable for cases where \( u/u_{e} > 1 \) (as transformation involves \( \sqrt{1 - (u/u_{e})} \)), and therefore, wall jet flows would be difficult to compute with the code.

Two examples are given here of the possible problems and inaccuracies which can be encountered in numerical solutions. Figure 24 (taken from ref. 169) indicates the error in \( C_{f} \) associated with a change in nodal point spacing. The usual assumption made in numerical analysis is that the solution becomes more accurate as the integration interval is reduced. However, as the figure shows, for single-precision IBM machines with approximately 7 decimal place accuracy, decreasing the step size can actually increase the discrepancy between a theoretical and numerical solution (as a result of roundoff error). When double-precision arithmetic is used (approximately 15 decimal place accuracy), the expected trend is obtained. The other example is given in figure 25 (from ref. 170). Here \( K = \Delta n_{n+1}/\Delta n_{n} \) and is the conventional means of including variable nodal spacing in a finite-difference procedure (e.g., ref. 53). These results from reference 170 show that, for \( K \neq 1.0 \), there is some error involved in using variable grid spacing (at least as used in ref. 170) and that this error can become appreciable for \( K \approx 1.05 \) (usual values of \( K \) used in solution procedures are 1.02 to 1.04). These examples are only given to indicate that certain simple accuracy checks should be made on any numerical code before the numbers can be fully believed.
MEAN FIELD CLOSURE

This is probably the most widely used recent closure approach for computing compressible turbulent boundary layers. In this procedure the Reynolds stress \((\overline{p u' v'})\) is related directly to the mean velocity and density fields. This assumption is nearly exact for equilibrium and near equilibrium boundary layers where turbulence production is approximately equal to turbulence dissipation (e.g., pp. 275-299 of ref. 45). That is,

\[
\frac{\overline{u' v'} \partial \overline{u}}{\partial y} \propto \frac{e^{3/2}}{l}
\]  

(33)

Using the Prandtl model (ref. 45, pp. 275-299)

\[
\overline{u' v'} \propto \sqrt{e} \frac{1}{l} \frac{\partial \overline{u}}{\partial y}
\]  

(34)

Solving equation (34) for \(e\) and inserting this into equation (33), one obtains

\[
\frac{\overline{u' v'} \partial \overline{u}}{\partial y} \propto \frac{(u' v')^3}{l^4(\partial \overline{u}/\partial y)^3}
\]  

(35)

or

\[
\overline{u' v'} \propto l^2 \left(\frac{\partial \overline{u}}{\partial y}\right)^2
\]  

(36)

which is the usual mixing-length model; that is, \(\overline{u' v'}\) is a function of mean velocity profile. The popularity of the mean field closure procedures is due to several reasons: (1) Many compressible turbulent-boundary-layer flows are either equilibrium or near equilibrium; (2) a wide range of boundary conditions can be easily and accurately incorporated (\(p(x), v_w(x), \) variable edge entropy, transition, roughness, etc.); (3) the higher order mean turbulence field models usually use a mean field model in the near-wall region (e.g., refs. 171 and 172); (4) less computer time and storage are required, compared with mean turbulence field methods; (5) except for highly nonequilibrium flows, mean field methods yield almost the same answers as mean turbulence field methods. (For example, see refs. 8, 24, and 173.)

Details of the mean field closure can be conveniently discussed using sketch (a) where the boundary layer is shown subdivided into the usual three regions. The sublayer is the region nearest the wall. The no-slip and usual impervious wall boundary conditions ("wall discipline") require a wall damping expression as a modifier to whatever turbulence model is used in the other two regions. In the law-of-the-wall or fully turbulent region of the boundary layer, the turbulent motions are scaled as a function of \(y\) and experience indicates that the mixing length model is almost universally valid. The outer or wake region can be strongly influenced by "history" or relaxation effects and here the turbulent motions are scaled as a function of \(\delta\).
Starting in the law-of-the-wall region, the most commonly used mean field closure model is the mixing length

\[ -\bar{u}'\bar{v}' = \frac{1}{2} \left[ \frac{\partial u}{\partial y} \frac{\partial u}{\partial y} \right] \]  \hfill (37)

For \( y \ll \delta \) (law-of-the-wall region)

\[ -\bar{u}'\bar{v}' = \kappa^2 y^2 \left[ \frac{\partial u}{\partial y} \frac{\partial u}{\partial y} \right] \]  \hfill (38)

where \( \kappa \) is the Prandtl or Von Kármán constant, which is approximately equal to 0.4.

In the outer, or wake region, there are two expressions commonly used, a mixing length expression

\[ -\bar{u}'\bar{v}' = \left( \frac{\delta(1 - e^{-y/A})}{\delta_{\text{max}}} \right)^2 \left[ \frac{\partial u}{\partial y} \frac{\partial u}{\partial y} \right] \]  \hfill (39)

and an eddy viscosity expression (ref. 1)

\[ -\bar{u}'\bar{v}' = u_{e\delta_i} \delta_i^* \left[ \frac{\partial u}{\partial y} \right] \]  \hfill (40)

where, from computational experience (ref. 28), \( \delta_i^* \) must be used instead of \( \delta^* \) for the compressible case.

For the sublayer, or wall damping region, several expressions are available (e.g., refs. 34, 40, 174, and 175), but results in reference 169 indicate that, at least for some cases, most of these expressions yield very similar results. The most commonly used wall damping expression was developed by Van Driest (ref. 174) and is an exponential damping upon the mixing length \( \frac{A u_T}{v} = A^+ \) \( (A^+ = 26 \text{ for } dp/dx \text{ and } v_w \approx 0) \)  \hfill (41)
Therefore, in the wall region the mixing length model becomes

\[-u'v' = (Ky)^2 (1 - e^{-y/A})^2 \frac{\partial u}{\partial y} \frac{\partial u}{\partial y} \]  

(42)

In addition to modeling the Reynolds stress term in equation (2), solutions of the compressible turbulent boundary-layer problem also require a model for the Reynolds heating term (eq. (4)). This is generally handled through use of a turbulent conductivity expression

\[-v'h' = \kappa_t \frac{\partial h}{\partial y} \]  

(43)

which then allows the definition of a turbulent Prandtl number

\[\kappa_t = \frac{\epsilon}{N_{Pr,t}} \]  

(44)

Once \(N_{Pr,t}\) is known, the model for \(-u'v'\) can be used in equation (44) to determine \(\kappa_t\).

Therefore, in the usual mean field closure models, there are four closure "constants" which must be determined or specified before a solution can be obtained: \(A^+\) for the wall damping region, \(K\) for the law-of-the-wall region, \((1/\delta)_{\text{max}}\) or \(\alpha\) for the wake or outer region, and \(N_{Pr,t}\). These constants will now be examined in some detail, with particular emphasis upon their possible variation due to (1) compressibility, (2) low Reynolds number, (3) wall blowing, (4) pressure gradient, and (5) roughness.

Wall Damping Constant

This is the most variable of the four "constants." Available data, although not definitive, show no discernible effect of Mach number upon \(A^+\) (e.g., refs. 176 and 177). However, when \(A\) is computed from \(A^+\) for compressible flow \((A = A^+/\sqrt{\gamma/\rho})\), \(\gamma\), \(\tau\), and \(\rho\) can be evaluated at several places (at the wall, locally as a function of \(y\), and at the edge of the sublayer). Computational experience indicates evaluation at the wall is slightly better (refs. 53, 176, 177, and 178).

There is no discernible influence of low Reynolds number upon \(A^+\), or at least none has yet been identified (e.g., ref. 179). The influence of wall blowing upon \(A^+\), however, is quite large, and there exist several ways of accounting for this effect. One of the first procedures was a simple correlation plot of \(A^+\) and the wall injection similarity parameter \(2F/C_f\) (ref. 53). The data for this plot (fig. 26) were obtained from an examination of low-speed blowing data. By analogy to the \(F = 0\) case, the \(A^+\)-value was taken as the \(y^+\)-location where the data were faired into a typical law-of-the-wall variation (i.e., the outer edge of the sublayer-buffer region). This variation (fig. 26) worked quite well (ref. 53), and most of the other models tend to give similar results (ref. 180). The changes in \(A^+\) with wall mass transfer are quite large and some modification to \(A^+\) must be included in order to obtain a reasonable \(C_f\).
prediction. Probably the most general method of adjusting $A^+$ for blowing is given by Launder (ref. 171), who suggests $A^+ = 26/(\tau/\tau_w)^{1.7}$ (i.e., since $\tau/\tau_w = f(y)$, $A^+ = f(y)$).

The variation of $A^+$ with pressure gradient is also quite large. Cebeci and Smith (ref. 12) give a correction procedure, as does Kays (ref. 181). (See also ref. 19.) Again, probably the simplest approach is to use the Launder expression $A^+ = 26/(\tau/\tau_w)^{1.7}$ which accounts quite well for the effects of both mass injection and pressure gradient. The basic problem with the variation of $A^+$ with blowing and pressure gradient is indicated in sketch (b). As shown in the sketch, the wall damping occurs in a region where $\tau \neq$ Constant, and the various approaches involve either letting $A^+ = f(\partial \tau/\partial y)$ (ref. 179) or using the local $\tau(y)$ to correct the $A^+$ or $\tau_w$ value; that is (from ref. 171, for example),

$$A^+ = \frac{26}{(\tau/\tau_w)^{1.7}}$$

Another parameter which has a large effect upon $A^+$ is roughness. McDonald and Fish (ref. 182) give a correction term for roughness as follows:

Wall damping with roughness $\bar{\tau}$ (Wall damping without roughness)

$$+ \left[ 1 + \frac{k^+}{30(y^* + 1)} \right] \exp\left(\frac{-2.3y^+}{k^+}\right)$$

where

$$k^+ = \frac{ku_\tau}{\nu}$$

$$y^* = y \frac{u_\tau}{\nu}$$

34
$u_\tau$ is based on $\tau(y)$ and $k$ is the roughness height. It should be noted that this damping term (with roughness) could be greater than 1.

The influence of moderate acceleration (approaching laminarization, involving significant alteration of the sublayer) upon $A^+$ was investigated by Launder and Jones (ref. 183). They proposed the following expression:

$$A^+ = 11 + 7900L$$

where the pressure gradient parameter $L$ is

$$L = C_T^{-2/3} \frac{v}{u_e^2} \frac{du_e}{dx}$$

This expression has not yet received detailed consideration for compressible flows.

**Prandtl Wall Constant**

From available data, there is no appreciable effect upon $K$ of either compressibility (general computational experience, ref. 184), low Reynolds number (refs. 179, 185, and 186), wall blowing (refs. 184 and 185), or surface roughness (refs. 45 (pp. 396-398) and 187). There is, however, a moderate influence of pressure gradient upon $K$, as shown in figure 27 (taken from ref. 188). From the work of Lewis, Gran, and Kubota (ref. 139), the Clauser pressure gradient parameter $\beta_K$ (and therefore fig. 27) is applicable to compressible flow if $\delta^*$ is taken as $\delta\tau$. Except for apparent pressure gradient effects (results similar to those in fig. 27 were also observed in ref. 30 (pp. 10-1 - 10-13) for supersonic flow), the wall slope $K$ is the most constant of the mean field closure "constants."

**Outer or Wake Constant**

The present section discusses the behavior of $(1/\delta)_{\text{max}}$ (eq. (39)) instead of $\alpha$; the behavior of $\alpha$ (eq. (40)) is quite similar, and in most instances the two expressions (eqs. (39) and (40)) give similar results. Figure 28 provides a typical comparison between the use of equations (39) and (40) for calculation of surface shear. For this low hypersonic case with moderate wall cooling, the mixing length is in slightly better agreement with the data. From the classic work of Maise and McDonald (ref. 79), there is no appreciable Mach number effect upon $(1/\delta)_{\text{max}}$. There is, however, a large low Reynolds number effect. Recent research (ref. 189) has strongly indicated that this low-Reynolds-number effect is caused by the residue of the boundary-layer transition process.

One method of correlating the low-Reynolds-number effect is with a scaling parameter $\delta^+$, where

$$\delta^+ = \delta u_{\tau,max}$$

$$\frac{1}{(\nu)_{\tau,max}}$$
In references 176, 186, and 189 this parameter successfully correlated the low Reynolds number effect over a wide Mach number range. For the low-speed case a parameter \( R_e \) is sometimes used (ref. 175) when \( \frac{dp}{dx} = 0 \). Working with this expression for \( \delta^+ \), one can show that \( \delta^+ \propto \epsilon/\mu \) or a turbulence Reynolds number as follows:

\[
\delta^+ = \frac{\delta \sqrt{\epsilon}}{\mu} \tag{49}
\]

From the Townsend or Bradshaw assumption of the relationship between \( \epsilon \) and \( \tau \),

\[
\rho \overline{u'v'} \propto \epsilon \rho \propto \tau \tag{50}
\]

Therefore,

\[
\delta^+ \propto \delta \frac{\rho \sqrt{\epsilon}}{\mu} \tag{51}
\]

From the Prandtl model for \( \epsilon \)

\[
\epsilon \propto \frac{\eta}{\rho} \sqrt{\epsilon} \tag{52}
\]

and identifying \( \eta \) with \( \delta \) one obtains

\[
\delta^+ \propto \frac{\epsilon}{\mu} \tag{53}
\]

The extent of the low-Reynolds-number amplification is indicated in figure 29(a) (from ref. 189) where \((\eta/\delta)_{\text{max}}\) was derived from experimental velocity profiles for flat plates, cones, and cylinders. Increases of a factor of 2 or more in \((\eta/\delta)_{\text{max}}\) above the usual levels (0.07 to 0.09) for values of \( \delta^+ \) near 100 are observed. Since \( \overline{u'v'} \approx (\eta/\delta)_{\text{max}}^2 \), this represents an increase of a factor of 4 or more in turbulent shear. The evidence that this increase is a function of distance downstream of transition is given by comparison of figure 29(a) with figure 29(b) (also from ref. 189) where \((\eta/\delta)_{\text{max}}\) actually decreases with decreasing \( \delta^+ \) for the nozzle wall case (where transition generally occurs far upstream in the settling chamber). The correlation for inclusion of low Reynolds number effects should therefore be a function of both \( \delta^+ \) (or \( \epsilon/\mu \)) and \( \Delta x/\delta \); the number of boundary-layer thicknesses downstream of the end of transition. (According to ref. 190, a value of \( \Delta x/\delta \) of approximately 30 to 50 is needed to "wash out" the low Reynolds number effect.)

As Mach number increases, edge Reynolds numbers (such as \( R_{e,x} \), \( R_{e,\theta} \), etc.) become proportionally much larger than Reynolds numbers based on wall conditions (such as \( \delta^+ \), etc.). This is due to the large difference, at least for wind-tunnel conditions, between the wall and boundary-layer-edge temperatures. Therefore, as shown quantitatively in figure 30, a given value of \( \delta^+ \) (which seems to correlate the transition-induced low Reynolds number effects) can correspond to a very large value of \( R_{e,x} \). This implies that the low Reynolds number amplification can effect a rather large portion of the boundary-layer flow on a hypersonic vehicle.
The outer or wake constant \((\nu/\delta)_{\text{max}}\) is also influenced by wall blowing and pressure gradient. From the data shown in figure 31 (from ref. 53) the level of \((\nu/\delta)_{\text{max}}\) decreases as the profile becomes less full and \(d\delta/dx\) increases. Reference 186 contains an excellent explanation for this effect, which involves the problem of flow history effects. A detailed discussion of this problem is postponed until later when nonequilibrium flows are considered. As a final point, for the outer region model there seem to be only moderate increases of \((\nu/\delta)_{\text{max}}\) with roughness (ref. 191).

### Turbulent Prandtl Number

When dealing with \(N_{\text{Pr},t}\), there is a basic question of which term to model, \(v'H'\) or \(v'h'\). Since \(H' = h' + \bar{u}u'\), then \(v'H' \approx v'h' + \bar{u}u'v'\) and therefore \(N_{\text{Pr},T} \equiv \epsilon/\kappa_T\) (where \(v'H' = \kappa_T(3H/3y)\)) cannot equal \(N_{\text{Pr},t} \equiv \epsilon/\kappa_t\) (where \(v'h' = \kappa_t(3h/3y)\)), except for \(M \approx 0\). This question is discussed in some detail in references 53 and 192. The results of reference 177 clearly show that \(N_{\text{Pr},t}\) is much more invariant with Mach number than \(N_{\text{Pr},T}\). Typical \(N_{\text{Pr},t}\) distributions inferred from profile data are shown in figure 32 for low-speed and supersonic flow. Figure 32 shows data from references 181 and 193 to 196. An expanded scale is used, and the scatter, except near the wall, is typically \(+10\) percent about a hypothetical mean line through the data. The scatter near the wall is probably caused by inaccurate mean profile data in that region. The data from reference 177 showing the problems with \(N_{\text{Pr},T}\) and \(N_{\text{Pr},t}\) are shown in figure 33. The \(N_{\text{Pr},t}\) data at \(M = 7.2\) agree fairly well with the low-speed data in figure 32, indicating that there may not be a strong Mach number effect on \(N_{\text{Pr},t}\).

The evidence for a possible influence of low Reynolds number upon \(N_{\text{Pr},t}\) is primarily circumstantial, but indicates little or no influence (at least for air). Figure 34 summarizes some of the more accurate Reynolds analogy data (from refs. 197 to 201) for the low Reynolds number case. Since \(N_{\text{Pr},t}\) is a measure of the relationship between turbulent shear and turbulent heating, any large changes in \(N_{\text{Pr},t}\) might be expected to affect the Reynolds analogy factor. The available data do not seem to exhibit a strong trend with \(\delta^+\), at least over the limited range for which data are available.

A second piece of circumstantial evidence as to the effect of low Reynolds number on \(N_{\text{Pr},t}\) is indicated in figure 35 (taken from ref. 12). The curves shown in this figure are the result of purely theoretical calculations based upon an assumed model of turbulence, but they do indicate only a very small effect of \(\epsilon/\mu\) upon \(N_{\text{Pr},t}\) for the \(N_{\text{Pr}} \approx 0.7\) case. However, if the low Reynolds number effect comes from the persistence of transitional flow structures, this evidence is not really generic. In the absence of further data no firm conclusion can be drawn concerning the influence of low Reynolds number on \(N_{\text{Pr},t}\) except that no large effect has yet surfaced.

The variations of \(N_{\text{Pr},t}\) with \(y^+\) (fig. 32) could also be interpreted as indicating a variation of \(N_{\text{Pr},t}\) with \(\epsilon/\mu\), but this influence of low Reynolds number near the wall results in reduced shear (e.g., ref. 202) as opposed to the influence of low Reynolds number in transitional flow structures, which results in increased shear. Therefore, as stated previously, \(\epsilon/\mu\) (or \(\delta^+\) or \(y^+\)) is
not sufficient to adequately identify or correlate the various low Reynolds number influences. These low Reynolds number problems obviously require further definitive research, especially in regard to $N_{Pr,t}$.

The data from references 196, 203, and 204 indicate only a small influence of wall blowing on $N_{Pr,t}$. In reference 204, data with near equilibrium pressure gradients indicate that $N_{Pr,t}$ values do become somewhat lower in the law-of-the-wall region for the positive $dp/dx$ case.

Intermittency Normal to the Wall

Another ingredient often used in mean field closure procedures is a normal intermittency factor $\Gamma_{y}$, which is generally defined as the percentage of time that the flow is turbulent. This factor accounts for the superlayer or intermittent nature of the outer region in turbulent boundary layers. For high Reynolds number calculations ($(\varepsilon/\mu)_{\text{max}} > 100$ to 200) the inclusion of $\Gamma_{y}$ is generally a second-order effect (refs. 45 (pp. 366-374), 53, and 205); however, $\Gamma_{y}$ can become important for $(\varepsilon/\mu)_{\text{max}} < 100$ (ref. 53).

Typical distributions of $\Gamma_{y}$ are shown in figure 36 (from ref. 25) for the low-speed and hypersonic cases. These data indicate that a fully turbulent flow occurs much farther out in the boundary layer for the high Mach number case, and that, even for the simple $dp/dx \approx 0$ case, $\Gamma_{y}$ is a function of $M$. In addition to this Mach number effect, the data of Fiedler and Head (ref. 206) show that $\Gamma_{y}$ can be a strong function of pressure gradient. The influence of roughness on $\Gamma_{y}$ is evidently fairly small (ref. 207).

APPLICATIONS OF MEAN FIELD CLOSURE METHODS

Particularly during the last 8 years the literature has been rife with comparison between theory and data for mean field prediction methods. These comparisons indicate that mean field closure approaches yield predictions which are quite accurate over a wide range of conditions. (See especially refs. 12, 28, and 30.) In this section, comparisons of prediction and data for the more usual cases (such as $dp/dx = 0$, wall blowing, or equilibrium flows) will not be considered, but instead, the more unusual situations, such as a diagnosis of nozzle wall boundary layers and transitional flows, among others, will be treated.

The Case of the Nozzle Wall Turbulent Boundary Layer

Thick, fully developed turbulent boundary layers have been difficult to obtain on models at high Mach number as a result of (1) the increase in transition Reynolds number with Mach number (e.g., ref. 208), (2) the notorious difficulty in tripping high Mach number boundary layers, and (3) the general decrease in test section size of hypersonic facilities compared with the low-speed facilities. Therefore, a great deal of the hypersonic turbulent boundary-layer profile data were taken in nozzle wall boundary layers, which are readily available and usually of the order of 5 to 50 cm thick.
However, as can be readily seen with the help of sketch (c), there are several problems with these nozzle wall boundary-layer data. First of all, the measuring stations usually cover a small range of $\Delta x/\delta$ (data taken in test section region only with $\delta$ quite large). These data are therefore not very satisfactory as test cases for the mean field methods, as these procedures are of the parabolic type and the solution proceeds in the streamwise direction. The data are usually taken at a specified $x$-location for a range of unit Reynolds number, rather than over a respectable range of $\Delta x/\delta$ at a given unit Reynolds number.

The second problem involves so-called "history" effects. Although the flow may locally inhabit a region where $dT_w/dx = 0$ and $dp/dx = 0$, just a few boundary-layer thicknesses upstream the gradients in these quantities are often quite large and there is a question of how fast the wall boundary layer loses the "memory" of these gradients. Of course, it is possible to make measurements at several stations along the nozzle, starting with a measured profile near the high gradient (large $dp/dx$) region (such as in ref. 47). This produces a quite interesting but theoretically undemanding test case (boundary-layer recovery from a favorable pressure gradient).

An additional problem in the nozzle wall data is that of low Reynolds number similitude. This problem concerns the early transition (usually in the settling chamber) and subsequent early "loss of memory" of the transitional flow structures. This problem was covered previously in connection with figure 29 and indicates that the nozzle wall data for low values of $\delta^+$ may be applicable only to a limited class of applied flows, such as where roughness induces early transition on the nose of a vehicle before the boundary layer undergoes the expansion to afterbody flow conditions.

Several investigators have studied these various nozzle wall problems (such as refs. 176 and 209 to 212). Figure 37 indicates typical total temperature and velocity relationships obtained on flat plates, cones, and cylinders (small $dp/dx$ and $T_w(x)$ history), and figure 38 illustrates the nozzle wall case. These figures (taken from ref. 208) are now quite old (made up approximately 8 years ago); however, the differences between the flat plate and nozzle wall data (that is, the differences between a linear variation of $T_\ell$ and $u$ as opposed to a quadratic variation) have also been consistently observed in the more recent data. The development of the quadratic variation is clearly seen in figure 39 (taken from ref. 212).
In summary, the current status of the nozzle wall data indicates that for $M > 5$, an adiabatic wall, and large values of $\delta^+$ (greater than 2000), the nozzle wall profile data in the test section region are fairly typical (in approximate agreement with the flat plate case). (See ref. 190.) For large values of $\delta^+$, $M > 5$, and a nonadiabatic wall, the velocity profiles are still fairly typical (ref. 190), but the $T_t$ profiles are evidently out of equilibrium (ref. 209). For low values of $\delta^+$ (without relaminarization downstream of the throat), the profiles are evidently correct only for practical cases where transition is far upstream.

Influence of Wall Blowing Upon Skin Friction

The conventional method of representing the influence of porous wall injection upon skin friction is with a similarity plot of $C_f/C_{f_0}$ against $2F/C_{f_0}$. An exhaustive review article by Jeromin (ref. 33) presents the available data in these coordinates. (See fig. 1 of ref. 33.) Jeromin's plot indicates a strong Mach number effect upon skin friction reduction due to blowing. Several predictors (refs. 213 to 215) have examined the question of a Mach number effect in these coordinates, two of these since the publication of Jeromin's review. Reeves (ref. 30 (pp. 6-1 - 6-A2-2)) calculated a weak Mach number effect, but his results indicated separate influences of Reynolds number and the ratio of wall temperature to total temperature, which could account for some of the apparent Mach number effects. Squire and Verma's (ref. 213) calculations (fig. 28 of ref. 213), in which a conventional mean field turbulence modeling procedure was employed, indicated very little influence of either Reynolds number, Mach number, or $T_w/T_t$, at least for $R_e^{*} > 8000$ (the lower limit for their calculations). This result is similar to conclusions reached in the early work of Rubesin (ref. 215). Landis (ref. 214) attributed most of Jeromin's Mach number effect to the influence of $T_w/T_t$.

The data which indicate the strongest Mach number effect are those of Danberg (ref. 216). When the low Reynolds number effects are included in the analysis of Danberg's data (value of $C_{f_0}$ redefined), the final result for the available Mach number range (data and theory) is shown in figure 40 (from ref. 217). There is apparently very little Mach number effect, and the data are correctly predicted by the mean field closure methods. There is insufficient experimental data to determine any possible effect of the ratio of wall temperature to total temperature.

Transitional Flow Calculations

The now classical approach to calculation of the transitional flow region between laminar and turbulent flow is to multiply the usual $\overline{u'v'}$ model by a streamwise intermittency factor $\Gamma_x$ to account for the increasing presence of turbulent bursts (e.g., refs. 142, 153, and 164). Using the definitions of $x_{tr}$ and $x_{tr,end}$ indicated in sketch (d), the expression usually used for the intermittency factor is (from ref. 218)

$$\Gamma_x = 1 - e^{-0.412 \left( \frac{x-x_{tr}}{\Delta x} \right)}$$
where

\[ \Delta x = \frac{x_{\text{tr,end}} - x_{\text{tr}}}{2.96} \]  

(54)

To apply this expression to a problem or a set of data, the \( x_{\text{tr}} \) and \( x_{\text{tr,end}} \) values must be known. Typical values of the ratio \( x_{\text{tr,end}}/x_{\text{tr}} \) (or the ratio \( R_{\text{tr,end}}/R_{\text{tr}} \), which is the same thing for \( dp/dx = 0 \)) are shown in figure 41 (from ref. 219). A nominal value of 2 for the ratio of \( x_{\text{tr,end}} \) and \( x_{\text{tr}} \) is seen to be a reasonable approximation over a large Mach number range. (See also ref. 201.) Therefore, the problem is reduced to a specification of \( x_{\text{tr}} \), a discussion of which is beyond the scope of the present report. (See, for example, ref. 220.) A comparison between equation (54) (which gives low-speed values of \( \Gamma_X \)) and hypersonic measurements of \( \Gamma_X \) is shown in figure 42 (from ref. 59). The high-speed data are in relatively good agreement with the curve from equation (54), indicating that the latter may be used with some confidence.

For accurate transitional flow calculation in compressible flow, the precurser effects (discussed previously) should be included. An example of the increased agreement obtained when these precurser and low Reynolds number effects are included is given in figure 43 (from ref. 58).

Additional "Nontypical" Applications of Mean Field Closure Methods

Variable edge entropy. - This problem is caused by the mass "swallowed" or entrained by the boundary layer along the afterbody of slightly blunted vehicles. (See sketch (e).) As the boundary layer swallows the high entropy streamlines, the edge properties possess a variable entropy condition. The actual edge conditions are determined by equating mass flow in the boundary layer at a given body station \( \dot{m}_{\text{bl}} \) to the mass flow in an entering stream tube \( \dot{m}_{\text{shock}} \). This problem was treated in reference 221, using a mean field closure approach. Figure 44 (from ref. 221) indicates the better agreement resulting from consideration of variable entropy.

Transverse curvature influence. - Cebeci and Smith (ref. 12) have treated this case quite well, using a mean field approach. The problem arises when \( \delta/r_c \approx O(1) \), where \( r_c \) is the local body transverse radius of curvature.
According to reference 12, the outer region formulation is unchanged. The inner and wall damping regions are altered to

$$l = K\left(\frac{r}{r_0}\right)^{1/2} r_0 \ln \left(\frac{r}{r_0}\right) \left[ -\frac{r_c}{A} \ln \left(\frac{r}{r_c}\right) \right]$$

Using this expression Cebeci and Smith obtained good agreement with data on a 0.061-cm-diameter needle at $M_\infty = 5.8$.

**Longitudinal curvature influence.**—The definitive review work in this area is a fairly recent report by Bradshaw (ref. 222). The usual effect is for convex curvature to reduce the Reynolds stress and entrainment and for concave curvature to increase the shear over the no-curvature level for the same pressure gradient. There are evidently four distinct effects of longitudinal curvature upon compressible turbulent boundary-layer physics and calculations. The first of these effects is the influence of additional longitudinal curvature terms in the mean flow equations. These extra terms were included in the calculation methods of references 167 and 223 and calculations in reference 224 using the code of reference 167; the results of these calculations indicate better agreement for a flow with mild concave curvature when these terms are included.

The second effect is unique to compressible flows, in that longitudinal curvature can induce a large pressure gradient in the boundary layer. This influences the flow even beyond the boundary layer and creates a nonuniform free stream (i.e., induces $u_\infty(y)$ for $y > \delta$). This occurs primarily as a result of the Prandtl-Meyer type of flow turning which is a typical feature of compressible flows. Therefore, the expression $\partial p/\partial y = 0$ is no longer true, and the correct $p(x,y)$ behavior must be included in boundary-layer prediction procedures; that is, one must have the correct value of $p_{local}$ to compute a density, and $\partial p/\partial x$ is now a function of $y$. The codes of references 167 and 223 both include this capability.
The third effect of longitudinal curvature is the influence on the turbulent structure itself. Bradshaw (see ref. 222) has given a correction factor of the form

$$\frac{1}{\delta} = \left(\frac{1}{\delta}\right)_t \left(1 - B \frac{\bar{u}/r_c}{\delta u/\delta y}\right)$$

where

$$B \approx 0(10)$$

This term is generally applied in an average sense to the outer region flow using the wall curvature for $r_c$ and provides a first-order correction for moderate curvature (herein defined as convex radius of curvature large enough so that laminarization does not occur and concave radius of curvature large enough so that Goertler vortices are not important and separation does not occur).

The fourth effect of longitudinal curvature is illustrated by experimental data, which indicate that for $\delta/r_c > 0.005$ (which is not very large), concave curvature can generate steady, longitudinal Goertler vortices embedded in the outer portion of the turbulent boundary layer (ref. 225). The presence of Goertler vortices converts a readily solvable two-dimensional boundary-layer problem into a more complex three-dimensional turbulence problem of the parabolic-elliptic type (recirculation in the crossplane). Much more research is needed before quantitative predictions about the latter type of flow can be made.

Calculations for adverse pressure gradient in compressible flow. - There is currently some controversy concerning the capacity of turbulent boundary-layer calculation methods to compute adverse pressure gradient flows for the compressible case. Bradshaw, in reference 226, partly on the basis of disagreements between his procedure and data, suggests that a mean dilitation correction is needed before satisfactory results can be obtained. Part of the problem in this area is due to the fact that the early (pre-1969) data for adverse pressure gradient compressible turbulent flows were taken on bodies where positive values of $dp/dx$ were induced by longitudinal curvature, thereby introducing all the possible problems just discussed. However, Bradshaw (in ref. 226) considers the newer data, where the waves causing the adverse pressure gradient are impressed upon a flat surface.

One example of an adverse pressure gradient calculation (on a flat surface) is given in figure 45, and the external Mach number distribution for this flow is given in figure 46. The data are from reference 139 and the mean field calculation method is taken from reference 12. In this case the comparison is quite good. A check calculation made by the present authors using the method of reference 53 yielded similar agreement with the data. On the basis of these results, the question of whether Bradshaw's dilitation correction factor (ref. 226) is really required to compute adverse pressure gradient compressible flow is still open.
NONEQUILIBRIUM AND MEAN TURBULENT FIELD CLOSURE

Physical Problem

Primarily as a result of the large difference in scales between the inner (near wall) and outer portion of turbulent boundary layers, the inner portion (with scales of the order of $y$) reacts much faster to a change in boundary conditions than the outer region (with scales of the order of $\delta$). This is shown schematically in sketch (f). The value of $\Delta x/\delta$ necessary for the outer region flow to "relax" and equilibrate with the new boundary conditions is of the order of 30 to 90.

Therefore, as discussed earlier, for rapid changes in boundary conditions a rate equation is needed for the computation of $u'v'$ in the outer region (and perhaps for $A^+$ as well; see ref. 181). The turbulent field cannot instantaneously follow or track rapid changes in the mean flow. A nonequilibrium situation can be set up by the sudden removal or application of a pressure gradient, wall injection or suction, wall temperature gradient, or wall roughness. Larger changes, such as those caused by shock-wave impingement and large obstacles or steps, may cause the flow to violate the boundary-layer assumptions. In addition to the computation of nonequilibrium flows, $u'v'$ rate equations and mean turbulence field models are also necessary for including free-stream disturbance boundary conditions and for calculating the various turbulent second-order correlations.

There is a zeroth-order nonequilibrium modeling which involves a rate equation for the mixing length or outer region constant. This equation is usually of the form

$$\delta_i^* \frac{d\varepsilon}{dx} = C(\varepsilon_{\text{equilibrium}} - \varepsilon)$$  \hspace{1cm} (57)

Simple expressions or rate equations of this general form were used in references 30 (pp. 6-1 - 6-A2-2 and 29-1 - 29-10), 45 (pp. 375-383), 227, and 228 and do tend to give at least the correct qualitative behavior, the problem being that the decay constant can change with the particular flow involved; that is, the equation has the correct form but the absolute rate of return to equilibrium is uncertain. Therefore, the balance of this section is devoted to methods based upon rate equations derived from the Navier-Stokes equations (second-order correlations or Reynolds stress transport equations).

Second-order equation closure methods (or mean turbulence field methods) have been applied rather extensively to low-speed turbulent boundary layers.
(See refs. 21, 172, and 229 for reviews of this work.) However, their application to compressible flows has been rather limited, perhaps as a result of the following: (1) relative success of the mean field methods, (2) dearth of accurate data for nonequilibrium compressible turbulent boundary-layer flows (i.e., very few interesting test cases), (3) questions concerning possible importance of \( p' \) and \( \rho' \) terms in these equations, and (4) lack of sufficient data on the fluctuation field for the compressible case (for modeling purposes).

Equations and Modeling for Compressible Nonequilibrium Flows

The Reynolds stress transport equations (or \( R_{ij} \) equations) are generally obtained by (1) multiplying the Navier-Stokes equations for \( u_i \) by \( u_j \), (2) multiplying the Navier-Stokes equations for \( u_j \) by \( u_i \), (3) adding these two equations, (4) substituting for instantaneous values the usual mean plus fluctuation portions and Reynolds averaging, (5) subtracting out the kinetic energy equation for the mean flow, and (6) imposing the usual boundary-layer order of magnitude analysis. The result of all this is an equation for \( u_i' u_j' \). From this equation one can obtain (1) an equation for \( u_j'^2 \) (\( i = j = 1, 2, 3 \)), (2) an equation for \( \overline{u_i' v_j'} \) (\( i = 1, j = 2 \)), and (3) an equation for \( e = u_1'^2 + v_1'^2 + w_1'^2 \) (sum of equations for \( u_j'^2 \)). The general form of these second-order equations is usually the following:

\[
\text{Convection} = \text{Production} + \text{Diffusion} - \text{Dissipation}
\]

Details of the derivation of these equations are available, for example, in references 36, 42, 148, 159, and 230 for the compressible turbulent kinetic energy equation and in references 43 and 156 for more complete sets of the second-order compressible equations. Reference 12 represents probably one of the best references for general discussion and derivation of the second-order equations for both low and high speeds.

From computational experience, the critical portion of the development of a good nonequilibrium turbulent boundary-layer calculation procedure is an accurate equation for the length scale of turbulence (refs. 36 and 45, pp. 275-299). The results of the calculations for nonequilibrium flows are usually not extremely sensitive to the details of the modeling used in the second-order correlation equation, but the length scale determination must be nearly correct for a good prediction.

There are three approaches usually used to derive a length scale equation: (1) form an equation for the turbulence dissipation \( D \) and relate this to \( L \) through the usual model for \( D \) (\( D \approx e^{3/2}/L \)) (ref. 8); (2) use an equation for the two-point correlations, which introduces a length scale quite naturally; and (3) use the turbulent vorticity equation and relate to \( L \), for example, through the expression \( L = e^{1/2}/\omega \) (refs. 231 and 232). An alternate approach to the length scale problem is to use an algebraic relationship (such as a function of \( y/\delta \), for example), which is assumed known (ref. 148). This latter method does not yield satisfactory results for truly nonequilibrium flows (if the length scale being solved for is the one used in modeling the Reynolds stress).
Single-Equation Models

Bradshaw approach.- This method (ref. 36) gives a second-order outer region solution based upon the conversion of the turbulent kinetic energy equation into an equation for $u'v'$ by assuming that $a_1 = \tau/2\rho e = \text{Constant}$. The only length scale which appears is in the dissipation term; this is a very important aspect of the method, as Huffman (ref. 233) has shown that this length scale $L$ (for dissipation) is much more invariant than the usual mixing length (which is used to model turbulence production), as can be seen by comparing figures 47 and 48 (from ref. 233). Bradshaw's $a_1$ is also relatively invariant, even for nonequilibrium flows, as indicated in figures 49 (from ref. 233) and 50 (from ref. 234). Bradshaw, by using $a_1$, evidently sidesteps the problem of modeling the production term with a length scale and then having to derive and solve an equation for that scale. The length scale which Bradshaw uses (in the dissipation term) is evidently fairly invariant, and therefore, his one-equation method ($u'v'$ equation plus algebraic length scale) should yield predictions which are as accurate as the more common two-equation models ($e$ and $\omega$ equations).

Method of Shamroth and MacDonald and method of Chan.- These procedures (refs. 159 and 155) use the integral form of the turbulent kinetic energy equation and Bradshaw modeling ($a_1$ and $L$). The procedures are, at least approximately, integral forms of the Bradshaw closure approach and seem to be fairly successful in predicting nonequilibrium flows. The method is inherently fast and relatively accurate and is a good choice for a simple nonequilibrium boundary-layer calculation procedure.

Two-Equation (and More) Models for Compressible Flows

Method of Wilcox and Alber.- This procedure (ref. 231) solves the turbulent kinetic energy equation and the fluctuating vorticity equation. Mass-weighted averages are used along with the Prandtl model for $\overline{u'v'} = \overline{e^{1/2} \frac{d u}{d y}}$.

Method of Spalding and Gibson.- This method (ref. 232) is quite similar to that of reference 231 in basic concept ($e$, $\omega$ equations used) but quite different in detail and application. Neither of these procedures has yet seen much application in highly nonequilibrium compressible flows.

Method of Donaldson and Sullivan.- This method (ref. 156) solves the complete set of second-order boundary-layer equations ($u'^2$, $v'^2$, $w'^2$, $u'v'$, $T'T'$, $u'T'$, and $v'T'$) and therefore does not have to use a turbulent Prandtl number. However, most of the $p'$ type terms were neglected. The latest version of this method (in publication) includes a length scale equation. The type of detailed information obtainable from this type of closure is shown in figures 51 and 52.

At the present time there are insufficient detailed data available to develop "calibrated" nonequilibrium (mean turbulence field) closure methods for compressible turbulent boundary layers.
CALCULATION OF THREE-DIMENSIONAL COMPRESSIBLE TURBULENT BOUNDARY LAYERS

Status

A general method for solving three-dimensional compressible turbulent boundary-layer flows is not currently available; however, a limited number of problems have been solved using special techniques for each case. Blottner (ref. 235) presents a review of specialized and approximate solutions of the laminar three-dimensional boundary-layer equations. General three-dimensional boundary-layer computer codes require the following: (1) an efficient and accurate numerical procedure, including the required logic to automatically change the difference molecule as a function of local flow conditions, (2) a generalized curvilinear coordinate system convenient for the design engineer, (3) accurate turbulence models for the Reynolds stresses, and (4) a three-dimensional inviscid flow field solution which is compatible with the boundary-layer coordinate system.

A number of general numerical procedures and computer codes have been developed for two-dimensional and axisymmetric turbulent boundary-layer flows (e.g., see refs. 53, 142, 164, 202, and 236 to 242). This success was extended to a particular class of three-dimensional flows designated as quasi-two-dimensional (retains computational advantage of two independent variables while allowing large cross-flow components) by Hunt, Bushnell, and Beckwith (ref. 243). Numerical solutions for general three-dimensional flows did not materialize until after 1972 because of computer systems limitations (storage and processing speed) and the lack of accurate three-dimensional inviscid flow field solutions required for boundary-layer-edge input conditions.

Since 1972 an intensive research program on three-dimensional boundary-layer flows has been under way at a number of research centers. This effort is stimulated by increasing awareness of potential savings in cost and man-hours through numerical simulation of complex flows; often these flows cannot be simulated in ground test facilities (such as, for example, real-gas boundary-layer flow over the space shuttle). Substantial progress has been made as a result of the increasing availability of large-storage, high-speed computer systems and the maturing of accurate numerical procedures and computer codes for solving the three-dimensional inviscid equations for complex configurations. (See refs. 244 and 245.) Some of the more important areas of three-dimensional boundary-layer research are as follows: coordinate systems and transformations, numerical solution techniques, turbulence modeling, geometry, initial conditions, inviscid boundary conditions, inflow lines, regions of influence and dependence, and numerical optimization for perfect-gas flows as opposed to real-gas flows. These specific areas will be discussed in detail in subsequent sections of the present review.

The primary purpose of this section is to discuss in detail problems associated with development of three-dimensional compressible turbulent boundary-layer codes for application to complex aeronautical and aerospace vehicles and to indicate progress to date in achieving this goal. Comparisons of numerical results
with experimental data for several operational three-dimensional boundary-layer computer codes will also be presented.

Problems associated with obtaining accurate three-dimensional flow field results will not be discussed. In reality, the prediction of the inviscid flow field with the accuracy required presents a challenge as difficult as solving the three-dimensional compressible turbulent boundary-layer equations; however, the inviscid flow field input will be assumed available in the present review. (See, for example, ref. 246.)

Boundary-Layer Equations

An orthogonal curvilinear coordinate system is generally used to define the surface over which the boundary layer is flowing. The coordinate normal to the surface is \( x_3 \) with \( x_3 \) being zero on the surface. The lines \( x_1 = \text{Constant} \) and \( x_2 = \text{Constant} \) define the system of orthogonal coordinates on the surface. (See fig. 53.) The square of the element of arc \((ds)\) in the boundary layer is

\[
ds^2 = (h_1 \, dx_1)^2 + (h_2 \, dx_2)^2 + (h_3 \, dx_3)^2 \quad (58)
\]

where \( h_1, h_2, \) and \( h_3 \) are metric coefficients (or scale factors):

\[
h_1 = h_1(x_1,x_2) \quad h_2 = h_2(x_1,x_2) \quad h_3 = 1
\]

The metric coefficient \( h_3 \) is generally assumed to be unity \((h_3 = 1)\) since the boundary layer is assumed to be thin. The governing equations (first order) can then be written as follows, where \( h_3 \) is included for generality:

Continuity

\[
\frac{\partial}{\partial x_1}(h_2h_3\rho u_1) + \frac{\partial}{\partial x_2}(h_1h_3\rho u_2) + \frac{\partial}{\partial x_3}(h_1h_2\rho u_3) = 0 \quad (60)
\]

\( x_1 \)-momentum

\[
\frac{\rho u_1}{h_1} \frac{\partial u_1}{\partial x_1} + \frac{\rho u_2}{h_2} \frac{\partial u_2}{\partial x_2} + \frac{\rho u_3}{h_3} \frac{\partial u_3}{\partial x_3} - \rho u_1u_2K_1 + \rho u_2^2K_2 = - \frac{1}{h_1} \frac{\partial p}{\partial x_1} + \frac{1}{h_3} \frac{\partial}{\partial x_3} \left( \frac{\mu}{h_3} \frac{\partial u_1}{\partial x_3} - \rho u_1u_3 \right) \quad (61)
\]

\( x_2 \)-momentum

\[
\frac{\rho u_1}{h_1} \frac{\partial u_2}{\partial x_2} + \frac{\rho u_2}{h_2} \frac{\partial u_2}{\partial x_2} + \frac{\rho u_3}{h_3} \frac{\partial u_2}{\partial x_3} - \rho u_1u_2K_2 + \rho u_1^2K_1 = - \frac{1}{h_2} \frac{\partial p}{\partial x_2} + \frac{1}{h_3} \frac{\partial}{\partial x_3} \left( \frac{\mu}{h_3} \frac{\partial u_2}{\partial x_3} - \rho u_2u_3 \right) \quad (62)
\]
The geodesic curvatures of the surface coordinate lines are

\[ K_1 = -\frac{1}{h_1 h_2} \frac{\partial h_1}{\partial x_2} \]  
\[ K_2 = -\frac{1}{h_1 h_2} \frac{\partial h_2}{\partial x_1} \]  

The governing equations are completed with the perfect-gas equation of state (see refs. 245 and 247 for real-gas flows)

\[ p = \rho RT \]  

and suitable models or transport equations for the Reynolds stress terms appearing in the equations.

Obvious differences between the equations for three-dimensional (eqs. (60) to (67)) and two-dimensional (eqs. (1) to (4)) boundary-layer flows are the appearance of the cross-flow terms in the continuity and streamwise momentum equations and the addition of a second momentum equation for the cross-flow direction. However, the most important difference between two- and three-dimensional equations is that the three-dimensional equations are hyperbolic rather than parabolic in coordinate planes parallel to the wall boundary. The hyperbolic character of the three-dimensional equations has been recognized by a number of authors (see refs. 248 to 257) and arises naturally from the variation of the cross-flow along the coordinate line normal to the wall boundary \((x_1 = \text{Constant}, \ x_2 = \text{Constant})\). Streamlines originating at different points along this normal line diverge in the downstream direction. These streamlines originate at different upstream locations; consequently, wedge-shaped regions of influence and dependence are associated with three-dimensional viscous flow and extend upstream and downstream from the computational point along the normal to the wall boundary. This characteristic of the three-dimensional boundary-layer equations is generally referred to in the literature as the influence.
principle. (See ref. 248.) A schematic showing the regions of dependence and influence is presented in sketch (g). A more complete discussion is presented by Raetz (ref. 248), Kitchens, Gerber, and Sedney (refs. 254 and 257) and Wang (ref. 256).

The required boundary conditions for the governing equations (eqs. (61) to (63)) are as follows:

\[
\begin{align*}
  x_3 &= 0 \text{ (wall)} \\
  u_1 &= 0 \\
  u_2 &= 0 \\
  u_3 &= u_w(x_1, x_2) \\
  T &= T_w(x_1, x_2) \quad \text{or} \quad \left(\frac{\partial T}{\partial x_3}\right) = \dot{q}_w(x_1, x_2) \\
\end{align*}
\]

\[\text{Eq. (68)}\]

\[
\begin{align*}
  x_3 &= \infty \text{ (edge)} \\
  u_1 &= u_{1,e}(x_1, x_2) \\
  u_2 &= u_{2,e}(x_1, x_2) \\
  T &= T_e(x_1, x_2) \\
\end{align*}
\]

The governing equations for the inviscid flow at the outer edge \((x_3 = \infty)\) are evaluated from equations (61) to (63) as follows, where the static temperature is utilized instead of the total enthalpy:

\[
x_1\text{-momentum}
\]

\[
\frac{u_{1,e}}{h_1} \frac{\partial u_{1,e}}{\partial x_1} + \frac{u_{2,e}}{h_2} \frac{\partial u_{1,e}}{\partial x_2} - u_{1,e} u_{2,e} e K_2 + u_{2,e}^2 e K_1 = - \frac{1}{\rho_e h_1} \frac{\partial p}{\partial x_1} \]

\[\text{Eq. (69)}\]
Variable entropy and vorticity can be treated by the system of equations; however, its proper treatment is very complex for three-dimensional flows. The reader is referred to Mayne (ref. 255) and Popinski and Davis (ref. 258) for blunt cone flows and to Kendall et al. (ref. 245) for general configurations. The proper treatment of variable entropy is of primary importance for some applications such as the space shuttle.

Two general approaches may be followed in evaluating the required edge conditions: (1) the pressure gradient terms required in equations (61) and (62) \((dp/dx_i, \ i = 1, 2)\) can be obtained by direct solution of the Euler equation, or (2) the pressure \(p(x_1,x_2)\) can be specified together with the appropriate boundary and initial conditions and \(u_{1,e}, u_{2,e}\) and \(T_e\) obtained directly from the solution of equations (69) to (71). The optimum approach is to develop simultaneously an accurate three-dimensional inviscid computer code with a coordinate system compatible with that developed for the three-dimensional boundary-layer program; in any event, accurate and consistent edge conditions are required if accurate boundary-layer solutions are to be obtained.

The sufficient conditions required to start the boundary-layer solution have been presented by Ting (ref. 259). In principle the system of equations can be solved numerically by marching parabolically away (downstream) from specified initial data planes; however, the influence principle must be carefully treated.

Coordinate Systems and Transformations

Coordinate systems.- A number of papers have been published on the solution of three-dimensional boundary-layer flows; however, these papers have generally used specialized coordinate systems for each test problem. The coordinate system was generally chosen to simplify the governing equations for the particular flow of interest. The primary concern in the selection of a coordinate system is that it should allow the procedure to start from the initial conditions and proceed in a logical fashion over the entire surface without having to match together two or more distinct coordinate systems.

The majority of the references available have used streamline coordinates. (See ref. 260, for example.) The streamline coordinate system is an orthogonal surface coordinate system formed by the projection of the inviscid streamlines and their orthogonal trajectories on the surface. The system has a number of distinct advantages; however, its calculation is a major numerical effort in itself and must unfortunately be repeated for each change in flow conditions.
Another disadvantage is encountered in the design of wings and control surfaces where displacement surface effects must be accurately treated (for example, supercritical wing design). The coordinate system has to be recalculated for each iterative cycle between the three-dimensional inviscid and boundary-layer programs. This represents a major undertaking and would be prohibitively expensive in terms of computer processing time. Streamline coordinates also tend to diverge greatly in highly three-dimensional flow, which results in large truncation errors unless additional streamlines are introduced in order to retain a reasonable mesh point spacing in the direction normal to the streamline coordinate. Consequently, it should be realized that while streamline coordinates are optimum for many simple flows (for example, three-dimensional stagnation point flows), they do not appear at the present to be optimum for general three-dimensional flows. It is then advantageous to seriously consider the development of a body-oriented surface coordinate system that would completely eliminate the requirement of coordinate system recalculation for each change in flow condition.

A body-oriented surface coordinate system could be made compatible with the inviscid three-dimensional computer program used to specify the required edge conditions. The only disadvantage of an orthogonal surface coordinate system is that the initial data lines cannot in general be made to coincide with the body coordinate lines even for simple geometry, such as a blunt nose cone at angle of attack. Blottner and Ellis (ref. 261) have presented an orthogonal surface coordinate system whose origin is at the stagnation point for analytic bodies at angle of attack. McGowan and Davis (ref. 262) have utilized an orthogonal surface coordinate system for sharp right circular and elliptical cones at angle of attack. However, these applications are for analytic bodies of revolution and not for general aerodynamic vehicles. The body-oriented coordinate system appears optimum from the viewpoints of geometry input and direct coupling with existing three-dimensional inviscid computer codes. The system completely avoids the time-consuming and difficult problem of streamline trajectory calculation for each change in flow conditions; consequently, inviscid-viscous coupling procedures can be greatly simplified for problems where displacement surface effects are important. The difficulties associated with developing efficient procedures for numerically generating the coordinates and required metric coefficients for an orthogonal body-oriented surface coordinate system are more than compensated by program flexibility from the viewpoint of the engineer who must use the computer program as a design tool.

The body-oriented surface coordinate system can be chosen to be either orthogonal or nonorthogonal. Regardless of whether the system is orthogonal or nonorthogonal, the $x_3$-coordinate must be chosen normal to the wall boundary. Consequently, for the orthogonal system the selection of one coordinate $x_1$ or $x_2$ automatically fixes the remaining coordinate because of the orthogonality requirements. For example, if $x_1$ is chosen to lie along the rays of a sharp right circular cone, then the $x_2$-coordinate for $x_1 =$ Constant cuts the cone in circular elements; however, for sharp elliptical cones a warped cross-section element is formed. (See fig. 2 of ref. 262.) The nonorthogonal coordinate system appears desirable from a number of viewpoints: (1) It can be made to coincide with the location of the initial data planes; (2) it can be generated as easily as the orthogonal system; and (3) the difference grid can be developed to completely cover the computational region of interest - for example, swept,
tapered wings. (See fig. 2 of ref. 244.) Further, the nonorthogonal system results in only a minimal increase in the number of correction terms arising from the metric tensor (see ref. 245) provided the cross-flow diffusion terms are not considered (in other words, if only the classical three-dimensional boundary-layer equations are used).

From the viewpoint of the design engineer it appears that a general computer code for solving the three-dimensional boundary-layer equations should be developed in nonorthogonal, body-oriented surface coordinates. This system would also allow complete flexibility and could utilize streamline coordinates if so desired for special flow cases. This viewpoint is currently being incorporated in the computer program under development by Cebeci et al. (ref. 244) for wing surfaces.

Transformations.- The selection of suitable transformations for three-dimensional laminar or turbulent boundary-layer flow presents a problem since no one transformation can be considered general. The selection of primitive variables introduces problems since the boundary layer is thin and grows at different rates in the $x_1$- and $x_2$-coordinates; furthermore, the equations are singular at the tip of sharp bodies. When physical coordinates are used, the solution is extremely sensitive to the surface mesh point distribution; consequently, procedures developed in physical coordinates generally require either excessively small grid spacing distributions for the $x_1$- and $x_2$-coordinates or special treatment of the spatial derivative terms in the finite-difference equations if accurate results are to be obtained. This results directly in excessive computer storage and/or processing time. Specific problems associated with physical coordinates are as follows: (1) excessive growth of the boundary layer in the computational domain requiring additional mesh points in the $x_3$-coordinate as the solution proceeds downstream, (2) large gradients in the wall region for turbulent flows requiring either closely spaced mesh points in the normal direction or the inclusion of the law-of-the-wall relationship (not really satisfactory for three-dimensional flow), and (3) singularities such as at the tip or leading edge of sharp-edge bodies. The problem associated with the boundary-layer growth can be treated with suitable stretching and normalization as presented by Kendall et al. (ref. 245).

Transformation to similarity variables has been shown to be useful for computing similar flows (see ref. 263, for example); however, their usefulness is questioned by some authors (see, for example, ref. 245) for three-dimensional flows which are highly nonsimilar in character. The majority of the three-dimensional boundary-layer solution procedures now in existence use similarity type variables. (See ref. 245.)

As in two-dimensional boundary layers the physics of turbulent flow presents a numerical problem since wall gradients are large in comparison to laminar flow; the viscous sublayer generally requires a minimum of two to three mesh points. Generally a geometric-progression grid point spacing normal to the wall is employed; that is $\Delta x_{3,i+1} = \hat{R} \Delta x_{3,i}$, where $\hat{R} = 0(1.02)$. It has also been suggested that a logarithmic spacing be utilized as indicated by the law of the wall. The main object, regardless of the transformation or stretching selected for the normal coordinate, is to minimize the total number of mesh points normal to the wall boundary in order to minimize the required computer
storage and processing time. It should also be noted that the $x_3$-transformation generally complicates the Reynolds stress models and their inclusion in the computer code; this is one advantage of using primitive variables with suitable thickness normalization. (See ref. 245.)

To date no general transformation has been developed for three-dimensional turbulent boundary-layer flows. Several transformations have been studied and are presented by Blottner (ref. 242). Some of these transformations will be discussed in a subsequent section where three particular computer programs will be presented, with numerical results compared with experimental data.

Turbulence Modeling

Research on turbulent transport equation models and turbulence structure (e.g., refs. 30 (pp. D-1 - D-14 and B-1 - B-24) and 264) is very important since the goal of this research is to develop one set of equations with one set of constants or functional values which will adequately model all turbulent flows; however, to date no transport equation procedure has been shown to be superior to simple eddy-viscosity models for two-dimensional, equilibrium, boundary-layer flows. (This is not the case for free shear, jet, and wake flows.) It is agreed by most investigators of turbulence modeling that eddy-viscosity models leave a great deal to be desired since the physics of the flow is in general neglected; however, from the design engineers viewpoint, the eddy-viscosity concept works well for a wide class of two-dimensional turbulent boundary-layer flows.

A number of rather difficult questions must be answered concerning the extension of the two-dimensional eddy-viscosity models to three-dimensional flow; for example, the cross-flow momentum equation requires the specification of an eddy-viscosity coefficient for the cross-flow direction. The simplest approximation is to assume that $\varepsilon_{x2} = \varepsilon_{x1}$ (isotropic eddy viscosity). This simple assumption implies that the turbulent stresses and mean flow rate of strain are always parallel. However, measurements of the complete turbulent stress tensor by Van den Berg et al. (ref. 265) indicate that the turbulent shear stress level in three-dimensional boundary-layer flow is lower than would be expected from extrapolation of two-dimensional theory; also, the local shear stress direction does not in general coincide with the direction of the velocity gradient. Experimental data indicate that the eddy viscosity should be lower in the cross-flow direction; that is, $\varepsilon_{x2}/\varepsilon_{x1} < 1$. One might assume that $\varepsilon_{x2} = \alpha \varepsilon_{x1}$ where $\alpha < 1$; however, this approximation is unsatisfactory (see ref. 266) unless some empirical functional relationship between $\varepsilon_{x1}$ and $\varepsilon_{x2}$ can be determined which allows accurate numerical prediction of a wide class of three-dimensional boundary-layer flows. This approach may become feasible as accurate three-dimensional experimental data are obtained. Two-dimensional eddy-viscosity models were developed to their current level primarily as a result of the availability of a wealth of experimental data; the same data base must be provided for three-dimensional flows for successful models to be developed.

In the present paper, closure through the solution of transport equations will not be discussed in further detail. The reader interested in this approach is referred to references 172 and 267 to 269 for examples.
Closure of equations (61) to (63) requires that an effective viscosity \( \mu_{\text{eff}} \) and thermal conductivity \( \kappa_{\text{eff}} \) be expressed in terms of the mean flow variables. The Reynolds stress terms are defined as follows:

\[
-\rho \, u_1 u_3 = \varepsilon_{x_1} \frac{\partial u_1}{\partial x_3} \\
-\rho \, u_2 u_3 = \varepsilon_{x_2} \frac{\partial u_2}{\partial x_3} \\
-\rho \, u_3' H' = \varepsilon_0 \frac{\partial H}{\partial x_3}
\]  

(72)

The further assumption is made that \( \varepsilon_{x_1} = \varepsilon_{x_2} = \varepsilon \) and a turbulent Prandtl number is defined which relates the effective conductivity and effective viscosity. The following relations can then be formulated:

\[
\mu_{\text{eff}} = \mu \left( 1 + \frac{\varepsilon}{\mu} \Gamma \right)
\]

(73)

and

\[
\kappa_{\text{eff}} = \frac{c_p \mu}{N_{Pr}} \left( 1 + \frac{\varepsilon}{\mu} \frac{N_{Pr}}{N_{Pr,t}} \Gamma \right)
\]

(74)

The streamwise intermittency function \( \Gamma_x \) (ref. 218) models the transitional region of flow and is a function of \( x_1 \) and \( x_2 \) (0 \( \leq \Gamma_x \leq 1; \ \Gamma_x = 0 \) for laminar flow; \( \Gamma_x = 1 \) for turbulent flow). The effect of pressure gradient on intermittency is presented in reference 270. For the present discussion it is assumed that the initiation and completion of the transitional flow region are specified; however, correlation relations could be directly incorporated into a computer code.

The simplest eddy-viscosity approach (see fig. 54) is to assume that the eddy viscosity is a scalar function independent of coordinate direction (refs. 168 and 243). The following models are considered since they are used in most current three-dimensional, compressible, turbulent, boundary-layer programs:

**Single-layer model**

\[
\varepsilon = \rho l^2 \left[ \frac{1}{h^2} \left( \frac{\partial u_1}{\partial x_3} \right)^2 + \left( \frac{\partial u_2}{\partial x_3} \right)^2 \right]^{1/2} \Gamma_y
\]

(75)

where

\[
\frac{1}{x_3, e} = k_2 \tanh \left( \frac{k_1 x_3}{k_2 x_3, e} \right)
\]

(76)
\[ D = 1 - \exp\left(-\frac{x_3}{A}\right) \]  

\[ A = A^* \left(\frac{\mu}{\rho}\right) \left(\frac{\tau_t}{\rho}/w\right)^{-1/2} \]  

\[ 1 - \text{erf}\left(k_5 \frac{x_3}{x_3,e} - k_6\right) \]  

\[ \Gamma_y = \frac{2}{2} \]  

\[ \tau_t = \left(\frac{2}{\Gamma_x 1 + \Gamma_x 2}\right)^{1/2} \]  

Two-layer model

Inner law

\[ \varepsilon_{\text{inner}} = \rho l_2^2 \left\{\frac{1}{h_3^2} \left[\left(\frac{\partial u_1}{\partial x_3}\right)^2 + \left(\frac{\partial u_2}{\partial x_3}\right)^2\right]\right\}^{1/2} \Gamma_y \]  

(0 \leq x_3 \leq x_3,c) \]  

(81)

Outer law

\[ \varepsilon_{\text{outer}} = k_4 \rho V_e \delta^* \Gamma_y \]  

(x_3,c < x_3 \leq x_3,e) \]  

where

\[ l_2 = k x_3 D \]  

\[ V_e = \left(u_{1,e} + u_{2,e}\right)^{1/2} \]  

The point where the inner and outer laws are matched \( x_3,c \) is obtained from the continuity of eddy viscosity; that is, \( \varepsilon_{\text{inner}} = \varepsilon_{\text{outer}} \). (See sketch (h).) No attempt is generally made to assure continuous derivatives for the two laws; however, the single law is continuous and appears to be as satisfactory as the more complex two-layer relationship (ref. 271). Three-layer models have been proposed by Pletcher (ref. 158), but there does not appear to be any advantage to the multilayer models above the single-layer mixing length model (eq. (75)) or the two-layer model (eqs. (81) and (82)). The normal intermittency factor

\[ \delta^* = \int_{x_3,0}^{x_3,e} \left[ 1 - \frac{\left(u_{1}^2 + u_{2}^2\right)^{1/2}}{V_e}\right] dx_3 \]  

(85)
The sketch (h) proposed by Klebanoff (ref. 272) is generally approximated by the following relationship:

$$\frac{\bar{x}_3}{\bar{x}_3, e}$$

$$\frac{\bar{\varepsilon}}{\bar{\mu}}$$

Sketch (h)

(eq. (79)) proposed by Klebanoff (ref. 272) is generally approximated by the following relationship:

$$\Gamma_y = \left[ 1 + 5.5 \left( \frac{x_3}{x_3, e} \right)^6 \right]^{-1}$$ (86)

For high Reynolds number flows the constants $k_1$ appearing in equations (75) to (83) are generally assigned the following values: $k_1 = 0.4$ to 0.435, $k_2 = 0.09$, $A^+ = 26$, $k_3 = 0.0168$, $k_5 = 5$, $k_6 = 0.78$, and $N_{Pr,t} = 0.95$. These values represent the classical values generally used for equilibrium, high Reynolds number, turbulent flows (see ref. 72); however, it should be noted that although the assigned values are sufficient over a broad range of flow and wall boundary conditions, modifications to these values are required for certain classes of flow as discussed in previous sections of the present review. Typical turbulent Prandtl number variations are given in references 273 to 275 and in figure 32.

The accuracy of the various eddy-viscosity models can only be assessed by careful numerical experimentation in which numerical results are compared with experimental data. These models have been compared over a wide range of test conditions for two-dimensional high Reynolds number, turbulent boundary-layer flows; however, their extension to general three-dimensional flows still requires extensive study. The models (eqs. (75), (81), and (82)) have been shown to accurately predict the data of Rainbird (ref. 276) for a sharp right circular cone (adiabatic wall) at angle of attack. (See refs. 271 and 277.) However, more demanding three-dimensional flows should be calculated and the numerical results compared with data. The previously mentioned problem with $\varepsilon_{x2}$ must receive careful attention. It is not sufficient to simply assume $\varepsilon_{x2}$ is some percentage of $\varepsilon_{x1}$; this has been indicated in reference 265 for infinite swept-wing flow.

Numerical Solution Procedures

A number of research papers have been published over the past few years dealing with numerical procedures for solving the three-dimensional boundary-layer equations for laminar and turbulent compressible flow. Currently, there is no general method for solving the three-dimensional boundary-layer equations (ref. 235); however, significant progress has been made in the development of
such procedures by Kendall et al. (ref. 245), Cebeci et al. (ref. 244), and Wortman (ref. 162). A limited number of problems have been solved by applying special techniques to each specific problem (stagnation point flow, leading-edge attachment flow, swept infinite wings, and sharp and blunt cones). For these particular problems it has been possible to use simple geometrical relationships (coordinate system) and similarity type transformations to reduce the complexity of the governing equations. As previously mentioned, two of the more difficult numerical problems are (1) the generation of the surface coordinate systems and the required metric coefficients \( h_1, h_2, \) and \( h_3 \) and (2) the proper treatment of the inviscid edge conditions.

The properties of the three-dimensional boundary-layer equations were first studied in detail by Raetz (ref. 248). It was noted that the governing equations are hyperbolic in planes parallel to the wall boundary as opposed to parabolic for two-dimensional flow. Raetz introduced the concept of the influence principle into the three-dimensional boundary-layer literature which results directly from the hyperbolic character of the equations. The zones of influence and dependence for the three-dimensional boundary-layer equations have also been examined in detail by Wang (ref. 256). The hyperbolic character of the equations has been utilized by Bradshaw (ref. 278) through the application of the well-known method of characteristics, the wall region of the flow being patched to the characteristics solution for the outer region.

In order to obtain the correct solution to the three-dimensional equations, the solution procedure must correctly account for the zone of dependence. The stability of the procedure and the region where the solution can be obtained with specified initial conditions are determined by the zone of dependence. This zone thus prescribes the minimum amount of initial data that must be specified in order to advance the solution. Kitchens, Gerber, and Sedney (refs. 254 and 257) have made detailed studies of this requirement during which they systematically violated the zone of dependence requirement. Their results clearly demonstrate that errors accumulate and grow in the numerical solution if the zone of dependence is not satisfied. The most interesting result, presented in reference 254, is that satisfying the zone of dependence criteria is not sufficient to assure stability in all cases. The conclusion is drawn that the zone of dependence concept is not necessarily the same as the concept of stability. This particular result of reference 254 should be further studied since most investigators have treated these concepts as either the same or closely related.

For two-dimensional boundary-layer flows the zone of dependence is automatically satisfied, the equations are parabolic, and the procedure can march downstream, provided the necessary initial, boundary, and edge conditions are specified. For three-dimensional boundary-layer flows it is necessary to internally adjust the mesh aspect ratio (see ref. 254), such that

\[
\frac{\Delta x_2}{\Delta x_1} \geq \max \left( \frac{|u_e|}{u_1} \right) \quad \text{over all interior points}
\]

(87)

Then the largest allowable \( \Delta x_1 \) from one data plane to the next is
\[ \Delta x_1 < \tilde{\alpha} \frac{\Delta x_2}{\max(|u_2|/u_1) \text{ over entire data plane}} \]  

where \( \tilde{\alpha} \leq 1 \). The requirement that equations (87) and (88) be satisfied in the solution plane introduces numerical complexity for the three-dimensional boundary-layer equations.

The numerical procedures currently being applied to solve the three-dimensional boundary-layer equations have resulted, in general, from a direct extension of procedures developed during the 1960's for two-dimensional and axisymmetric boundary-layer flows. The present review is not intended to give a complete survey of the extension of these methods to three-dimensional boundary-layer flows. Reviews of three-dimensional laminar boundary layers have been presented by Cooke and Hall (ref. 279), Stewartson (ref. 280), Crabtree, Küchemann, and Sowerby (ref. 281), Mager (ref. 282), and Hansen (ref. 283). A recent review of two- and three-dimensional flows has been presented by Blottner (ref. 235). Solutions of the three-dimensional boundary-layer equations have been considered by Fannelöp and Humphreys (ref. 253). The reader interested in the initiation of numerical research in three-dimensional boundary-layer flows should carefully review these references, as well as The European Research Programme on Viscous Flows (ref. 266) and the results of Euromech 60 (ref. 284).

The purpose of the present section is to present three solution procedures which are representative of current programs for three-dimensional turbulent boundary-layer flows; two of these procedures are currently being developed as production codes for arbitrary configurations.

Crocco variables.-- The primary advantage of a Crocco-type transformation is that the solution domain in the normal coordinate is bounded between the definite limits of \( 0 \leq \xi \leq 1 \). The procedure is also attractive in that the grid points can be uniformly placed in the velocity plane and still satisfy the demanding mesh point distribution for the wall region of turbulent flows. The only disadvantage of the method appears to be the restraint that velocity overshoot in the \( u_1 \) velocity component is not allowed; that is, \( u_1/u_1,e \) must increase monotonically from the specified wall value (slip at surface can be allowed) to unity at the edge boundary. The transformation has been used for laminar flows (see refs. 255, 258, 262, and 285) and for turbulent flows over cones at angle of attack by Harris and Morris (ref. 168). A discussion of Crocco-type variables is presented by Davis (ref. 286).

A three-dimensional compressible turbulent code has been developed at Langley Research Center for Crocco-type variables (ref. 168). The program is currently restricted to analytic geometry; however, work is currently under way to extend the code to a general curvilinear coordinate system for arbitrary geometry.

In the subject program, equations (60) to (63) are first nondimensionalized, and a similarity-type transformation is introduced for the \( x_3 \)-coordinate and velocity as follows:

\[ \bar{h}_3 = \xi \frac{\rho e}{\rho} \frac{L}{\sqrt{R_\infty L}} h_3 \]  

where \( \rho_0 \) is the density at \( \xi = 0 \), \( R_\infty \) is the Reynolds number based on \( \rho_0, \rho_0, \) and \( L \) is the length scale. The program is currently restricted to analytic geometry; however, work is currently under way to extend the code to a general curvilinear coordinate system for arbitrary geometry.
\[ \bar{u}_3 = \frac{1}{\xi} \frac{\partial}{\partial \xi} \rho \xi u_3 \]  

(90)

where \( u_0 \) is the reference velocity and \( \xi = \sqrt{\chi_1} \) for a sharp cone. The metric coefficients \( h_2 \) and \( h_3 \) are arbitrary functions of the coordinates. The equations are next cast into Crocco form as follows:

\[ \zeta = \left(1 - \frac{u_1}{u_1,e}\right)^{1/2n} \]  

(91)

The exponent \( n \) can be selected to minimize the number of nodal points in the \( \zeta \)-direction (normal to the wall boundary). The continuity and \( x_1 \)-momentum equations are combined to form the shear equations, where the shear parameter \( \Phi \) is defined by

\[ \Phi = -\frac{1}{h_3} \frac{\mu}{T} \left(1 + \frac{\varepsilon}{\mu} \Gamma_x \right) \]  

(92)

Consequently, \( u_1/u_1,e \) is replaced by \( \Phi \) as a new dependent variable, and \( \hat{H} = u_2/u_1,e \) is uncoupled from the system of equations. The system therefore reduces to three coupled nonlinear partial differential equations in \( \Theta \), \( G \), and \( \Phi \), together with an explicit algebraic relationship for \( \hat{H} \). The system assumes the following form:

\[ \frac{\partial^2 \omega}{\partial \zeta^2} + \alpha_1 \frac{\partial \omega}{\partial \zeta} + \alpha_2 \omega + \alpha_3 + \alpha_4 \frac{\partial \omega}{\partial \eta} + \alpha_5 \frac{\partial \omega}{\partial \eta} = 0 \]  

(93)

where \( \omega \) represents \( \Theta \), \( G \), and \( \Phi \), and \( \alpha_1 \), \( \alpha_2 \), \( \alpha_3 \), \( \alpha_4 \), and \( \alpha_5 \) are nonlinear coefficients.

The boundary conditions on equation (93) are as follows:

When \( \zeta = 0 \)

\[ G = 1 \quad \Theta = 1 \quad \Phi = 0 \]  

(94)

When \( \zeta = 1 \)

\[ \Theta = \Theta_w \quad \text{or} \quad \left(\frac{\partial \Theta}{\partial \zeta}\right)_w = f(\xi, \eta) \]

\[ G = 0 \]

\[ \left(\frac{\partial \Phi}{\partial \zeta}\right)_w = \left(-a_1 \hat{H} + a_2 \frac{\partial \Theta}{\partial \Phi} + \Phi\right)_w \]  

(95)

where \( a_1 \) and \( a_2 \) are functions of geometry and the inviscid edge conditions and \( \hat{T} = \mu/T \).

Equation (93) is solved in an iterative mode with a marching implicit finite-difference technique suggested by Dwyer (ref. 287) and modified by Krause.
The method is second-order accurate and unconditionally stable (conditional stability for reverse cross flow; see ref. 288). For turbulent flows a minimum of two mesh points in the \( \zeta \)-plane must be located in the viscous sublayer; consequently, careful consideration must be given to the placement of the mesh points in the \( \zeta \)-coordinate in order to minimize computer storage and processing time requirements. Two approaches to the minimization problem have been considered: the specification of a variable mesh point distribution for \( n = 1 \) of the form \( \Delta \zeta_{k+1}/\Delta \zeta_k = \text{Constant} \) for \( k = 2, 3, \ldots, N - 1 \) (geometric progression), and the selection of \( n = 3/4 \) which appears optimum for turbulent flows. Variable mesh-point distributions are also used in the \( \xi \)- and \( \eta \)-planes to minimize the computer processing time and storage requirements. A schematic of the difference molecule is presented in figure 55(a). Equation (93) is written at the point \((i-1/2,j,k)\) and solved for the values of the dependent variables \( \Theta, G, \Phi, \) and \( H \) at the point \((i,j,k)\). The difference relationships used in the procedure for the Krause (ref. 288) "zig-zag" scheme are presented in reference 271.

When a converged solution cannot be obtained at the most leeward plane, \( \phi = 180^\circ \) (for example, separation on leeward surface), a cubic Crank-Nicolson differencing scheme (see fig. 55(b)) is used at the maximum \( \eta \)-station (ref. 261). If this procedure were not incorporated into the program logic, one \( \eta \)-station would be lost for each additional \( \Delta \zeta \) step because of the Krause differencing scheme.

The marching procedure cannot be initiated without the existence of two orthogonal initial data planes. For a sharp right circular cone these planes of initial data are generated directly from the governing equations by using a second-order Crank-Nicolson scheme for the two planes \((\xi = 0, 0 \leq \eta \leq \eta_{\text{max}}\) and \(0 \leq \xi \leq \xi_{\text{max}}\), \( \eta = 0 \)) where similarity exists. A discussion of problems associated with obtaining initial data planes for general configurations is presented in reference 260.

Substitution of the difference quotients (see ref. 271) into equation (93) results in a system of coupled algebraic equations whose coefficient matrix is of tridiagonal form which can be efficiently solved for the dependent variables (Thomas' algorithm). The primary problem associated with equation (93) is that the coefficients \((\alpha_1, \alpha_2, \text{etc.})\) are highly nonlinear. The shear equation controls the convergence rate of the numerical procedure (the number of iterations required as the system is sequentially iterated). Equation (93) can be written for \( \phi \) as

\[
\frac{\partial^2 \Phi}{\partial \xi^2} + \left( \frac{h_1}{\xi} + \frac{\beta_1}{\phi^2} \right) \frac{\partial \Phi}{\partial \xi} - \frac{h_1}{\phi} \Phi + \frac{\beta_2}{\phi^2} \frac{\partial^2 \Phi}{\partial \xi^2} + \frac{\beta_3}{\phi^2} \frac{\partial \Phi}{\partial \eta} + \frac{\beta_4}{\phi^2} \frac{\partial \Phi}{\partial \eta} = 0
\]  

(96)

where the coefficients \( \beta_1, \beta_2, \beta_3, \) and \( \beta_4 \) are functions of geometry, inviscid edge conditions, and previous iterate values of the dependent variables \( u_1/u_{1,e} \) and \( \Theta \) and their derivatives. The problem is further complicated by the inclusion of the turbulence models (eqs. (75) to (85)) since in the transformed plane \( \Phi \) appears explicitly in the transformed relationships. Consequently, the coefficients \((\beta_1, \beta_2, \text{etc.})\) also depend on \( \Phi \) for turbulent flows (for laminar flows this dependence is removed). The system of equations will not converge if the shear equation is written as shown in equation (96) because
of the $\Phi^{-1}$ term. Convergence can be achieved by using a Taylor's series expansion of $\Phi^{-1}$ about the previous iterate value $\Phi_G$; that is,

$$\frac{1}{\Phi} = \frac{1}{\Phi_G} (2 - \Phi + O(\Phi^2)) \tag{97}$$

Substitution of equation (97) into equation (96) yields

$$\frac{2}{\Phi^2} + (\frac{h_1 + b_1}{\Phi} + \frac{\alpha}{\Phi^2}) \Phi + \left( -\frac{h_1 - b_2}{\Phi^2} \right) \Phi + \frac{2b_2}{\Phi_G} + \frac{b_3}{\Phi} + \frac{b_4}{\Phi^2} \Phi = 0 \tag{98}$$

Equation (98) converges in an average of five to seven iterations for high Reynolds number turbulent flow. The wall boundary condition on $\Phi$ (see eq. (95)) also presents a problem since $\Phi_W$ is unknown; however, the wall derivative relationship can be directly incorporated in the iterative solution procedure. In principle, it should be possible to reduce the average number of iterations substantially to a maximum of three. Research continues in the areas of (1) restructuring equation (98), (2) treatment of the $\Phi_W$ wall boundary conditions, and (3) the problems associated with $\Phi$ in the transformed turbulence models. Note however that the procedure requires essentially the same processing time per mesh point (0.002 sec) as the Cebeci-Keller box method (refs. 289 to 291) and that this time may be substantially reduced through convergence accelerator procedures and/or the inclusion of Newton-Raphson iteration.

The numerical procedure and turbulence models have been applied to a number of flows (current geometry limited to sharp right circular and elliptic cones). Numerical results compared with experimental wall and profile data for a cone (ref. 276) with a 12.5° semiapex angle at an angle of attack of 15.75° are shown in figure 56. The free-stream Mach number, total pressure, and total temperature for the experiments were 1.8, 172.4 kN/m², and 294 K, respectively. Transition was assumed to be initiated and completed in the region $0.03 \leq \xi_1/L \leq 0.08$ ($L = 105.6$ cm). The adiabatic wall boundary condition was imposed on the energy equation (see eq. (95)); that is, $(\partial \Theta/\partial \zeta)_w = 0$. No experimental data were input into the viscous flow solution. The inviscid pressure distribution $P_e = f(\zeta, n)$ was obtained from a numerical solution of the three-dimensional inviscid flow equations.

The numerical results for $u_1/u_1, e, G$, and $\Theta$ are compared with experimental data in figure 56 for circumferential locations of $\phi$, $0^\circ$ and $135^\circ$. In order to evaluate the effect of nodal-point spacing in the $\zeta$-plane, a parametric study was made for $N = 301, 201, 101, 61$, and $21$ with $\Delta \zeta_{k+1}/\Delta \zeta_k = 1.02$. The results for $N = 301$ and $201$ were essentially identical, and those for $N = 101$ were within 0.5 percent of the $N = 301$ results. The agreement between the experimental and numerical results is very good for 301 points and, in general, good for as few as 21 points. The two turbulence models (eqs. (75) to (85)) produced essentially identical results. The two-layer model results presented in figure 56 are for $N = 301$; however, the two-layer results for $N = 21$ were essentially identical to the $N = 21$ results of the single-layer model.

The results for $\Delta \zeta_{k+1}/\Delta \zeta_k = 1.02$ presented in figure 56 were obtained for $n = 1$. (See eq. (91).) Numerical results for $\Delta \zeta_{k+1}/\Delta \zeta_k = 1$ (equal mesh distribution), $N = 21$, and $n = 3/4$ were in slightly better agreement with the
N = 301 results for the variable mesh point distribution; however, the difference between the two cases for N = 21 was minor. The major points that should be noted in these comparisons are that the numerical procedure is efficient and accurate and that the turbulence models are satisfactory for high Reynolds number equilibrium turbulent boundary-layer flows. The Crocco-type transformation and the numerical procedure allow the generation of accurate solutions for a minimum of 21 points normal to the wall boundary. The computer code requires 600008 storage (the \( i-1,j,k \) data plane is stored on disk) and approximately 0.002 second per grid point processing time on a CDC-6600 computer system. Current studies indicate that it may be possible to substantially reduce the processing time through convergence accelerators for the shear equation (eq. (98)) and/or the inclusion of a Newton-Raphson iteration procedure. The current program is comparable in both storage and processing time with the Cebeci-Keller box method (ref. 260).

Arbitrary wings.- An accurate and efficient computer code for the three-dimensional boundary-layer flow over wings is required for the design and evaluation of supercritical wings and laminar flow control surfaces \( u_3, w = g(x_1, x_2) \). Studies of wing geometry specifications have indicated that a nonorthogonal surface coordinate system is optimum from the viewpoint of the design engineer. Cebeci et al. (ref. 260) have developed an efficient and accurate procedure for solving the three-dimensional boundary-layer equations for laminar, transitional, and turbulent perfect-gas flow over general wings. The advantage of this geometry routine are as follows: (1) Calculation of the coordinate system \( (\xi, \eta, \xi_1, \ldots) \) for each angle-of-attack or flow condition change is eliminated; (2) discontinuities associated with patched coordinate systems are eliminated; (3) the method is optimum from the viewpoint of the design engineer. (See sketch (i).) Only one disadvantage is encountered in that the nonorthogonal metric tensor results in additional terms appearing in the system of equations. However, this increase in terms is minimal if cross-flow diffusion terms are neglected. Preliminary results obtained in reference 260 for a typical supercritical wing indicate computation times of approximately 30 sec on an IBM 370/165 computer system for a \( 30 \times 20 \times 20 \) grid.

The governing equations are written for a nonorthogonal surface coordinate system for ease of placement of the mesh distribution over the wing surface. The wing is defined in an orthogonal \( \bar{x}_1, \bar{x}_2, \bar{x}_3 \) coordinate system where \( \bar{x}_1 \) is in the direction of the airplane's longitudinal axis, \( \bar{x}_2 \) is in the spanwise direction, and \( \bar{x}_3 \) is orthogonal to the plane of \( \bar{x}_1 \) and \( \bar{x}_2 \). The wing is described
by a series of airfoil sections in planes of $x_2 = \text{Constant}$. The equations relating the airfoil section specification and the $x_1$-coordinate system are presented in reference 260 in terms of percentage chord. The equations required for the calculation of the nonorthogonal surface mesh and the procedure with which they are solved are also presented in reference 260.

The $x_1$-momentum equation is as follows (see eq. (61) for orthogonal system):

$$
\frac{\rho u_1}{h_1} \frac{\partial u_1}{\partial x_1} + \frac{\rho u_2}{h_2} \frac{\partial u_1}{\partial x_2} + \frac{\rho u_3}{h_3} \frac{\partial u_1}{\partial x_3} - \rho K_1 u_1 u_2 \cot \theta + \rho K_2 u_2^2 \csc \theta + \rho K_{12} u_1 u_2
$$

$$
= - \frac{\csc^2 \theta}{h_1} \frac{\partial p}{\partial x_1} + \frac{\cot \theta \csc \theta}{h_2} \frac{\partial p}{\partial x_2} + \frac{\partial}{\partial x_3} \left( \mu \frac{\partial u_1}{\partial x_3} - \rho u_1 u_2 \right)
$$

(99)

where $\theta$ represents the angle between the coordinate lines $x_1 = \text{Constant}$ and $x_2 = \text{Constant}$. For an orthogonal coordinate system, $\theta = \pi/2$ and equation (99) reduces identically to equation (61) for $h_3 = 1$. The geodesic curvatures are given by the relationships

$$
K_1 = \frac{1}{h_1 h_2 \sin \theta} \left[ \frac{\partial}{\partial x_1} (h_2 \cos \theta) - \frac{\partial h_1}{\partial x_2} \right]
$$

(100)

$$
K_2 = \frac{1}{h_1 h_2 \sin \theta} \left[ \frac{\partial}{\partial x_2} (h_1 \cos \theta) - \frac{\partial h_2}{\partial x_1} \right]
$$

(101)

The parameter $K_{12}$ is defined as follows:

$$
K_{12} = \frac{1}{\sin \theta} \left[ - (K_1 + \frac{1}{h_1} \frac{\partial \theta}{\partial x_1}) + \cos \theta (K_2 + \frac{1}{h_2} \frac{\partial \theta}{\partial x_2}) \right]
$$

(102)

It should be noted that the addition of correction terms for nonorthogonal coordinates to the system of equations is minor for the classical three-dimensional boundary-layer equations; however, if one wants to include the cross-flow diffusion terms in the $x_2$-momentum equation, then a significant number of additional correction terms would be required. (See ref. 12.)

The system of governing equations is transformed through the introduction of a two-component vector potential and a similarity-type transformation. (See ref. 260.) The Cebeci-Keller box method (refs. 162, 289, and 290) is used to solve the resulting system of equations. One of the basic concepts of the box procedure is to rewrite the system of equations as a first-order system of partial-differential equations. Consequently, derivatives of some quantities with respect to the $x_3$-coordinate must be introduced as new unknown variables. Derivatives with respect to all other variables occur only to the first order because of the boundary-layer approximations. Centered difference quotients and averages at the midpoints of net rectangular and net segments are used in order to produce second-order-accurate finite-difference equations. The method is unconditionally stable for positive cross flow but appears to be unconditionally unstable for negative cross flow. The equations are highly nonlinear and implicit in structure. Newton's method is used to solve the system; a block-tridiagonal factorization scheme is used which is efficient and stable. The
Numerical formulation of the system of equations is presented in detail in reference 28. Numerical results are compared with the data of East and Hoxy (ref. 291) in figure 57 for several $x_L$ stations. Numerical results for a supercritical wing calculation are presented in figure 58.

Reactive gas flows. - The Aerotherm Division of the Acarex Corporation is currently developing a laminar, transitional, and turbulent three-dimensional boundary-layer computer code for application to arbitrary configurations such as the space shuttle. (See ref. 245.) The numerical procedure is an extension of the method presented in reference 238. This particular code includes equilibrium and/or frozen flow chemistry. A body-oriented orthogonal surface coordinate system is used to describe the vehicle surface. The computer code is compatible with the accurate three-dimensional supersonic inviscid flow field program developed by Marconi, Yaeger, and Hamilton (ref. 246). Three-dimensional entropy-layer effects are included in the solutions. This is of particular importance for vehicles such as the space shuttle.

The governing equations are numerically solved in primitive variables with suitable stretching and normalization in the $x_3$-coordinate. Similarity transformations are not used since the program will in general be applied to highly nonsimilar flows. The governing equations, in nondimensional form, are presented in reference 245. The solution domain is covered by a mesh point network shown in figure 59. The initial data planes are assumed known at $x = x_1$; the solution is to be obtained at $x_{i+1} = x_i + \Delta x_i$. The functional form of the derivatives is specified and substituted into the governing equations together with the required boundary conditions. This procedure results in a system of algebraic relations between the unknown dependent variables at the nodal point in the plane located at $x_{i+1}$. Normal to the wall boundary, the dependent variables are represented by a splined Taylor series between each mesh point as follows (where the prime denotes differentiation with respect to $x_3$):

$$
\begin{align*}
&u_{j+1} = u_j + u'_j \Delta x_3 + u''_j \frac{\Delta x_3^2}{2} \\
&u'_{j+1} = u'_j + u''_j \Delta x_3
\end{align*}
$$

(103)

Where $j$ represents the grid index ($j = 1, 2, \ldots, JMAX$). The equations are integrated with respect to $x_3$ between each mesh point. This insures that the conservation laws are satisfied exactly between the mesh points and greatly simplifies the calculation of the diffusion terms since all second derivatives are eliminated.

In the cross-flow direction ($x_2$-coordinate) second-order-accurate centered difference quotients are used to replace the cross-flow derivatives; that is, for equal nodal spacing

$$
\left( \frac{\partial u}{\partial x_2} \right)_{i,j,k} = \frac{u_{i,j,k+1} - u_{i,j,k-1}}{\Delta x_{2,k+1}}
$$

(104)

where $k$ is the index for the $x_2$ mesh point distribution. Axial derivatives are approximated by the backward difference quotients of the form
\[
\frac{\partial u}{\partial x_1} = \frac{u_{i+1,j,k} - u_{i,j,k}}{\Delta x_1,i}
\]  

(105)

where \( i \) is the index associated with the mesh point distribution in the \( x_1 \)-coordinate.

A fully implicit solution is obtained in the procedure. Equations (103) and (104) are evaluated in the \( x_1,i+1 \)-plane; introduction of the boundary conditions results in a nonlinear system of algebraic equations for the primary variables (\( u, p, \) etc.) and secondary variables (\( u', p', \) etc.) at each nodal point in the \( x_1,i+1 \)-plane. The resulting system of algebraic equations is solved by the Newton-Raphson procedure. The solution reduces to the problem of inverting a large matrix and thereby solving the linear system of algebraic equations for the unknowns. The large matrix is treated as a system of submatrices. The matrix is block tridiagonal in structure since cross-flow derivatives are represented by centered difference quotients. The submatrices along the main diagonal are dense, whereas the off-diagonal submatrices are sparse. A schematic of a typical diagonal submatrix for one \( x_1 \)-plane is presented in figure 4 of reference 245.

Data required for the initial planes are obtained directly in the solution procedures at stagnation points and leading-edge attachment lines. Similarity variables are used for the generation of these solutions. The entropy layer effects and the procedure through which they are included in the numerical procedure are discussed in detail by Kendall et al. (ref. 245). The eddy-viscosity model currently used is the single-layer model (see eq. (75)), where the Lewis and Prandtl numbers are assumed constant. More general models will be incorporated into the code once it is completely operational and detailed studies have been made of comparisons of numerical results with experimental data.

A comparison of numerical results with experimental data (ref. 292) for flow over a sharp cone at angle of attack is presented in figure 60. Solutions for the flow over a space shuttle are currently being obtained.

Status of Three-Dimensional Boundary-Layer Computational Techniques

Significant progress has been made in the development of numerical techniques for solving the compressible three-dimensional turbulent boundary-layer equations (see appendix, table A2); however, more efficient and reliable methods must be developed for application to general aerospace vehicles. This becomes clearly apparent when one considers the prohibitive computer processing times which can occur for either complex reacting gas flows or turbulent flows over large-scale aerospace vehicles (ref. 260). Early numerical research in the area of compressible, three-dimensional boundary-layer flows was primarily directed towards developing second-order-accurate, stable numerical schemes for solving the laminar equations for flows where simplifying assumptions were possible. Current research is directed towards (1) improving these numerical procedures for application to complex flows, (2) developing more general coordinate systems, and (3) detailed studies of turbulence modeling procedures for three-dimensional turbulent-boundary-layer flows. Reynolds stress models require accurate experimental data where the stress components are measured. The wealth of
two-dimensional experimental data which made possible the successful development of two-dimensional eddy-viscosity models may provide little reliable guidance for complex three-dimensional boundary-layer flows.

The problems associated with solving the three-dimensional inviscid equations for complex vehicles are rapidly being solved; however, efficient and reliable procedures for automatically coupling the inviscid and boundary-layer codes must be developed in order to avoid time-consuming and difficult data manipulation. Compatible coordinate systems must be developed for the inviscid and viscous codes in order to reduce or completely eliminate the inherent errors associated with data interpolation and smoothing between coordinate systems. Current studies and experience indicate that a nonorthogonal curvilinear coordinate system is optimum for general three-dimensional boundary-layer flows.

Flexibility must also be programmed into the boundary-layer codes which will assure that the optimum difference scheme is utilized at various locations in the solution domain in order to automatically satisfy the zone of dependence requirements as well as to maximize the region over which the solution may be obtained. Care must be exercised in order to determine whether the classical three-dimensional boundary-layer equations are valid in a specific area where the solution is required; that is, boundary region flow, inflow regions, and regions of separated flow require special treatment. For these cases either the parabolized Navier-Stokes or the full Navier-Stokes equations must be solved.

A number of difficult problems remain to be solved before compressible, three-dimensional, turbulent, boundary-layer codes are developed to the current confidence level of existing two-dimensional codes. An optimistic review of current research programs, together with the increasing capabilities of digital computer systems and the maturing of three-dimensional inviscid flow field codes, indicates that general purpose, three-dimensional boundary-layer codes for compressible turbulent flows will become available in the near future.

RÉSUMÉ

Compressibility Influence on Turbulent Boundary-Layer Shear Stress

From comparisons of high-speed data with low-speed closure procedures using variable mean density, there does not appear to be any appreciable influence of compressibility upon turbulent shear stress modeling in compressible turbulent boundary layers, even for extreme cases such as Mach 14 to 20 with a change in density across the layer of up to a factor of 100. Other evidence of an apparent lack of any compressibility-caused new physics which may alter the shear stress for the compressible boundary-layer case includes the following: (1) Fluctuation Mach number is generally less than 1; (2) the shear stress distribution through the boundary layer is not a function of Mach number for zero pressure gradient flows; (3) the Morkovin hypothesis is valid up to Mach 5 (based on fluctuation data); (4) profile $N$ power is not a function of Mach number, at least up to Mach 10; (5) the nondimensional burst period is approximately the same as that for low speed.
Computational Capability of Existing Procedures

Compressible equilibrium and near equilibrium boundary-layer flows, at least up to Mach 20, can be computed fairly accurately using mean field methods, provided that the influence of such items as low Reynolds number effects, pressure gradient, and wall blowing are properly accounted for through adjustments in the modeling constants. By and large, these adjustments can be obtained from low-speed data. Several calculation methods are available for the nonequilibrium case, but these have not yet received sufficient exercise on compressible highly nonequilibrium flows (primarily as a result of lack of data) to determine their relative or absolute accuracy.

Important Unresolved Issues

The following is a brief listing of what are the more important unresolved issues (or research frontiers) in the calculation of compressible turbulent boundary layers, as discussed in the present paper:

1. Considerable further development and calibration of nonequilibrium methods is required for compressible flows. Further experimental data in highly nonequilibrium flows must be obtained before this effort can be really meaningful.

2. Definitive experiments are needed to determine directly the influence of compressibility upon shear stress production in compressible turbulent boundary layers (conditional sampling measurements at high Mach numbers and measurements of fluctuating pressure terms in Reynolds stress equations).

3. Further definition is needed of the boundaries between boundary-layer and nonboundary-layer behavior; that is, where can boundary-layer methods be reasonably expected to work? Bradshaw addresses this with his "extra rates of strain," but definitive guidelines are needed (such as how much concave longitudinal curvature is necessary before longitudinal vortices develop and the flow is no longer a boundary layer). Obviously one can employ Navier-Stokes codes and complete Reynolds stress modeling to attack such situations, but currently the boundary-layer codes are so much faster to run on computers (as compared with the Navier-Stokes solvers) that they should be used whenever possible; this necessitates having well-known limits within which accurate answers can be obtained (without new physics making either the basic equation set or the turbulent modeling inapplicable).

4. Obviously, three-dimensional compressible turbulent-boundary-layer calculations are in an early stage, and considerable experimentation is necessary before the efficacy of mean field methods for the three-dimensional case is determined.

Langley Research Center
National Aeronautics and Space Administration
Hampton, VA 23665
April 7, 1977
APPENDIX

SUMMARY OF CALCULATION PROCEDURES FOR NONSIMILAR TWO- AND
THREE-DIMENSIONAL COMPRESSIBLE TURBULENT BOUNDARY
LAYERS (FINITE DIFFERENCE, FINITE ELEMENT, AND
WEIGHTED RESIDUAL METHODS)

The purpose of this summary (or catalog) of numerical prediction methods is
to indicate the wide variety of such procedures which are available. (Table A1
has 31 entries.) In most cases information for the various entries were
obtained from the individual authors, who kindly filled out and returned data
sheets on their methods. Several authors have made a special effort to optimize
the numerical solution procedure (e.g., refs. 9, 12, 162, and 163) to reduce
the required machine time and storage per case. Most of the procedures have
detailed user's manuals, and in many cases the codes are available either with-
out charge or for the cost of mailing. Many of the special effects treated by
the various methods (such as nonequilibrium or equilibrium chemistry, transi-
tion, roughness, etc.) are indicated in tables A1 and A2, as is the fundamental
closure approach.

If these tables serve no other function, they should at least cause
researchers to think quite carefully before producing the 32nd entry; that is,
several of the procedures are quite similar, and future research should obvi-
ously concentrate on using the best numerical methods to investigate such items
as nonequilibrium flows, rather than on producing another compressible turbulent-
boundary-layer deck.
### TABLE A1 - CALCULATION PROCEDURES FOR NONSIMILAR TWO-DIMENSIONAL COMPRESSIBLE TURBULENT BOUNDARY LAYERS

<table>
<thead>
<tr>
<th>Method</th>
<th>Major reference</th>
<th>User's manual</th>
<th>Numerical solution method</th>
<th>Closure assumption</th>
<th>Unique features</th>
<th>Computing machine; language; storage</th>
<th>Code source</th>
</tr>
</thead>
<tbody>
<tr>
<td>Adams</td>
<td>154, 293</td>
<td>Not available</td>
<td>Iterative implicit finite difference</td>
<td>Mixing length; Np,t = Constant; wall damping uses local properties</td>
<td>Originally developed for transitional flows</td>
<td>IBM 370/165; FORTRAN IV; 47000q core storage</td>
<td>J. C. Adams, Jr. Von Kármán Gas Dynamics Facility, AEDC Arnold Air Force Station TN 37389</td>
</tr>
<tr>
<td>Albers and Gregg</td>
<td>160, 40</td>
<td>Ref. 160</td>
<td>Shooting method</td>
<td>Herring and Moller inner and outer ε formulation (ref. 40)</td>
<td>Longitudinal wall curvature effects</td>
<td>IBM 7094/7044; FORTRAN IV; 67000q core storage</td>
<td>COSMIC 112 Barrow Hall University of Georgia Athens, GA 30602 (request LEB-12178; can also be punched from ref. 160)</td>
</tr>
<tr>
<td>Anderson and Lewis</td>
<td>294</td>
<td>Ref. 146</td>
<td>Implicit finite difference</td>
<td>Mixing length; Cebeci wall damping modifications (ref. 12); Clusser d in outer region</td>
<td>Both perfect gas and equilibrium chemistry</td>
<td>IBM and CDC; FORTRAN IV</td>
<td>C. H. Lewis Aerospace Engineering Dept., VPI &amp; SU Blacksburg, VA 24061 (can also be punched from ref. 146)</td>
</tr>
<tr>
<td>A. J. Baker</td>
<td>166</td>
<td>Ref. 295, 296</td>
<td>Finite element</td>
<td>Either eddy viscosity or second-order turbulent field closure</td>
<td>Small number of nodal points (due to finite element approach); wall shear specifiable; ideal gas and equilibrium hydrogen-air chemistry</td>
<td>IBM 360/65 or CDC 6600; FORTRAN IV; 165000q core storage</td>
<td>R. E. Smith, Jr. M.S.125, NASA Langley Research Center Hampton, VA 23665</td>
</tr>
<tr>
<td>R. J. Baker and Launder</td>
<td>186, 203</td>
<td>Ref. 9</td>
<td>Finite difference (Imperial College micro-integral method)</td>
<td>Mixing length with universal wall damping function; low Reynolds number amplification on outer mixing length</td>
<td>Unique near wall damping function</td>
<td>IBM or CDC; FORTRAN; 47000q core storage</td>
<td>Mechanical Engineering Dept. Imperial College of Science and Technology Exhibition Rd., London SW7, England (price, ~$100)</td>
</tr>
<tr>
<td>Method</td>
<td>Major reference</td>
<td>User's manual</td>
<td>Numerical solution method</td>
<td>Closure assumption</td>
<td>Unique features</td>
<td>Computing machine; language; storage</td>
<td>Code source</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>-----------------</td>
<td>---------------</td>
<td>---------------------------</td>
<td>------------------------------------------------------------------------------------</td>
<td>---------------------------------------------------------------------------------</td>
<td>-----------------------------------------------------------------------------------------------</td>
<td>------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Bradshaw and Ferriss</td>
<td>36, 297</td>
<td>Ref. 298</td>
<td>Method of characteristics</td>
<td>Mean turbulence field; uses relationship between $u'^{2}$ and $e$ ($\varepsilon$ parameter)</td>
<td>Includes models for extra rates of strain (Ref. 172); companion code will handle infinite swept wings with non-equilibrium turbulent field</td>
<td>CDC 6400; FORTRAN IV; less than 50000g core storage</td>
<td>P. Bradshaw, Aeronautics Dept. Imperial College of Science and Technology Exhibition Rd. London SW7, England</td>
</tr>
<tr>
<td>Bushnell and Beckwith</td>
<td>53, 167, 243, 299</td>
<td>Ref. 241, 299</td>
<td>Iterative implicit finite difference</td>
<td>Mixing length; latest versions include low Reynolds number amplification for outer region mixing length</td>
<td>Alternate version for slot injection; also, code available in nondimensional physical variables with $p(x,y)$ and longitudinal curvature terms; original version includes infinite swept wings</td>
<td>CDC 6600; FORTRAN IV; 75000g core storage</td>
<td>Barbara H. Pitts M.S. 163, NASA Langley Research Center Hampton, VA 23665</td>
</tr>
<tr>
<td>Cebeci and Keller</td>
<td>12, 290</td>
<td>Not available</td>
<td>Keller box method</td>
<td>Mixing length near wall; Cluaser in outer region (two-layer model); wall damping includes influence of $\hat{\varepsilon}$ and $d\eta/dx$</td>
<td>Uses very efficient numerical scheme; run times are quite short</td>
<td>IBM 370; FORTRAN IV; 77000g core storage</td>
<td>Generally available for company use only; limited outside distribution</td>
</tr>
<tr>
<td>Method</td>
<td>Major reference</td>
<td>User's manual</td>
<td>Numerical solution method</td>
<td>Closure assumption</td>
<td>Unique features</td>
<td>Computing machine; language; storage</td>
<td>Code source</td>
</tr>
<tr>
<td>-----------------</td>
<td>-----------------</td>
<td>---------------</td>
<td>---------------------------</td>
<td>----------------------------------------------------------------------------------</td>
<td>--------------------------------------------------------------------------------</td>
<td>-------------------------------------</td>
<td>-----------------------------------------------</td>
</tr>
<tr>
<td>Chan</td>
<td>155, 300</td>
<td>Not available</td>
<td>Implicit finite difference</td>
<td>Essentially a version of Bradshaw closure using integral form of turbulent kinetic energy equation; this equation tracks outer region mixing length constant</td>
<td>Relatively simple (but accurate) method for nonequilibrium turbulent flows</td>
<td>IBM 360/67; FORTRAN IV</td>
<td>Y. Y. Chan High-Speed Aerodynamic Section National Aeronautical Establishment Ottawa, Canada</td>
</tr>
<tr>
<td>Crawford and Kays</td>
<td>301</td>
<td>Ref. 301</td>
<td>Patankar-Spalding (ref. 9)</td>
<td>Turbulent kinetic energy equation or mixing length/eddy viscosity; mixing length always used in sublayer</td>
<td>Wall damping is a function of wall injection and pressure gradient; both internal and external flows</td>
<td>Can be punched from ref. 301</td>
<td></td>
</tr>
<tr>
<td>Deboy and Abbott</td>
<td>165</td>
<td>Included in ref. 165</td>
<td>Method of weighted residuals</td>
<td>Nominal Cebeci-Smith two-layer eddy viscosity (ref. 12)</td>
<td>Code used for studying numerical problems (with MWR) for turbulent boundary-layer flows</td>
<td>Code can be punched from ref. 165; procedure is not meant as a general-use prediction method</td>
<td></td>
</tr>
<tr>
<td>Donaldson and Sullivan</td>
<td>156</td>
<td>Ref. 302</td>
<td>Implicit finite difference</td>
<td>Complete mean turbulence field approach</td>
<td>Includes equations for all pertinent second-order correlations</td>
<td>CDC 7600 or IBM 1130; FORTRAN</td>
<td>M. W. Rubesin M.S. 229-3 NASA Ames Research Center Moffett Field, CA 94035 (code is still considered a research tool for studying second-order correlations, rather than a general predictive method)</td>
</tr>
</tbody>
</table>
## APPENDIX

<table>
<thead>
<tr>
<th>Method</th>
<th>Numerical solution method</th>
<th>Closure assumption</th>
<th>Unique features</th>
<th>Computing machine; storage</th>
<th>Code source</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dwyer</td>
<td>Implicit finite difference</td>
<td>Nominal Cebeci-Smith two-layer viscosity (ref. 12)</td>
<td>Includes transverse curvature, equilibrium</td>
<td>CDC 6600; FORTRAN IV; 6200B core storage</td>
<td>Not available</td>
</tr>
<tr>
<td>Fivel and Masek</td>
<td>Implicit finite difference</td>
<td>Nominal Cebeci-Smith two-layer viscosity (ref. 12)</td>
<td>Nonequilibrium, and frozen, chemical, catalytic</td>
<td>Could be punched from ref. 232 (uses basic Patankar-Spalding code)</td>
<td>Julius E. Harris, M.S., Langley Research Center, Hampton, VA 23665</td>
</tr>
<tr>
<td>Carr and Fox</td>
<td>Implicit finite difference</td>
<td>Nominal Cebeci-Smith two-layer viscosity (ref. 12)</td>
<td>Uses physical coordinates</td>
<td>CDC 6600; FORTRAN IV; 6200B core storage</td>
<td>continuum closure</td>
</tr>
<tr>
<td>Gibson and Spalding</td>
<td>Explicit finite difference</td>
<td>Mean turbulence field; equations</td>
<td>Second-order, two-equation turbulence model</td>
<td>Simultaneous solutions of equations, including the region closure streamwise intermittency function</td>
<td>Julius E. Harris, M.S., Langley Research Center, Hampton, VA 23665</td>
</tr>
<tr>
<td>Ref. 142</td>
<td>Not available</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ref. 143</td>
<td>Not available</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**TABLE A1 - Continued**

<table>
<thead>
<tr>
<th>Method</th>
<th>Numerical solution method</th>
<th>Closure assumption</th>
<th>Unique features</th>
<th>Computing machine; storage</th>
<th>Code source</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dwyer</td>
<td>Implicit finite difference</td>
<td>Nominal Cebeci-Smith two-layer viscosity (ref. 12)</td>
<td>Includes transverse curvature, equilibrium</td>
<td>CDC 6600; FORTRAN IV; 6200B core storage</td>
<td>Not available</td>
</tr>
<tr>
<td>Fivel and Masek</td>
<td>Implicit finite difference</td>
<td>Nominal Cebeci-Smith two-layer viscosity (ref. 12)</td>
<td>Nonequilibrium, and frozen, chemical, catalytic</td>
<td>Could be punched from ref. 232 (uses basic Patankar-Spalding code)</td>
<td>Julius E. Harris, M.S., Langley Research Center, Hampton, VA 23665</td>
</tr>
<tr>
<td>Carr and Fox</td>
<td>Implicit finite difference</td>
<td>Nominal Cebeci-Smith two-layer viscosity (ref. 12)</td>
<td>Uses physical coordinates</td>
<td>CDC 6600; FORTRAN IV; 6200B core storage</td>
<td>continuum closure</td>
</tr>
<tr>
<td>Gibson and Spalding</td>
<td>Explicit finite difference</td>
<td>Mean turbulence field; equations</td>
<td>Second-order, two-equation turbulence model</td>
<td>Simultaneous solutions of equations, including the region closure streamwise intermittency function</td>
<td>Julius E. Harris, M.S., Langley Research Center, Hampton, VA 23665</td>
</tr>
<tr>
<td>Ref. 142</td>
<td>Not available</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ref. 143</td>
<td>Not available</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Method</td>
<td>Major reference</td>
<td>User's manual</td>
<td>Numerical solution method</td>
<td>Closure assumption</td>
<td>Unique features</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>-----------------</td>
<td>---------------</td>
<td>---------------------------</td>
<td>--------------------</td>
<td>----------------</td>
</tr>
<tr>
<td>Kendall and Anderson (BLIMP)</td>
<td>34, 163, 304</td>
<td>Ref. 305</td>
<td>Integral matrix</td>
<td>Two-layer eddy viscosity with unique wall damping treatment</td>
<td>Ideal gas or equilibrium chemistry; transverse curvature and entropy layer treatments; ablation cases with chemistry; numerical method permits small number of nodes</td>
</tr>
<tr>
<td>Kuhn</td>
<td>164</td>
<td>Ref. 306</td>
<td>Method of weighted residuals</td>
<td>Two-layer eddy viscosity</td>
<td>Transitional flow calculation</td>
</tr>
<tr>
<td>Levine (MABLE)</td>
<td>149</td>
<td>Ref. 150</td>
<td>Implicit finite difference</td>
<td>Nominal Cebeci-Smith two-layer eddy viscosity (ref. 12)</td>
<td>Transformation for variable mode spacing in y; equilibrium hydrogen-air chemistry and ideal gas</td>
</tr>
<tr>
<td>Mayne</td>
<td>223, 221 for earlier version (without normal pressure gradient and curvature)</td>
<td>Not available</td>
<td>Implicit finite difference with Von Mises variables (modification and extension of Patankar-Spalding method, ref. 9)</td>
<td>Mixing length</td>
<td>Includes p(y) due to longitudinal curvature and other longitudinal curvature terms; transition region calculation</td>
</tr>
<tr>
<td>Method</td>
<td>Major reference</td>
<td>User's manual</td>
<td>Numerical solution method</td>
<td>Closure assumption</td>
<td>Unique features</td>
</tr>
<tr>
<td>------------------</td>
<td>-----------------</td>
<td>---------------</td>
<td>---------------------------</td>
<td>------------------------------------------------------------------------------------</td>
<td>-----------------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
| McDonald         | 159, 307        | Ref. 308      | Implicit finite difference | Integral turbulent kinetic energy (with Bradshaw-type assumptions) or mixing length | Transitional flow; rough walls; freestream turbulence; equilibrium and nonequilibrium chemical reactions | UNIVAC 1108 or 1110 or CDC 6600; FORTRAN; 2500010 or 4000010, depending upon option or overlay | B. H. Anderson  
M.S. 86  
NASA Lewis Research Center  
21000 Brookpark Rd.  
Cleveland, OH 44135 |
| Omori            | 148             | Ref. 148      | Implicit finite difference (uses basic code of ref. 150) | Turbulent kinetic energy equation with algebraic length scale and Prandtl model relating u'v' and e | Both ideal gas and equilibrium chemistry; one-equation turbulent kinetic energy closure | UNIVAC 1108; FORTRAN IV; 5900010 core storage                                                    | Klaus Gross  
M.S. EP23  
NASA Marshall Space Flight Center  
Huntsville, AL 35812 |
| Patankar and Spalding | 9               | Ref. 9        | Imperial College micro-integral method | Mixing length or turbulent kinetic energy closure (one- and two-equation models) | Generality of closure methods; also handles pipe and diffuser flows | Decks available for most computers using FORTRAN language                                    | D. B. Spalding  
Mechanical Engineering Dept.  
Imperial College of Science and Technology  
Exhibition Rd.  
London SW7, England |
| Fletcher         | 158             | Unpublished guide and notes available from author | Explicit finite difference | Mixing length                                                                | Explicit numerics                                                                    | IBM 360/66 or IBM 370/158; FORTRAN IV                                                        | R. H. Fletcher  
Iowa State Univ.  
Ames, IA 50010 |
| Rehner           | 45 (pp. 375-383) | Ref. 309      | Implicit finite difference | Two-layer eddy viscosity with simplified $\epsilon$ equation for nonequilibrium flows | Uses physical coordinates; simplified differential equation to compute $\epsilon$ in nonequilibrium flows | CDC 6600; FORTRAN IV; 2700010 core storage                                                      | Available by purchase from The Boeing Co.,  
Seattle, WA 98124 |
<table>
<thead>
<tr>
<th>Method</th>
<th>Major reference</th>
<th>User's manual</th>
<th>Numerical solution method</th>
<th>Closure assumption</th>
<th>Unique features</th>
<th>Computing machine; language; storage</th>
<th>Code source</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rie</td>
<td>144, 310</td>
<td></td>
<td>Implicit finite difference</td>
<td>Two-layer eddy viscosity</td>
<td>Transverse curvature effect; equilibrium chemistry</td>
<td>Honeywell 6060; FORTRAN</td>
<td>A. Martellucci GE-RESRD, P.O. Box 8555 Philadelphia, PA 19101 (user's manual available by purchase)</td>
</tr>
<tr>
<td>Shang and Hankey</td>
<td>147, 170, 274</td>
<td>Not available</td>
<td>Implicit finite difference</td>
<td>Two-layer eddy viscosity</td>
<td>Transition region using intermittency function</td>
<td>CDC; FORTRAN</td>
<td>J. S. Shang Wright-Patterson Air Force Base OH 45433</td>
</tr>
<tr>
<td>Verma</td>
<td>213</td>
<td>Ref. 311</td>
<td>Implicit finite difference</td>
<td>Two-layer eddy viscosity</td>
<td>Well exercised</td>
<td>IBM 360/165</td>
<td>Can be punched from ref. 311</td>
</tr>
<tr>
<td>Waiter and LeBlanc</td>
<td>153, 312</td>
<td>Ref. 153</td>
<td>Implicit finite difference</td>
<td>Two-layer eddy viscosity</td>
<td>Uses physical variables; variable entropy</td>
<td>IBM 360/370; FORTRAN IV</td>
<td>Not available</td>
</tr>
<tr>
<td>Wilcox and Chambers</td>
<td>231, 313</td>
<td>Not available (basic code same as ref. 240)</td>
<td>Implicit finite difference</td>
<td>Two-equation mean turbulence field (Saffman model)</td>
<td>Surface roughness; transition</td>
<td>UNIVAC 1108 or CDC; FORTRAN</td>
<td>DCW Industries 13535 Ventura Blvd. Sherman Oaks, CA 91423</td>
</tr>
<tr>
<td>Wortman</td>
<td>162, 314</td>
<td>Not available</td>
<td>Operator method</td>
<td>Mixing length or one- or two-equation mean turbulent field</td>
<td>Efficient numerical method</td>
<td>IBM or CDC</td>
<td></td>
</tr>
</tbody>
</table>

**TABLE A1.- Concluded**
<table>
<thead>
<tr>
<th>Method</th>
<th>Major reference</th>
<th>Numerical solution procedure</th>
<th>Turbulence closure procedure</th>
<th>Unique features</th>
<th>Code source</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kendall et al.</td>
<td>245</td>
<td>Integral matrix</td>
<td>Scalar-invariant eddy-viscosity mixing-length model</td>
<td>Equilibrium and frozen flow; orthogonal body-oriented coordinate system; primitive variables; fast, low storage</td>
<td>R. M. Kendall Acurex Corp. Aerotherm Division 485 Clyde Ave. Mountain View, CA 94043 or J. E. Harris, LaRC</td>
</tr>
<tr>
<td>Cebeci et al.</td>
<td>244, 260</td>
<td>Keller box method, implicit finite difference</td>
<td>Scalar-invariant eddy-viscosity two-layer model</td>
<td>Perfect gas; nonorthogonal body-oriented coordinate system; fast, low storage</td>
<td>T. Cebeci Douglas Aircraft Co. 3855 Lakewood Blvd. Long Beach, CA 90808 or J. E. Harris, LaRC</td>
</tr>
<tr>
<td>Harris</td>
<td>271</td>
<td>Implicit finite difference</td>
<td>Scalar-invariant eddy-viscosity two-layer model; mixing-length model</td>
<td>Perfect gas; orthogonal body-oriented coordinate system; Crocco transformation; fast, low storage</td>
<td>J. E. Harris M.S. 163 NASA Langley Research Center Hampton, VA 23665</td>
</tr>
<tr>
<td>Frieders and Lewis</td>
<td>248</td>
<td>Implicit finite difference</td>
<td>Scalar-invariant eddy-viscosity two-layer model</td>
<td>Binary gas mixture with wall mass transfer; orthogonal, body-oriented coordinate system; stream function with Levy-Lees transformation</td>
<td>M. L. Laster R&amp;D Directorate AEDC, Arnold Air Force Station, TN 37389</td>
</tr>
<tr>
<td>Adams</td>
<td>315, 316</td>
<td>Implicit finite difference</td>
<td>Scalar-invariant eddy-viscosity two-layer model</td>
<td>Perfect gas; streamline coordinate system</td>
<td>J. C. Adams, Jr. Von Kármán Gas Dynamics Facility, AEDC Arnold Air Force Station, TN 37389</td>
</tr>
<tr>
<td>Method</td>
<td>Major reference</td>
<td>Numerical solution procedure</td>
<td>Turbulence closure procedure</td>
<td>Unique features</td>
<td>Code source</td>
</tr>
<tr>
<td>--------------</td>
<td>----------------</td>
<td>-------------------------------</td>
<td>-----------------------------</td>
<td>------------------------------------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Wortman</td>
<td>162</td>
<td>Operator procedure</td>
<td>Scalar-invariant eddy-viscosity two-layer model</td>
<td>Equilibrium and frozen flow; orthogonal coordinate system</td>
<td></td>
</tr>
<tr>
<td>Spalding</td>
<td>319, 320</td>
<td>Micro-integral (ref. 9)</td>
<td>Mixing-length and higher order modeling</td>
<td>Equilibrium and frozen flow; orthogonal coordinate system</td>
<td>D. B. Spalding Imperial College of Science and Technology Exhibition Rd. London SW7, England</td>
</tr>
<tr>
<td>Nash and Patel</td>
<td>267</td>
<td>Explicit finite difference</td>
<td>Scalar-invariant eddy-viscosity mixing-length model and higher order modeling</td>
<td>Perfect gas; orthogonal coordinate system</td>
<td>J. F. Nash Sybucon Inc. 9 Perimeter Rd. NW Atlanta, GA 30339</td>
</tr>
<tr>
<td>Fannelöp and Humphreys</td>
<td>253</td>
<td>Implicit finite difference</td>
<td>Scalar-invariant eddy-viscosity two-layer model</td>
<td>Perfect gas; streamline coordinates</td>
<td>T. K. Fannelöp The Aeronautical Research Institute of Sweden Stockholm, Sweden</td>
</tr>
<tr>
<td>Krause et al.</td>
<td>321</td>
<td>Implicit finite difference &quot;Mehrstellen&quot; method</td>
<td>Scalar-invariant eddy-viscosity two-layer model</td>
<td>Perfect gas; Nonorthogonal body-oriented coordinate system</td>
<td>E. Krause Aerodynamisches Institut der RWTH 51 Aachen Wullnerstra Be zw 5-7 West Germany</td>
</tr>
</tbody>
</table>
REFERENCES


95. Bertram, Mitchel H.; and Neal, Luther, Jr.: Recent Experiments in Hypersonic Turbulent Boundary Layers. Presented at the AGARD Specialists' Meeting on Recent Developments in Boundary-Layer Research (Naples, Italy), May 1965. (Also available as NASA TM X-56335.)


Figure 1.- Typical distribution of $p$ with $y$ at high Mach number.

Figure 2.- Wall pressure increase at high Mach number (from ref. 50).
Figure 3. - Comparison of $T_{\text{static}}$ with $\frac{u'^2}{2cp}$ at high Mach numbers. $\sqrt{(u'^2/u_e^2)} = 0.05$; $T_w/T_t = 1$; $\gamma = 1.4$.

$\frac{T}{T_t}$ vs $M$

$\frac{3}{2} \left[ \frac{u'^2}{2cpT_t} \right]$

Figure 4. - Illustration of disturbances upstream of transition.
Figure 5.— Flow field schematic of precursor transition for Mach 18 cone experiments (taken from ref. 57).

GROWTH OF TURBULENCE IN A HYPERSONIC BOUNDARY LAYER BASED ON SCHLIEREN PHOTOGRAPHS AND SURFACE HEAT TRANSFER

Figure 6.— Typical velocity profile at beginning of conventional transition region for $M >> 1$ (from ref. 58). $M_e = 8.96; \frac{T_w}{T_t} = 0.656; x_{tr} = 54.4 \text{ cm};$ end of transition at approximately 109.3 cm.
Figure 7.- Influence of precursor transition effect on wall heating (from ref. 60).

Figure 8.- Height of critical layer as function of Mach number (from ref. 62).
Figure 9.- Influence of Mach number on fluctuating wall pressure levels for turbulent boundary-layer flow (from ref. 47).

Figure 10.- Distribution of fluctuating static pressure through a turbulent boundary layer in high Mach number flow. $M_e = 9.4$; $Re_\theta = 36,800$; $T_w/T_t = 0.38$. (Reprinted from ref. 62 with permission from Cambridge University Press.)
Figure 11.- Illustration of difference between pitot (or density) and velocity boundary-layer thicknesses at high Mach number (from ref. 47). $\delta_{\text{density}}$ is y-location where $\rho/\rho_e = 0.995$ and $\delta_{\text{velocity}}$ is y-location where $u/u_e = 0.995$.

Figure 12.- Prediction of flat-plate skin friction by transformation methods of Baronti and Libby (ref. 75) and Economos and Boccio (ref. 78).
Figure 13.- Prediction of flat-plate skin friction and heating by transformation method of Coles (ref. 15).

\[
\frac{dp}{dx} = 0 \\
M = 4 \text{ TO } 13 \\
\frac{T_w}{T_t} = 0.14 \text{ TO } 0.7 \\
\text{ASSUMED} \\
\frac{N_{St}}{C_f} = 1.0
\]
Figure 14.—Prediction of boundary-layer characteristics on a body of revolution (data from ref. 83).

(a) Economos and Boccio (ref. 78).

(b) Lewis et al. (ref. 77).
HEAT TRANSFER

\( \frac{F_c}{C_f} \) vs. \( \text{Re}_{x,v_0} \)

1291 DATA POINTS

Figure 15.—Prediction of flat-plate skin friction and heating by correlation method of Van Driest (ref. 87). \( \text{M} = 0 \) to 10; \( \text{T}_W/\text{T}_t = 0.14 \) to 0.7.

SKIN FRICTION

185 DATA POINTS

HEAT TRANSFER

(VON KÁRMAÍÍN'S R.A.F. ASSUMED)

\( \frac{F_c}{C_f} \) vs. \( \text{Re}_{x,v_0} \)

1291 DATA POINTS

Figure 16.—Prediction of flat-plate skin friction and heating by correlation method of Spalding and Chi (ref. 81). \( \text{M} = 0 \) to 10; \( \text{T}_W/\text{T}_t = 0.14 \) to 0.7.

(R.A.F., Reynolds analogy factor.)
Figure 17.—Prediction of flat-plate skin friction and heating by correlation method of Eckert (ref. 72). \( M = 0 \) to 10; \( T_W/T_t = 0.14 \) to 0.7. (R.A.F., Reynolds analogy factor.)

Figure 18.—Prediction of turbulent skin friction on conventional flight vehicles by Eckert's method (ref. 72).
Figure 19.- Prediction of turbulent heating on sharp and blunt tipped axisymmetric bodies in flight by Eckert's method (ref. 72); data from reference 96.
(a) Flat plate with favorable pressure gradient (experimental data from ref. 138).

(b) Waisted body of revolution (experimental data from ref. 83).

Figure 20.- Prediction of boundary layer properties for turbulent flows with pressure gradient by Flaherty's integral method (ref. 124).
(a) Waisted body of revolution (experimental data from ref. 83).

(b) Flat cylinder (experimental data from ref. 139).

Figure 21.- Prediction of boundary-layer properties for flows with pressure gradients by Green's integral method (ref. 135).
Figure 22. Prediction of boundary-layer development on a concave surface by Pinckney's integral method (ref. 133). Experimental data from reference 141.
Figure 23.- Prediction of boundary-layer properties for flows with pressure gradient using Reeves' integral method (ref. 30, pp. 6-1 - 6-A2-2).
Figure 24.- Effect of grid spacing on computational error (from ref. 169).

Figure 25.- Effect of variable grid spacing on calculated momentum thickness and skin friction. (Reprinted from ref. 170 with permission from the American Institute of Aeronautics and Astronautics.)
Figure 26. - Effect of wall blowing on wall damping parameter (from ref. 53).

Figure 27. - Effect of $dp/dx$ on $K$ (from ref. 188).
Figure 28.- Comparison of mixing length and Clauser form of eddy viscosity.
Figure 29.- Variation of $(1/\delta)_{\text{max}}$ with $\delta^+$ for various types of flow (from ref. 189).
Figure 30.- Illustration of increasing importance of low Reynolds number effects at high Mach number.

Figure 31.- Effect of wall injection and positive pressure gradient on \((1/\delta)_{\text{max}}\) (from ref. 53).
MEIER AND ROTTA (REF. 193)
KAYS (REF. 181)
BLOM (REF. 194)
MEIER, VOISINET, AND GATES (REF. 195)
SIMPSON, WHITTEN, AND MOFFAT (REF. 196)

Figure 32.- Variation of turbulent Prandtl number with $y^+$.

Figure 33.- Comparison of static with total turbulent Prandtl number at Mach 7.2. (Reprinted from ref. 177 with permission from the American Institute of Aeronautics and Astronautics.)
Figure 34.- Variation of Reynolds analogy factor with $\delta^+$.

Figure 35.- Theoretical variation of turbulent Prandtl number with $\varepsilon/\mu$. (Reprinted with permission from ref. 12.)
Figure 36.— Variation of normal intermittency through turbulent boundary layer up to Mach 9.4 (from ref. 25).
Figure 37.- Total temperature-velocity profiles for flat-plate type turbulent flow (from ref. 209).
Figure 38.- Total temperature-velocity profiles for nozzle wall-type turbulent flow (from ref. 209).
Figure 39. - Development of total temperature-velocity profile along a nozzle wall (data from ref. 212).

Figure 40. - Influence of wall blowing on turbulent skin friction (from ref. 217).
Figure 41.- Extent of transition zone in high speed flow. (Reprinted from ref. 219 with permission from the American Institute of Aeronautics and Astronautics.)

Figure 42.- Effect of Mach number on streamwise intermittency. (Reprinted from ref. 59 with permission from the American Institute of Aeronautics and Astronautics.)
Figure 43.- Comparison of transitional calculations including precursor and low Reynolds number effects with data (from ref. 58). The profile is near center of transition region ($x = 78.7$ cm).

Figure 44.- Effects of variable entropy on wall heating data and calculations (from ref. 221). $M = 10$; $T_t = 1111$ K; $T_w = 294$ K; $A^* = 26$; $K = 0.435$; $(1/\delta)_{\text{max}} = 0.09$; $N_{\text{Pr,t}} = 0.9$. 

131
Figure 45.— Application of mean field closure for turbulent boundary-layer flow with pressure gradient. $K = 0.4; \alpha = 0.016; A^+ = f(p^+)$. (Reprinted with permission from ref. 12.)

Figure 46.— Flow conditions for data in figure 45.
Figure 47.- Effect of fluctuating stream velocity on mixing length (from ref. 233).

Figure 48.- Effect of fluctuating stream velocity on Bradshaw scale (from ref. 233).
Figure 49.- Effect of fluctuating stream velocity on $a_1$ (from ref. 233).

Figure 50.- Values of $a_1$ through a shock-boundary-layer interaction at Mach 3.88 (from ref. 234).
Figure 51. Calculation of $u_i u_j$ with a turbulent boundary layer using nonequilibrium turbulence modeling (from ref. 156).

Figure 52. Calculation of $u'v'$ with a turbulent boundary layer using nonequilibrium turbulence modeling (from ref. 156).
\[ p_{t,\infty}; T_{t,\infty}; M_{\infty} \]

\[ \text{SHOCK WAVE} \]

\[ \frac{d}{V_\infty} \]

\[ \text{LAMINAR} \]

\[ \text{TURBULENT} \]

\[ \text{TURBULENT SPECIFY: p, = g(C, Q_1)} \]

\[ \frac{d s^2}{d} = (h_1 d \xi)^2 + (h_2 d \eta)^2 + (h_3 d \zeta)^2 \]

**Figure 53.** Schematic of geometry and flow field.

\[ \eta, x_2 \]

\[ \xi, x_1 \]

\[ \zeta, x_3 \]

\[ \tau_{x_1} = \rho l^2 \left[ \left( \frac{\partial u_1}{\partial x_3} \right)^2 + \left( \frac{\partial u_2}{\partial x_3} \right)^2 \right]^{1/2} \frac{\partial u_1}{\partial x_3} \]

\[ \tau_{x_2} = \rho l^2 \left[ \left( \frac{\partial u_2}{\partial x_3} \right)^2 + \left( \frac{\partial u_3}{\partial x_3} \right)^2 \right]^{1/2} \frac{\partial u_2}{\partial x_3} \]

\[ \varepsilon = \varepsilon_{x_1} = \varepsilon_{x_2} = \rho l^2 \left[ \left( \frac{\partial u_1}{\partial x_3} \right)^2 + \left( \frac{\partial u_2}{\partial x_3} \right)^2 \right]^{1/2} \]

**Figure 54.** Invariant turbulence concept (from ref. 243).
Figure 55.- Illustration of finite-difference grid schemes.
Figure 56.- Comparison of three-dimensional prediction scheme with data (from ref. 276) for a cone at angle of attack in a supersonic stream ($d = 2.54$ cm).
### Storage and Processing with CDC 6600

<table>
<thead>
<tr>
<th>N</th>
<th>CPU/Mesh Pt., sec</th>
<th>Storage</th>
<th>Test Case Time, sec</th>
</tr>
</thead>
<tbody>
<tr>
<td>301</td>
<td>.0123</td>
<td>1700000</td>
<td>900</td>
</tr>
<tr>
<td>21</td>
<td>.0123</td>
<td>600000</td>
<td>60</td>
</tr>
</tbody>
</table>

**Numerical Results**

<table>
<thead>
<tr>
<th>N</th>
</tr>
</thead>
<tbody>
<tr>
<td>301</td>
</tr>
<tr>
<td>21</td>
</tr>
</tbody>
</table>

- **Mixing Length**
- **Two Layer**

#### Graphical Representation

- **Figure 56.** Concluded.
Figure 57.— Comparison of numerical results with experimental data (d = 2.54 cm).

Figure 58.— Numerical results for supercritical wing. $M_\infty = 0.5; \ R_\infty = 1.5 \times 10^6$. 
BOUNDARY LAYER IS COVERED BY A NODAL NETWORK

SPLINED 2nd ORDER TAYLOR SERIES NORMAL TO WALL

CENTERED FINITE DIFFERENCE FOR CROSS-FLOW DERIVATIVES

SOLUTION KNOWN

SOLUTION TO BE DETERMINED
FULLY IMPLICIT

Figure 59.- Mesh point network for integral-matrix procedure.

\[ N_{st}/(N_{st})_{\alpha} \]

- NUMERICAL RESULTS
- EXPERIMENTAL DATA

Figure 60.- Comparison of numerical results with surface heating data (from ref. 292) for a cone at angle of attack in hypersonic flow. \( M_{\infty} = 7.95; \gamma = 7.95; \alpha = 4^\circ \).