SOME STUDIES ON THREE DIMENSIONAL TURBULENT BOUNDARY LAYER FLOWS

R. Swaminathan* and R. Balu**

Abstract

This paper describes some studies on flow separation and on the extended Cebeci-Smith turbulence and transition models, carried out using a three-dimensional turbulent boundary layer code developed at the Vikram Sarabhai Space Center (VSSC), Thiruvananthapuram, India. This code uses an orthogonal curvilinear surface fitted co-ordinate system. The governing equations are solved, by an implicit space marching numerical method called the Keller Box Scheme with suitable modifications for reverse crossflows. Initial data on two intersecting planes for starting the solution procedure, are obtained by solving the appropriate limiting form of the governing equations. Three transition models available in the literature are studied. A few sample results are presented and compared with experimental measurements.

Key words: Boundary Layer, Turbulence and Transition Models, Separation

Symbols

\[ \delta_1 = \text{streamwise displacement thickness} \]
\[ \delta_2 = \text{crossflow displacement thickness} \]
\[ C_f = \text{skin friction coefficient} \]
\[ \mu = \text{coefficient of viscosity} \]
\[ \theta = \text{non-dimensional total enthalpy} \]
\[ \theta_{11} = \text{streamwise momentum thickness} \]
\[ \theta_{22} = \text{crossflow momentum thickness} \]
\[ \text{Pr} = \text{Prandtl number} \]
\[ \text{Re} = \text{Reynolds number} \]
\[ \text{Pr}_t = \text{turbulent Prandtl number} \]

* Scientist/Engineer, ** Head, Aerodynamics Division, Vikram Sarabhai Space Centre, Thiruvananthapuram-695 022, India

Manuscript received on 25 Jun 2002; Paper reviewed, revised and accepted on 24 Dec 2002
\( \lambda \) = sweep angle \\
\( \omega \) = angular velocity \\
\( \pi \) = wake parameter in the outer eddy viscosity law \\
\( \beta \) = cross flow angle \\

**Subscripts**

e = edge of the boundary layer \\
w = on the wall \\
\( \infty \) = free stream \\
t = onset of transition \\
\( \xi, \omega \) = derivatives w.r.t. \( \xi, \omega \) \\

**Subscripts**

\( '(prime) \) = derivatives w.r.t. \( \eta \), \\
fluctuating quantities \\
\(-(bar)\) = time averaged quantities \\

**Introduction**

Computation of three dimensional boundary layers plays an important role in the design of turbo machinery, automobiles, ships, aircraft, missiles and rockets. Provided there are no strong interactions, like shocks or separation, it takes less computational effort (compared to full Navier Stokes simulations), to treat the inviscid and viscous regions separately, in an iterative manner. Studies on turbulence and transition are imperative for optimal (from the point of view of skin friction drag and heat transfer) configuration design of aerospace vehicles, yet modelling of turbulence and transition (especially for 3-D flows) still remains a challenging task of fluid mechanics. A 3-D boundary layer code thus serves as an ideal test bed for evaluating different turbulence and transition models, and for flow separation studies. Moreover, it is easier to obtain accurate estimates of skin friction and heat transfer using a boundary layer code compared to an N-S code. 

In this paper, details of a three dimensional boundary layer code developed in VSSC are given.

This code computes the flows over blunt bodies with a plane of symmetry and requires as input the inviscid velocity distribution. It should be emphasized that the present formulation breaks down when streamwise separation occurs and cannot be used for analysing separated regions of flow. However, using this code one can march very close to the separation line. Also described are some studies on flow separation and the extended Cebeci-Smith turbulence and transition models, using this code. The extended Cebeci-Smith model (for 3-D flows and for low Reynolds numbers) is found to give better results compared to the original version. Three transition models are studied: Chen - Thyson, Dey - Narasimha and Arnal. Of these, the Arnal model can be readily extended for 3-D flows, while the other two can be extended for certain types of flow.

**Co-ordinate System**

An orthogonal co-ordinate system is preferable, since boundary conditions can be specified easily. For boundary layer calculations, it is enough if the co-ordinate system is orthogonal on the surface, (since the boundary layer thickness is assumed to be small compared to the local radius of curvature of the body) with the third co-ordinate being normal to the surface. There are several ways in which a surface orthogonal system can be chosen, e.g. lines of curvature, stream line co-ordinates etc. However, for practical configurations, it is difficult to determine these co-ordinates.

An orthogonal curvilinear surface fitted co-ordinate system proposed by Blottner and Ellis [1] has been used in the present formulation (Fig.1(a)). In this system, the origin is placed at the stagnation point, to remove the flow singularity. It is assumed that the flow is along the body plane of symmetry. One set of co-ordinate lines (\( \omega = \text{const} \)) is formed by the intersection with the body surface of the meridional planes passing through an axis containing the stagnation point. This axis is parallel to the body axis through the vertex. The other set of orthogonal co-ordinate lines (\( \xi = \text{const} \)) are calculated numerically. The third co-ordinate is the surface normal. The only drawback of this co-ordinate system is that the location of 

![Fig. 1a Blottner's orthogonal surface coordinate system for blunt body](image-url)
the stagnation point must be known a priori (from inviscid solution) and for every angle of incidence the surface grid must be generated numerically. The code has options for 3-D, 2-D or axisymmetric flows.

The 3-D boundary layer code requires, as input, the body geometry and the inviscid velocity components on the body. The body surface is specified in the form \( r = r(x, \phi) \), where 'r' is the distance of any point on the surface from the body axis, \( x \) is the axial location of the cross section from the vertex and \( \phi \) is the meridional angle measured from the symmetry plane. From the body geometry and the location of the stagnation point, Blottner’s surface co-ordinates and the metric coefficients \( h_1 \) and \( h_2 \) are calculated numerically. For two dimensional and quasi 3-D flows, \( h_1 \) and \( h_2 \) are taken as unity. The third metric coefficient \( h_3 \) is taken as unity.

Since the inviscid velocities are usually specified in body co-ordinates, they have to be transformed into Blottner’s co-ordinate system for blunt bodies. Details are given in [1]. The velocity gradients in the external flow, \( u_e, w_e, w_e^2, w_e \xi, w_e \omega \) are calculated numerically by fitting quadratic polynomials.

### Governing Equations

The steady time averaged turbulent boundary layer equations can be written, in the above coordinate system, as [2]

**Continuity Equation**

\[
\rho \frac{h_2}{h_1} \frac{du}{d\xi} + \left( \rho \frac{h_1}{h_2} \frac{dw}{d\omega} \right) + \left( \frac{h_1}{h_2} \frac{\partial \bar{v}}{\partial \xi} \right) = 0
\]

**\( \xi \)- Momentum Equation**

\[
\frac{\rho u}{h_1} \frac{du}{d\xi} + \frac{\rho w}{h_2} \frac{dw}{d\omega} + \bar{v} u - \rho u w K_2 + \rho w^2 + K_1 \\
= -\frac{P_2}{h_1} + \left( \mu \frac{u}{\xi} - \rho u' v' \right)
\]

**\( \omega \)- Momentum Equation**

\[
\frac{\rho u}{h_1} \frac{dw}{d\xi} + \frac{\rho w}{h_2} \frac{dw}{d\omega} + \bar{v} w - \rho u w K_1 + \rho u^2 K_2 \\
= -\frac{P_2}{h_2} + \left( \mu \frac{w}{\omega} - \rho w' v' \right)
\]

**Energy Equation**

\[
\rho \frac{u}{h_1} H_x + \rho \frac{w}{h_2} H_\omega + \bar{v} H_z \\
= \left[ \frac{h_2}{h_1} \frac{1}{P_r} \left( \frac{u^2 + w^2}{2} \right) - \rho v' H_z \right]
\]

**Equation of State**

\[
p = \rho RT \text{ (Compressible Flow)}
\]

\[
p = \text{constant} \text{ (Incompressible Flow)}
\]

The boundary layer edge conditions are

\[
\rho_e u_e u_e^2 / h_1 + \rho_e w_e w_e / h_2 - \rho_e u_e w_e K_2 \\
+ \rho_e w_e K_1 = -p_e h_1
\]

\[
\rho_e u_e w_e w_e^2 / h_1 + \rho_e w_e w_e / h_2 - \rho_e u_e w_e K_1 \\
+ \rho_e u_e K_2 = -p_e h_2
\]

Special forms of the boundary layer equations are described in Appendix-1. The extended Cebeci- Smith algebraic turbulence model is described in Appendix-2.

### Boundary Conditions

At the wall, no slip conditions are imposed for the velocity. The code has provision for mass transfer at the wall, but \( v_w \) is set to zero in all the computations reported in this paper. Wall temperature is prescribed (300 K). At the outer edge, velocity and enthalpy are assumed to reach the inviscid values asymptotically.

At \( \xi = 0, u = w = 0, v = v_w (\xi_w, \omega) = 0, H = H_w \)

As \( \xi \rightarrow \infty, u \rightarrow u_e (\xi, \omega), H \rightarrow H_e \)

### Transformations

A two component stream function \((\Psi, \Phi)\) is defined to eliminate the continuity equation as follows:

\[
\rho \frac{h_2}{h_1} u = \frac{h_2}{h_1} \Psi, \rho \frac{h_1}{h_2} w = \frac{h_1}{h_2} \Phi
\]

\[
= h_1 h_2 (\rho v) w - (\Psi + \Phi)
\]

The following transformations are also introduced.

\[
\xi = \xi_w
\]
\[ \omega = \omega \]
\[ d \eta = \left[ \frac{u_e}{\rho_e \mu_e \xi} \right]^{1/2} \rho \, d \zeta \]
\[ \Psi = \left( \frac{u_e}{\rho_e \mu_e \xi} \right)^{1/2} h_2 f(\xi, \omega, \eta) \]
\[ \phi = \frac{\omega}{u_e} \left( \frac{u_e}{\rho_e \mu_e \xi} \right)^{1/2} h_1 g(\xi, \omega, \eta) \]
\[ f' = \frac{u}{u_e}, \quad g' = \frac{\omega}{w_e}, \quad \theta = \frac{H}{H_e} \]

With the above transformations, the governing equations become

**\( \xi \) Momentum**

\[ [C (1 + \epsilon^+(\xi)) f'' + P_2 f'' + P_3 g' f'' + P_4 (\frac{\rho_e}{\rho} - f'')] \]
\[ + P_5 \left( \frac{\rho_e}{\rho} g' - \frac{\rho_e}{\rho} \right) + P_6 f'' \]
\[ = P_8 \left( f' g' - f'' g' + P_9 \left( g' - \frac{\epsilon^+(\xi)}{\rho_e} \right) \right) \]

**\( \omega \) Momentum**

\[ [C (1 + \epsilon^+(\omega)) g'' + P_2 f' g'' + P_3 g g'' + P_4 (\frac{\rho_e}{\rho} - f'')] \]
\[ + P_5 \left( \frac{\rho_e}{\rho} g' - \frac{\rho_e}{\rho} \right) + P_6 g'' \]
\[ = P_8 \left( f' g' - g'' f' + P_9 \left( g' - \frac{\epsilon^+(\omega)}{\rho_e} \right) \right) \]

**Energy**

\[ [C \left( 1 + \epsilon^+ \frac{Pr}{Pr_f} \right) \Theta' + \left( 1 - \frac{1}{Pr} \right) \left( \frac{u_e^2}{H_e} f'' + \frac{w_e^2}{H_e} g'' \right)]' \]
\[ + P_2 f \Theta' + P_5 g \Theta' + P_9 g' (f' - \frac{\epsilon^+(\omega)}{\rho_e}) \]
\[ + P_9 \left( g' \frac{\epsilon^+(\omega)}{\rho_e} - \frac{\epsilon^+(\omega)}{\rho_e} \right) \]

where

\[ \begin{align*}
P_1 &= \frac{\xi}{h_2} \frac{W_{e_a}}{U_e} \\
P_2 &= \frac{1}{2 h_1} \left[ 1 + \xi \frac{U_e}{U_e} + \xi \frac{(\rho_e \mu_e)_o}{\rho_e \mu_e} - 2 K_1 h_1 \xi \right] \\
P_3 &= \frac{W_e}{U_e} \frac{1}{2 h_2} \left[ 2 \xi \frac{W_{e_a}}{U_e} - \xi \frac{W_{e_a}}{U_e} + \xi \frac{(\rho_e \mu_e)_{o}}{\rho_e \mu_e} - 2 K_2 h_2 \xi \right] \\
P_4 &= \frac{W_e}{U_e} K_3 \xi, \quad P_5 = \frac{W_e}{U_e} h_2 \left[ K_3 h_2 \xi - \xi \frac{W_{e_a}}{U_e} \right] \\
P_6 &= \frac{U_{e_a}}{W_e} K_2 \xi, \quad P_7 = \frac{1}{h_1} \left[ K_1 h_1 \xi - \xi \frac{W_{e_a}}{W_e} \right], \quad P_8 = \xi \frac{1}{h_1} \\
P_9 &= \frac{\xi W_e}{U_e}, \quad P_{10} = \xi \frac{U_{e_a}}{U_e} \]

The boundary conditions are also transformed as follows:

At \( \eta = 0 \), \( f = f' = g = g' = 0 \), \( \Theta = \Theta_w \)

As \( \eta \to \infty \), \( f' \to 1 \), \( g' \to 0 \), \( \Theta \to 1 \)

**Numerical Method**

Since the three dimensional boundary layer equations are hyperbolic along surfaces parallel to the body, the solution propagates along characteristic lines. The solution at a point is influenced by a zone of dependence - a curvilinear wedge shaped region bounded by the characteristics [12]. To avoid stability problems, especially in the presence of reverse cross-flows, the numerical domain of dependence should contain the domain of dependence of the differential equation. Krause [13] has suggested a conditionally stable difference scheme which can be used even for negative cross-flows. In the present formulation, the Keller's box scheme has been used for integrating the equations numerically. This is an implicit space marching scheme and is known to be unconditionally stable for positive cross-flows and second order accurate in all the variables. But when negative cross-flows are encountered, as near the separation line, a modified scheme called the Zig-Zag box scheme (Krause scheme) is used. The modified scheme, however, is only conditionally stable, the stability criterion being,

$$\frac{h_1 \Delta \xi}{h_2 \Delta \omega} < \left| \frac{\alpha}{w} \right|$$

**Features of the Box Scheme**

In this scheme, the given set of equations is written as a first order system. The derivatives are approximated by central differences evaluated at the centre of the box formed by the lines of a non-uniform grid in the computational domain. All the variables occurring in the differential equation are averaged at the centre of the box, using the values at the corners of the box. The resulting non-linear algebraic difference equations are solved by a Newton-Raphson procedure, starting from initial profiles for all the variables. The linear system arising from the Newton-Raphson procedure is block tri-diagonal in nature, after augmenting the difference equations at the boundaries. The block tri-diagonal system is solved by a standard L-U decomposition scheme.

The major advantages of the Keller's box scheme are its programming ease, second order accuracy (even for a non-uniform grid) and fast convergence, all of which reduce the computing time and memory considerably. Since all the equations are reduced to a first order system, derivative boundary conditions can easily be specified. Moreover, all the dependent variables and their derivatives are calculated to the same degree of accuracy simultaneously.

In the 'Zig-Zag' box scheme, the difference equations are formed by centering the derivatives and the variables at the centre of the box formed by replacing one of upstream edges in the streamwise direction by a downstream edge in cross flow direction. Cebeci [14,15] has suggested an improvement to the box scheme, in which centering is done taking into account the direction of the characteristics (local streamlines) along which the flow propagates. The centering of the regular Keller's box scheme and the 'Zig-Zag' box scheme are diagrammatically shown in Fig 1(b).

**Solution Procedure**

The solution of the boundary layer equations requires initial data along two intersecting planes. One of them is obtained by solving the limiting form of the governing equations along a plane of symmetry or attachment line. The other initial data plane is obtained by solving the limiting form of the equations at the stagnation point or leading edge, along which the momentum equations become singular. The limiting equations are also transformed and suitable boundary conditions applied on the same lines as the original equations as indicated in Appendix 1.
The present formulation adopts the following steps for calculating the flow over blunt bodies or wings.

i) Obtain the stagnation point (leading edge) solution for each $\omega$.
ii) Obtain the plane of symmetry (attachment line) solution for one step along the body.
iii) March in the circumferential (spanwise) direction for different $\omega$ using the regular box scheme for positive cross flows and the zig-zag box scheme for negative cross flows.
iv) Repeat steps (ii) and (iii) till separation line is encountered, by changing the marching direction if necessary (windward to leeward or vice versa).

The solution of the boundary layer equations amounts to solving a two point boundary value problem at every station. The present code uses a general purpose routine for solving two point boundary value problems [16]. The routine can handle different regions of the flow when the appropriate equations and their Jacobians are specified in functional form. [Appendix-3].

**Initial Profiles, Grid and Convergence**

To start the computations, quadratic velocity and enthalpy profiles are taken as the initial profiles at the stagnation point (leading edge). At subsequent stations, the profiles from the previous station are taken as initial profiles for faster convergence of the iterations. For laminar flows, the length of the transformed normal co-ordinate $\eta$ is taken as a constant equal to 6. When a switch over is made from laminar to transitional or turbulent flow, $\eta$ is increased to allow for growth of the boundary layer thickness, by adding a few more grid points in the normal direction. The profiles at the previous station are correspondingly extended in an asymptotic manner. To improve the accuracy of the wall parameters, the grid is clustered near the wall. The iterations for each station are continued till there is convergence of the profiles to within a specified tolerance.

**Transition Zone Models**

The problem of transition consists of two parts: prediction of onset of transition and computation through the transition zone. The onset of transition is generally decided by a large number of factors like free stream turbulence, pressure gradient, surface roughness, mass and heat transfer etc. This phenomenon is still not understood properly and one has to rely on experimental evidence. The most accepted theory is the laminar instability theory. Many empirical correlations for predicting the onset of transition are found in the literature. In the present study, the location of the onset of transition is assumed to be known.

There are two types of transition models - Linear Combination and Algebraic. In the linear combination model, the mean flow in the transition zone is computed as a weighted average between the mean laminar and turbulent flows, using the intermittency as the weighting factor. In the algebraic model, an effective eddy viscosity is calculated by using the intermittency as a scaling factor.

In both the models, the conditions at the onset of transition are assumed to be known. Strictly, these models have been derived only for 2-D flows. They have to be adapted suitably for 3-D flows. For example, Prabhu [17] has used a transformation for computing 3-D transitional flow using a 2-D model. In practical calculations, if we use a streamline coordinate system, $\theta$ can be interpreted as $\theta_{11}$ and $U_\theta$ can be interpreted as $U_\theta$ (total external velocity). In this paper, we study three transition models available in the literature.

**Dey - Narasimha Model [18]**

This is a linear combination model in which the intermittency is calculated as

$$\gamma = 1 - \exp \left( -0.411 \left( \frac{x - x_1}{\lambda} \right)^2 \right)$$

where $x_1$ is the location of transition onset, $\lambda$ is the stream-wise distance between the points $\gamma = 0.25$ and $\gamma = 0.75$. Since $\lambda$ is not known a priori, it is calculated from

$$\lambda = \left( \frac{0.411 R_{th}^3}{N_2^2} \right)^{1/2} v/U_\theta$$

where $U_\theta$ is the external velocity at the beginning of transition and $R_{th}$ is the Reynolds number based on laminar momentum thickness at the start of transition.

$N_2$ is a non-dimensional turbulent spot formation rate given by

$$N_2 = N_0 + 0.24 L_i^2 (L_i > 0)$$
$$= N_0 - 323 L_i^3 (L_i < 0)$$
where $L_i$ is a pressure gradient parameter given by

$$L_i = \Theta_i^2 \frac{U_e}{\nu}$$

computed at $x_i$, $\Theta_{L_i}$ being the laminar momentum thickness and $U_e$ the external velocity gradient.

$N_0$ is a parameter based on freestream turbulence (for incompressible flows)

$$N_0 = -1.453 \times 10^{-3} \log q - 1.61 \times 10^{-3} (q < 0.2 \%)$$

$$= 0.7 \times 10^{-3} (q > 0.2 \%)$$

**Arnal Model [19]**

This is an algebraic model in which the intermittency is given by

$$\gamma = 1 - \exp \left( -4.5 \chi_1^2 \right) \quad (0 < \chi_1 < 0.25)$$

$$= 18.628 \chi_1^4 - 55.388 \chi_1^3 + 52.369 \chi_1^2$$

$$- 16.501 \chi_1 + 1.893 (0.25 < \chi_1 < 0.75)$$

$$= 1.25 \times 0.25 \sin \left[ \pi \left( 0.444 \chi_1, 0.823 \right) \right] (0.75 < \chi_1 < 3)$$

$$= 1.0 \quad (\chi_1 > 3)$$

where $\chi_i = \theta_i / \Theta_{L_i}$, $\theta_i$ being the momentum thickness and $\Theta_{L_i}$ the momentum thickness at the start of transition. This model can be readily extended for 3-D flows.

**Chen-Thyson Model [20]**

This is also an algebraic model in which the intermittency is given by

$$\gamma = 1 - \exp \left[ -G r_i \int_{x_i}^{x_f} \frac{dx}{r \left( x \right)} \frac{dx}{U_e} \right]$$

where $G = \frac{3 U_e^3}{A^2 v_e} R_{x_i}^{-1.34}$, $A = 60 + 4.68 M_e^{1.92}$

$x_i$, $R_{x_i}$ are taken at the beginning of transition.

**Flow Separation**

In the present approach, marching through the separated region is not possible, since the governing equations become singular near the separation line. The solution breaks down, if there is streamwise separation. However, the region of cross-flow separation can be circumvented by changing the marching direction (from the windward or leeward plane of symmetry). The exact nature and location of the separation line can be determined, only after a careful examination of the boundary layer thickness, displacement thickness, normal velocity component and skin friction near separation. The external flow itself gets considerably affected near separation and the present formulation cannot handle the interaction between the boundary layer and the external flowfield.

**Validation Results**

In this section, we describe some test cases wherein the above mentioned equations and models are applied and compare the computed results with experimental measurements or other computations reported in the literature. In all the cases, the solution is found to be grid independent.

**Laminar Flow over a Bulbous Heat Shield [34]**

In this experiment, heat transfer rates were measured on a scale model of a launch vehicle heat shield at the Indian Institute of Science Shock Tunnel, Bangalore at a freestream Mach number of 5.75, $\alpha = 5^\circ$, $T_0 = 1829$ K and a freestream Reynolds number of $2.3 \times 10^7$ m. The heat shield consists of sphere-cone-cylinder-boattail combination. The inviscid solution, for this configuration was obtained from an Euler code. In this test case, $\Delta \xi = 0.002$, $\Delta \eta_i = 0.005, k = 1.12$. The computed laminar heat transfer rates on the windward and leeward generators are com-
pared with measurements in Fig. 2(a) and Fig. 2(b). The agreement is generally good except at the stagnation point. The discrepancy is probably due to the perfect gas assumptions in the computations.

Laminar Incompressible Flow Past an 1:4 Prolate Spheroid

Cebeci [14,15] and Wang [21] have made extensive calculations for flows past a prolate spheroid, at large angles of incidence. Cebeci has used certain transformations for removing the singularity at the stagnation point. But the co-ordinate system used in the present formulation, does not require any special treatment. Large negative cross flows are encountered before separation. For \( \alpha < 6^\circ \), there is a closed or bubble type of separation. For \( 6^\circ < \alpha < 42^\circ \), there is an open or vortex type of separation. Again for \( \alpha > 42^\circ \), separation occurs very near the nose and is of the closed or bubble type. The separation line is close to the zero cross-flow skin friction \( (C_{f0} = 0) \) line and moves rapidly towards the nose as \( \alpha \) increases. The separation line itself is the envelope of the limiting streamlines. In the present computations, inviscid velocities are obtained from an analytical solution of the potential equation. Fig. 3(a) shows the variation of \( C_f \) along the windward and leeward planes of symmetry for different \( \alpha \). Cebeci’s results are given in graphical form and a direct comparison is not given here. For this test case, \( \Delta s = 0.1 \), \( \Delta \alpha = 0.005 \), \( k = 1.12 \). From the figure, nose separation on the leeside can be seen for \( \alpha = 45^\circ \). Fig. 3(b) shows the \( C_{f0} = 0 \) line for \( \alpha = 3^\circ \). For this case, \( \Delta s = 0.1 \), \( \Delta \alpha = 6^\circ \).

The trends shown by the present computations agree well with Cebeci’s results.

Turbulent Flow over Infinite Swept Plate

In this test case, extensive measurements were made by Van den Berg and Elsenaar [22] along 10 streamwise stations on an inclined plate \( (\lambda = 35^\circ) \) over which a streamwise adverse pressure gradient was imposed by means of an adjustable roof. The free stream velocity is 35 m/sec and the Reynolds number is \( 2.4 \times 10^6 \)/m. Many turbulence models (including anisotropic models) have been proposed for explaining these measurements [7,23,24,25]. For example, Prahlad [26] proposed an improvement of the Rotta model and the Klebenoff intermit-
teny factor. In the analysis reported earlier [10], the anisotropic model of Rotta (based on local streamline co-ordinate system) was found to give better results when a value of 0.4 was assumed for $T$ (ratio of crossflow to streamwise eddy viscosity).

In the present analysis, the quasi 3-D equations have been solved using the measured inviscid pressure and flow angle distribution and the measured initial profiles at station 1 ($x = 0.52$ m) where the flow is found to be fully turbulent. In these calculations, $\Delta \xi = 0.0082$, $\Delta \eta_1 = 0.01$, $k = 1.2$, $\eta_w = 61$. The resultant velocity profiles and the cross flow angle profiles are compared with measurements in Fig.4(a) and Fig. 4(b). The resultant velocity profiles match well with the measurements, but discrepancies are noticed in the cross flow angle profiles, especially after station 6 ($x = 1.02$ m). In Figs.4 (c) to Fig.4(f), boundary layer parameters $\delta, \delta_1, \delta_2, \delta_3$ (redefined in terms of streamline co-ordinates for purposes of comparison) are compared with measurements.
Several ad-hoc models have also been proposed by tuning certain constants in the Cebeci-Smith model. For example, Johnston [7] used a small correction in the outer law of the Cebeci-Smith model. Johnston has modified the constant $\alpha$ as follows:

$$\alpha = \max [0.0084, 0.0168 - 0.016 |\beta_w|]$$

In the present calculations, it is seen that the definition of \( \delta \) (boundary layer thickness, used in the Klebenoff factor of the outer eddy viscosity law) has an influence on the solution and the smoothness of the velocity profile. Calculations have been done for $\delta$ corresponding to $u/u'' = 0.995$ as well as 0.95. In Fig.4(g) and Fig.4(h), the computed $\beta_w$ (wall shear angle) and $C_f$ (skin friction coefficient) are compared with the measured values.

It can be seen from the figures that the results for $\delta_{95}$ compare very well with the measurements (even for the isotropic case). This is due to the fact the stresses in the outer layer are brought down in both cases (by modifying $\alpha$ in one case and modifying $\delta$ in the other). The same effect was sought to be achieved in the anisotropic case by reducing the cross flow stresses. In this test case, the solution is found to be equally sensitive to minor variations in the external flowfield as well as the turbulence model. It is therefore difficult to arrive at definite conclusions based on ad-hoc modifications to the turbulence model and a more systematic approach to 3-D turbulence modelling is needed.

**Results on Transition Models**

In this section, we describe some case studies carried out using three transition models available in the literature.

**Transitional Flow Over Infinite Swept Leading Edge**

In this test case, Cumpsty and Head [27] made extensive measurements over an infinite swept cylinder ($\lambda = 60^\circ$) of radius 114 mm, in the freestream velocity range of 8 to 50 m/s. Transition was found to occur when $K_{022}$ was around 100 and when the parameter $C^* = w^+_c/u_\infty u/v$ was around $0.6 \times 10^3$, where $w_c$ is the spanwise velocity and $u_\infty$ is the streamwise velocity gradient. The
flow was found to be turbulent when $C^*$ was around $1.4 \times 10^5$.

In this case, the 3-D stagnation line equations have been solved assuming potential solution for the inviscid flow and the extended Cebeci-Smith turbulence model. Cebeci adapted the Chen-Thyson transition correlation for 3-D flows, with suitable changes [2]. For 2-D flows without pressure gradient, the intermittency can be expressed as

$$\gamma = 1 - \exp \left[ G (x - x)^2 \right]$$

where

$$G = \frac{3 \left( \frac{u_x}{v} \right)^2}{A^2} R_{x'}^{1.34}$$

Using $w_e$ in place of $u_e$, the above correlation can be rewritten in terms of Reynolds number, as

$$\gamma = 1 - \exp \left[ \frac{3}{A^2} R_{x'}^{1.34} (R_x - R_x')^2 \right]$$

For this test case, the characteristic Reynolds number $R_x$ is taken as $C^*$, with $R_{x1} = 6.6 \times 10^5$. For this test case, computations have also been carried out using the Arnal model with $R_{x1}=135$. In this case $\Delta \eta_1 = 0.01$, $k = 1.2$, $\eta_{\infty} = 61$.

In Fig.5(a) to Fig.5(d), the computed spanwise velocity profiles are compared with measurements for $C^* = 0.6 \times 10^5, 0.8 \times 10^5, 1.2 \times 10^5, 1.8 \times 10^7$. The comparison is very good. In Fig.5(e), $C_f$ values are compared with measurements for different $C^*$ values. $C_f$ values match well with the measurements.
In these computations, the quasi 3-D equations have been solved with the following boundary conditions:
\[ y = 0, \quad u = 0, \quad w = \omega r \]
\[ y \to \infty, \quad u = 0, \quad w = 0 \]

The extended Cebeci - Smith turbulence model has been used by replacing \( w \) in the inner and outer regions by \( \hat{w} = \omega r - w \). The transverse intermittency (Klebenoff) factor has been assumed to be 1. Computations are started at \( r = 0 \) (axis of rotation), by solving the stagnation point equations from suitable initial profiles consistent with the boundary conditions.

Cebeci has modified the Chen - Thyson correlation as follows (using \( w \) in place of \( u \))
\[
\gamma - 1 - \exp \left[ -G \left( \frac{R}{r} \right)^2 \right]
\]
where
\[
G = \frac{3}{A^2} \left( \frac{R}{r} \right) R^{1.54}, \quad R = \omega r^2 / v,
\]
and \( R_{tr} \) is the transition Reynolds number [29]. For this test case, computations have been done using Dey-Narasimha and Arnal models also with \( R_{tr} = 255 \). In this test case, \( \Delta \eta = 0.01, k = 1.2, \Delta r = 0.01, R_{\infty} = 0.7056 \times 10^9 m \). The computed circumferential skin friction coefficient \( C_{p} = \tau_{\infty} / (1/2 \rho r^2 \omega^2) \) is compared with the measurements of Cham and Head in Fig. 6(a). Here again, the Arnal model shows an overshoot. Chen-Thyson model seems to follow the measurements more closely compared to Dey-

Transitional Flow over a Rotating Disk

In this test case, Cham and Head [28] have made extensive measurements on a rotating disk. In these experiments, transition was found to occur when the characteristic Reynolds number \( R_r = \omega r^2 / v \) was around \( 1.85 \times 10^5 \) and the flow was found to become turbulent when \( R_r = 2.85 \times 10^5 \). Here \( \omega \) is the angular velocity of the disk.

well with the measurements in the turbulent range. Arnal's model shows an overshoot in the transition phase. From Fig.5(f), we can see that the computed \( R_{022} \) values compare very well with the measurements in the complete range of \( C^* \). In these calculations, the transverse intermittency (Klebenoff) factor has been assumed to be 1.

### Fig. 5e Infinite swept cylinder

<table>
<thead>
<tr>
<th>0.0070</th>
<th>0.0060</th>
<th>0.0050</th>
<th>0.0040</th>
<th>0.0030</th>
<th>0.0020</th>
<th>0.0010</th>
<th>0.0000</th>
</tr>
</thead>
<tbody>
<tr>
<td>10^4</td>
<td>10^5</td>
<td>10^6</td>
<td>10^7</td>
<td>10^8</td>
<td>10^9</td>
<td>10^10</td>
<td>10^11</td>
</tr>
</tbody>
</table>

### Fig. 5f Infinite swept cylinder

<table>
<thead>
<tr>
<th>1000</th>
<th>900</th>
<th>800</th>
<th>700</th>
<th>600</th>
<th>500</th>
<th>400</th>
<th>300</th>
<th>200</th>
<th>100</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>900</td>
<td>800</td>
<td>700</td>
<td>600</td>
<td>500</td>
<td>400</td>
<td>300</td>
<td>200</td>
<td>100</td>
<td>0</td>
</tr>
</tbody>
</table>

### Fig. 5c Transitional Flow on a Rotating Disk

In this test case, Cham and Head [28] have made extensive measurements on a rotating disk. In these experiments, transition was found to occur when the characteristic Reynolds number \( R_r = \omega r^2 / v \) was around \( 1.85 \times 10^5 \) and the flow was found to become turbulent when \( R_r = 2.85 \times 10^5 \). Here \( \omega \) is the angular velocity of the disk.
Narasimha model. In Fig.6(b) and Fig.6(c), the computed radial \((u/r)\) and circumferential \((v/r)\) turbulent velocity profiles are plotted against \(y/\theta_{22}\) and compared with measurements for \(R_e=1.0 \times 10^6\). The agreement is very good.

**Transitional Flow over a 1:6 Prolate Spheroid**

Kreplin and Meier [30] have made extensive measurements on a 1:6 prolate spheroid at various angles of incidence, at a free stream velocity of 45 m/s and free stream Reynolds number of \(7.2 \times 10^6/m\). Many researchers have attempted to reproduce these measurements by theoretical models. Cebeci [4] has used the \(e^n\) method (laminar instability theory) for predicting the transition line. Arnal [19] has predicted the transition line fairly well for \(U_w=10\) m/s and \(\alpha=10^\circ\). Cebeci [31] and Cousteix [32] have computed the skin friction by assuming the measured locations of transition onset and by using intermittency methods in the transition zone. Iyer [33] has computed the skin friction for \(\alpha=2.5^\circ, 5^\circ\) using the Dey-Narasimha and Arnal intermittency correlations, by assuming the measured onset of transition locations.

For this test case, computations have been carried out for \(\alpha=0^\circ\) and \(5^\circ\) using the Chen-Thyson, Dey-Narasimha and Arnal intermittency correlations, assuming the measured locations of transition onset. Inviscid flowfield is obtained from potential solution. As the above mentioned correlations are for 2-D flows, they have to be adapted for 3-D flows. For example, in Arnal’s correlation \(\theta_1\) used in place of \(\theta\). In the present computations, the Dey-

Narasimha model is used as an algebraic model. The step sizes are: \(\Delta \xi = 0.02, \Delta \eta_1 = 0.01, k = 1.2\).

In Fig.7(a), the computed wall shear is compared with measurements for \(\alpha=0^\circ\). (Here the free stream velocity is 60 m/s). The onset of transition corresponds to an \(R_e\) of 1100. Chen-Thyson and Dey-Narasimha models give good results compared with measurements. Arnal’s correlation shows an overshoot. Dey-Narasimha model gives better results when the freestream turbulence parameter \(q\) is taken to be greater than 0.2%. Fig.7(b) shows the wall shear comparison using the Dey-Narasimha model, as a linear combination and as an algebraic model. The differences are marginal. In Fig.7(c) and Fig.7(d), the computed skin friction distribution \(C_f(\tau_w/1/2 \rho U^2)\) using Arnal’s model, is compared with measurements for \(\alpha=5^\circ\) along the planes of symmetry. Chen-Thyson and Dey-Narasimha models cannot be readily extended for this case. It can be seen from the figures that the computed results show the correct trends but the discrepancies between the measurements and the numerical results are quite large in the transition zone pointing to the need for evolving better 3-D transition models.

**Concluding Remarks**

A comprehensive 3-D boundary layer code has been developed in VSSC. It has been validated against a number of test cases available in the literature. This code can be used as a basic software tool for studying flow separation patterns as well as for studying engineering turbulence and transition models. This code can also be used in the design
of aircraft, ships, rockets and turbo machinery. Studies carried out using this code indicate that simple extensions of 2-D turbulence and transition models to 3-D flows give fairly good results. From the limited studies, the following observations are made:

The extended Cebeci-Smith turbulence model gives good results, compared to the measurements. Arnal’s transition model can be readily extended for 3-D flows, but the results show an overshoot compared to measurements.

Dey-Narasimha and Chen-Thyson models may be adapted for certain types of axisymmetric and 3-D flows.

Acknowledgement

This study was inspired by Dr T S Prahlad, National Aerospace Laboratories, Bangalore (formerly of VSSC, Trivandrum). Prof A Prabhu, Indian Institute of Science, Bangalore, also showed keen interest in this study. We are happy to present this paper in their honour.

References


APPENDIX 1

Special Forms of the Boundary layer Equations

The 3-D boundary layer equations exhibit singularity for certain flow conditions and hence require special treatment. The following are some of the special forms [10,11] we come across frequently.

Streamwise Attachment Line Equations

Typically seen along a plane of symmetry for flows with incidence or along wing - fuselage junction, where the circumferential or spanwise velocity is zero and the streamwise velocity is nonzero.

Here \( w = 0, \ w_{\omega} \neq 0 \quad K_2 = 0 \).

To remove the singularity, the continuity and momentum equations are differentiated with respect to \( \omega \)

The stream functions are chosen as

\[
\Psi = \rho \ h_2 \ u, \quad \Phi = \rho \ h_2 \ w_{\omega},
\]

\[
h_1 \ h_2 \ \mathbf{\bar{v}} = h_1 \ h_2 \ (\rho \ v)_{w} - \Psi - \Phi
\]

The following transformations are made

\[
d \eta = \left( \frac{u_e}{p_e \ u_e} \right) \rho \ d \xi
\]

\[
\Psi = (\rho_e \ u_e \ h_2) \xi^{1/2} \ h_2 \ f(\xi, \eta)
\]

\[
\Phi = (\rho_e \ u_e \ h_2) \xi^{1/2} \ h_1 \ \frac{w_{\omega}}{u_e} \ g(\xi, \eta)
\]

where \( f' = \frac{u}{u_e}, \ g' = \frac{w_{\omega}}{w_e}, \ \theta = \frac{H}{H_e} \)

3-D Stagnation Line Equations

Typically seen along swept blunt leading edges, where the chordwise velocity is zero and spanwise velocity is nonzero.

(Here \( u = 0, \ w \neq 0, \ h_1 = 1, \ h_2 = 1 \))
The equations are differentiated with respect to $\xi$

The following stream functions are chosen

$$\Psi_\xi = \rho h_2 u \xi; \quad \Phi_\xi = \rho h_1 w;$$

$$h_1 h_2 \bar{\rho} \nabla w = (\rho \bar{\nu}) w, h_1 h_2 - (\Psi + \Phi)$$

The following transformations are made

$$d \eta = \left( \frac{u}{\rho_e \mu_e} \right)^{1/2} \rho d \xi$$

$$\Psi = (\rho_e \mu_e u \xi)^{1/2} h_2 f(\omega, \eta)$$

$$\Phi = (\rho_e \mu_e u \xi)^{1/2} h_1 \left( \frac{w}{u \xi} \right) g(\omega, \eta)$$

The 3-D Stagnation Point Equations

Normally encountered in the case of blunt bodies, where all the velocity components are zero. Here

$$u = 0, \quad w = 0, \quad h_1 = 1, \quad h_2 = 0, \quad h_2 \neq 0$$

To remove the singularity, the continuity and momentum equations are differentiated with respect to $\xi$

The stream functions are chosen as

$$\Psi_\xi = 2 \rho h_2 u \xi; \quad \Phi_\xi = \rho h_1 w \xi;$$

$$h_1 h_2 \bar{\rho} \nabla w = - (\Psi + \Phi_\omega) + h_1 h_2 (\rho \nabla) w$$

The following transformations are used

$$d \eta = \left( \frac{u}{\rho_e \mu_e} \right)^{1/2} \rho d \xi$$

$$\Psi = 2 (\rho_e \mu_e u \xi)^{1/2} h_2 f(\omega, \eta),$$

$$\Phi = (\rho_e \mu_e u \xi)^{1/2} h_1 \left( \frac{w}{u \xi} \right) g(\omega, \eta)$$

where $u \xi / \mu_e = f$, $w \xi / u \xi = g$, $H / H_e = \theta$

Quasi 3-D Equations

Typically seen in the case of infinite swept wings or rotationally symmetric flows, where the flow parameters are assumed to be constant in the spanwise or circumferential direction.

For 2-D as well as Axisymmetric Flows,

$$u_e \neq 0, \quad w_e = 0, \quad (y_e = u_e, \quad z_e = u_e)$$

For Quasi 3-D flows,

$$u_e \neq 0, \quad w_e \neq 0 \quad (y_e = u_e, \quad z_e = w_e)$$

For rotating flows,

$$u_e = 0, \quad w_e \neq 0 \quad (y_e = z_e = w_e)$$

The following stream functions are chosen

$$\Psi_\xi = \rho h_2 u \xi, \quad \Phi_\xi = \rho h_1 w \xi, \quad h_1 h_2 \bar{\rho} \nabla w = h_1 h_2 (\rho \nabla) w - \Psi_\xi$$

The following transformations are made

$$\Psi = (\rho_e \mu_e y \xi)^{1/2} h_2 f(\xi, \eta)$$

$$\Phi = (\rho_e \mu_e y \xi)^{1/2} h_1 g(\xi, \eta)$$

$$d \eta = \left( \frac{y}{\rho_e \mu_e \xi} \right)^{1/2} \rho d \xi$$

APPENDIX-2

Extended Cebeci-Smith Turbulence Model

The most widely used engineering turbulence model is the Cebeci-Smith model. It is an algebraic two layer eddy viscosity model. Cebeci has extended this model for 3-D flows and has included low Reynolds number effects. The extended model equations are given below:[2]

Inner Law

$$\varepsilon_\xi = L^2 \left[ \frac{u^2}{\varepsilon} + \frac{w^2}{\varepsilon} \right]^{1/2} \gamma_{tr}$$

where $L = k \xi [1 - \exp (-\xi^2 / A)]$
\[ A = \frac{A^*}{N_{\tau}} \left( \frac{\rho_u}{\rho_w} \right)^{1/2}, \quad \xi = \frac{u'_x}{u}, \quad \chi = \left( \frac{\tau_w}{\rho_w} \right)^{1/2} \]

For low Reynolds numbers, \((Re < 6000)\) the constants \(k^*\) and \(A^*\) are modified as

\[ k^* = 0.4 + 0.19/(1 + 0.49 \zeta^2), \quad A^* = 26 + 14/(1 + \zeta^2) \]

where

\[ \zeta = R_0 \times 10^3, \quad \z_2 = 0.3, \quad \text{if} \quad R_0 < 300. \]

**Outer Law**

\[ \epsilon_0 = \alpha \left( 1 - \frac{(u_x - (u^2 + w^2)^{1/2})}{d \zeta} \right) \gamma_k \gamma_t \]

where \(\alpha = 0.0168 (R_0 > 6000)\)

For low Reynolds numbers, \(\alpha\) is modified as

\[ \alpha = \alpha_0 \frac{(1 + \pi_0)}{(1 + \pi)} \quad (425 < R_0 < 6000) \]

\[ \alpha = 10^{-3} (194.8 - 128 \log_{10} R_0) \]

\[ + 30.925 \log_{10} R_0 - 2.475 \log_{10} R_0 \] (\(R_0 < 425\))

where \(\alpha_0 = 0.0168, \quad \pi_0 = 0.55, \quad z_1 = R_0/425 - 1\)

\[ \pi = 0.55 \left[ 1 - \exp \left( -0.243 z_1^{1/2} - 0.298 z_1 \right) \right] \]

\(\gamma_k\), is the Klebenoff transverse intermittency factor and is given by

\[ \gamma_k = \left[ 1 + 5.5 \left( \frac{\zeta}{\delta} \right)^2 \right]^{-1} \]

where \(\delta\) is usually taken as that value of \(\zeta\) at which the resultant velocity is 99.5% of the external velocity. Change over from the inner law to the outer law is effected at the intersection.

\(\gamma_t\) is the longitudinal intermittency factor associated with transition.

**APPENDIX - 3**

**Solution Procedure**

The transformed boundary layer equations can be compactly written in vector form (as a system of 8 first order non-linear coupled equations) as

\[ \bar{Y}' = F(\bar{Y}) \]

where \(\bar{Y} = (f, f', f'', g, g', \theta, \Theta)^T\)

The boundary conditions are

At \(\eta = 0, (f, f', f'', g, g', \theta, \Theta)^T = [0 0 0 0 \theta^e]^T\)

As \(\eta \rightarrow \eta_{in}, [f', f'', \theta]^T = [1 1 1]^T\)

The difference equations are written centred at \(\xi_i - 1/2, \eta_j - 1/2, \omega_k - 1/2\). When the cross flow is negative, (i.e. \(w_{j-1/2} < 0\)) the centering is done at \(\xi_i - 1/2, \eta_j - 1/2, \omega_k\).

The non-linear difference equations can now be written in the form

\[ \bar{Y}_{i-1/2, j-1/2} - \bar{Y}_{i-1/2, j-1/2} = \Delta \eta \bar{F}(\bar{Y}_{i-1/2, j-1/2}), \quad \text{for} \quad j = 1, 2, \ldots, J \]

where \(\bar{Y}_{i-1/2, j-1/2}\) is approximated as

\[ \frac{1}{4} \left[ \bar{Y}_{i-1, j} + \bar{Y}_{i, j-1} + \bar{Y}_{i-1, j-1} + \bar{Y}_{i, j} \right] \]
\( \bar{Y}_{j}^{i,k} \) \( \bar{Y}_{j}^{i,k} \) is approximated as \( \frac{1}{2} \left[ \bar{Y}_{j}^{i,k} + \bar{Y}_{j}^{i,k} \right] \)

The above non-linear system is solved by a Newton-Raphson procedure, starting from an initial profile \( Y_{j}^{0} \) \( j=1,2,\ldots,J \) and expanding \( F(Y) \) as a Taylor series about \( Y^{0} \) and considering the first order term. The resulting linear system becomes (after dropping the superscripts \( i \) and \( k \) for convenience),

\[
R_{j} \delta \bar{Y}_{j} - L_{j} \delta \bar{Y}_{j-1} = \bar{Y}_{j} \quad (j = 1,2,\ldots,J)
\]

where

\[
L_{j} = \Delta \eta_{j}^{-1} I + \frac{1}{2} \left[ \frac{\partial F}{\partial \bar{Y}} \right]_{j-1/2}
\]

\[
R_{j} = \Delta \eta_{j}^{-1} I - \frac{1}{2} \left[ \frac{\partial F}{\partial \bar{Y}} \right]_{j-1/2}
\]

\[
\delta \bar{Y}_{j} = \bar{Y}_{j} - \bar{Y}_{j}^{0}, \quad \gamma_{j} = -F' \left( \bar{Y}_{j-1/2}^{0} \right)
\]

\[
(A-1)
\]

\( I \) is the unit matrix and

\[
\frac{\partial F}{\partial \bar{Y}} \]

is the Jacobian of \( F \) w.r.t \( \bar{Y} \).

The boundary conditions now become,

\[
\left[ \delta f_{0}, \delta f'_{0}, \delta g_{0}, \delta g'_{0}, \delta \theta_{0} \right]^{T} = [0, 0, 0, 0, 0]^{T}
\]

\[
\left[ \delta f_{j}, \delta g_{j}, \delta \theta_{j} \right]^{T} = [0, 0, 0]^{T}
\]

\[ (A-2) \]

When the equations (A-1) are used along with (A-2), we have after re-ordering the variables, a block ( \( 8 \times 8 \) ) tri-diagonal matrix equation of order \( J \)

\[
A \delta \bar{Y}_{j} = \gamma_{j}
\]

\[ (A-1) \]

The above block matrix equation is solved using the standard LU decomposition procedure.