DESIGN METHOD FOR PROBLEMS INVOLVING SHOCK BOUNDARY-LAYER INTERACTION

Anthony George Thomas Cross
BAE SYSTEMS, Warton Aerodrome, Warton
Lancashire, United Kingdom

Keywords: *supersonic flow, shock boundary layer interaction*

Abstract

Three-dimensional boundary layer theory has been applied to the air inlet design problem studied by GARTEUR Action Group 34. The essential part of the problem is a stream surface that passes through a conical shock, as generated by a right circular cone in supersonic flow. Ahead of the shock the stream surface represents a flat plate. Downstream of the shock the stream surface involves a bump of cross section that tends towards that of the cone surface, as the conical flow extends toward infinity.

Boundary layer calculations indicate that downstream of the shock, flow divergence is sufficient to overcome the effects of adverse pressure gradient on the forward part of the bump. Thus flow assessment can focus on the shock boundary layer interaction.

It is shown that the boundary layer condition at the shock is largely determined by the attitude of the incoming inviscid flow to the projection of the shock on the plate. The attitude of the inviscid flow together with the velocity jump through the shock determines the lateral inviscid streamline deflection within the local stream surface. The streamline deflection is a function of distance from the bump centre line and it is convenient to examine the boundary layer state as a function of streamline deflection or shock sweep.

Ahead of the shock the boundary layer is two-dimensional, with limiting streamlines parallel to the free stream. Through the shock these surface streamlines are deflected to a greater extent than for inviscid flow and rotate away from the bump centreline. Providing inviscid deflection is less than a critical value, boundary layer calculation indicates attached flow. For greater values, limiting streamlines rotate towards the shock plane and the boundary layer solution shows singular behaviour associated with separation.

Above the critical value, conditions downstream of the shock remain capable of supporting attached flow and so appear conducive to reattachment. Thus calculations appear consistent with formation of a vortex that develops along the shock foot from an identifiable critical point.

The method can be more generally used for flows involving shock boundary layer interaction. It is particularly suited for flows where shocks can be determined using Euler or shock-expansion theory methods. The resulting shock solution determines the inviscid stream deflection, so providing an important parameter for viscous calculations.

1 Introduction

Aerodynamic design often involves high Reynolds number turbulent flow, featuring shock boundary layer interaction and separation. Three-dimensional flow separation can result in vortex generation and the control of shocks and vortices often features in design. Accurate prediction of these flows is still a major challenge for CFD as the need for fine field grids, to capture the boundary layer,

***Author**: AGT Cross*
BAE SYSTEMS, Warton Aerodrome, Warton,
Lancashire, PR4 1AX, United Kingdom.

results in intensive computing requirements. The required fineness of the field grid increases with Reynolds number, such that modelling error is an issue when using computational methods to analyse Reynolds number effect. Thus to progress designs efficiently, application of Navier-Stokes methods needs to be supported by use of simpler methods that adequately represent the effects of shocks, separations and vortices.

Even when the upstream character of a boundary layer is two-dimensional, if a shock intersection is oblique to the advancing front of the layer its downstream development is three-dimensional. Thus, for example, while two-dimensional methods can be used with some success for the flow on wings, three-dimensional methods are more generally required when shocks are involved. For this reason the capability considered here is based on three-dimensional boundary layer theory.

Here a three-dimensional boundary layer method has been applied to the shock boundary layer interaction problem represented through consideration of a stream surface passing through a particular case of fully supersonic conical flow. A GARTEUR action group studied this flow as it involves a conical shock and isentropic recompression, making it suitable for air inlet applications as described by Seddon and Goldsmith [1]. Through application of the method considered in this paper, key features of the flow have been identified and quantified in a manner that allows improved designs to be considered. The key features relate directly to the three-dimensional nature of the boundary layer and the conditions for separation. It is reasoned through wider applicability of the method that it can be used for more general shock boundary layer interaction problems.

2 Design Problem and Analysis Method

2.1 GARTEUR test case

The test case studied by GARTEUR Action Group 34 is described by Bradbrook [2] and involves geometry that, for an appropriate supersonic Mach number and inviscid flow condition, generates a simple conical shock wave and supersonic flow solution. Upstream of the shock the flow is uniform, while downstream the flow is subject to isentropic compression.

While conical flow can be generated using a cone, by choosing a stream surface that passes through conical flow, other geometry can be defined that can be used to generate such flow. However for any real application a boundary layer develops on the defined surface. Thus, for the solution to remain valid the boundary layer must be thin requiring high Reynolds number and/or the stream surface must be corrected for viscous displacement.

The GARTEUR test case involves a free stream, of Mach number $M_\infty = 1.8$, set parallel to the 'x' direction and axis of a right circular cone having a semi vertex angle, α, of 23 degrees. The particular stream surface chosen involves a flat plate ahead of the shock generated by the cone apex. The apex is at the origin of a Cartesian x, y, z, co-ordinate system and the plate is offset from the cone axis, in the 'z' direction. The intersection of the plate and shock provides the starting front for tracing the downstream development of the stream surface and this results in a semi infinite, symmetric, bump that rises from the plate. The characteristic dimension of the geometry that uniquely defines the bump is the height, h, of the plate above the cone axis.

As the bump is semi infinite, for the GARTEUR test case it is truncated at down-stream location $x/h = 4$ and terminated with an after body to produce a finite bump suitable for a real intake. Thus, the flow over the rear of the GARTEUR bump is not conical. However the key features of the problem are the shock boundary layer interaction and boundary layer state during compression, so here it is only necessary to consider the forward conical element of the test case.

For a given cone angle and free stream Mach number two possible conical solutions can exist, as considered by Ferri [3]. The weak shock solution can involve fully supersonic flow downstream of the shock while the strong shock solution can involve fully subsonic downstream flow. With reduced Mach number the two solutions approach each other and for Mach numbers below this condition conical flow ceases and the shock detaches from the cone apex.

The GARTEUR test case conditions are summarised in table 1 and are appropriate to weak shock solution. Particular solution is determined by the flow backpressure and so intake operating condition. In table 1 ψ is the shock angle relative to the x axis.

Parameter	Value
M_∞	1.8
α	23°
ψ	43.92°
R_h	$5x10^4 < R_h < 5x10^7$

Table 1. Test case conditions

Figure 1a shows the conical element of the GARTEUR test case in terms of sectional cut through the bump on the symmetry plane, $y/h=0$. The figure shows the equivalent cone surface, stream surface and shock.

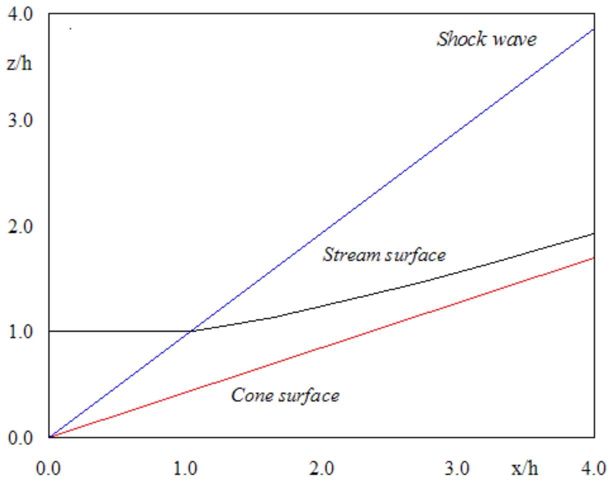


Figure 1a. Key flow features as viewed on symmetry plane

Figure 1b shows front views of the bump in terms of a series of sectional cuts. These cuts are normal to the free stream and plate. The geometry for the solution can be represented by a hyperbolic approximation, as reported by Seddon and Goldsmith [1] and attributed to Bower et al [4]. The approximation is very accurate for the bump considered here and is represented by the following parametric equations,

$$r^2 = x^2 \tan^2 \alpha + h^2\left(1-\tan^2\alpha/\tan^2\psi\right)\sec^2\phi \qquad (1)$$

$$y = r\sin\phi\,, \qquad z = r\cos\phi$$

Figure 1c shows a top view of the intersection of the upstream stream surface with the shock, so indicating the footprint of the bump on the plate. The intersection is precisely defined by the geometry of the shock cone and relative position of the plate, so is described by the hyperbolic curve,

$$y^2 = x^2\tan^2\psi - h^2 \qquad (2)$$

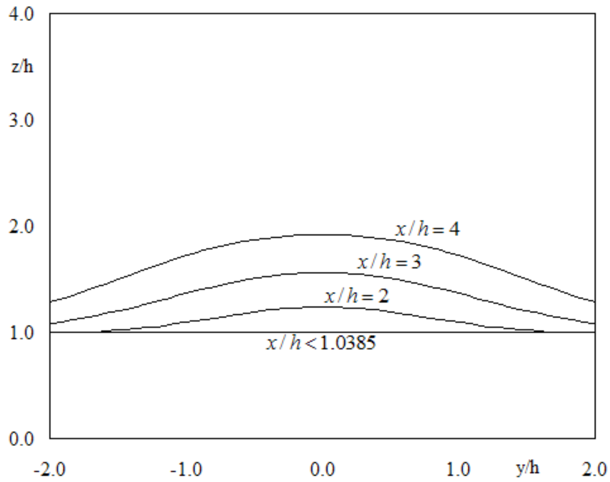


Figure 1b. Sections through bump at constant values of *x/h*

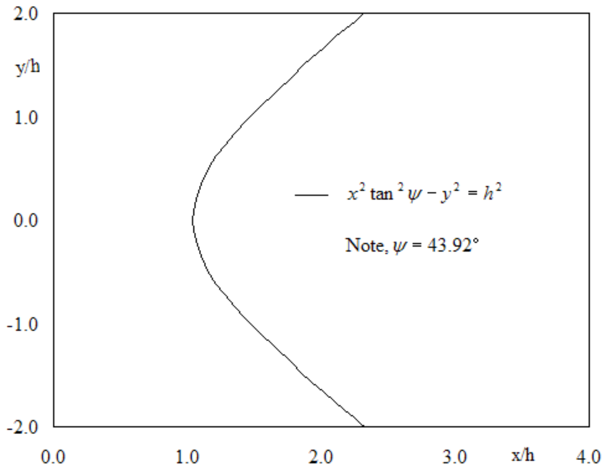


Figure 1c. View of bump defined by shock intersection with flat plate in plane *z/h=1*

For the GARTEUR test case, Bradbrook [2] obtained a Navier-Stokes solution corresponding to a Reynolds number based on plate height of $R_h = 7.87x10^5$ and for a value of boundary layer displacement thickness at the shock foot of $0.07h$. This solution involved wall functions that compromised accuracy in the near wall region but reduced computational requirements to enable a practical calculation.

For the boundary layer calculations considered here, conditions have been chosen appropriate to a small-to-medium size aircraft, for which $h = 0.1m$, operating at an altitude of $11{,}000m$ within an International Standard Atmosphere. This corresponds to a Reynolds number of $R_h = 1.36x10^6$.

2.2 Conical flow calculation

By definition, conical flow properties are constant along radials emanating from the cone apex. For the general case, the resulting form of the Euler equations are provided by Sears [5] in a convenient spherical co-ordinate system. For the right circular cone with axis parallel to the free stream, the conical flow is symmetric about the cone axis and the Euler equations can be reduced to a one-dimensional form in terms of angular displacement from the axis. This approach allows weak and strong shock solutions to be calculated to high precision.

The equations are solved iteratively as it is first necessary to estimate the shock angle. From the assumed shock position and free stream state the appropriate Rankine-Hugoniot shock jump conditions are calculated so allowing the Euler equations to be solved between the shock and cone surface. The flow tangency condition at the cone surface is then checked and the shock angle adjusted if necessary. If correction is needed the whole process is repeated until convergence is achieved.

Once a converged solution is obtained, the upstream stream surface can be tracked through the solution field in order to generate the bump stream surface and associated flow. This approach enables a computationally efficient and accurate analysis.

2.3 Boundary layer calculation

For air vehicle applications supersonic flow occurs at conditions of high Reynolds number so boundary layers are thin. Thus, the inviscid solution provided by conical flow theory is applicable to the forward part of the GARTEUR bump. Similarly this solution can be used as data for calculating the turbulent boundary layer development. However, as the flow downstream of the shock intersection line is three-dimensional, it is necessary to make use of a three-dimensional boundary layer method.

The boundary layer method of references [6] and [7] is considered here, where [7] considers specific application to supersonic high Reynolds number flow. This integral method solves the entrainment, stream-wise momentum and cross-flow momentum equations using a three-dimensional form of logarithmic velocity profile. As a consequence of the use of the logarithmic profile and compressibility transformations, the solution variables are equivalent incompressible values of the stream wise and cross flow components of friction velocity and the boundary layer thickness. The friction velocity is also made non-dimensional using the actual velocity at the outside edge of the boundary layer. The three solution variables are respectively designated q'_s, q'_c and δ'. Once the development of these variables has been calculated, the full boundary layer solution can be calculated from the equations of the velocity profile. Important for assessing the state of the three-dimensional boundary layer are the equivalent incompressible shape parameter, $\overline{H}$, the skin friction coefficient, C_f, and the angle of the surface streamline relative to the streamline at the outer edge of the boundary layer, β_0. The latter parameter, usually referred to as the limiting streamline angle, is calculated from the local direction of the friction velocity,

$$\beta_0 = \tan^{-1}\left(q'_c / q'_s\right) \tag{3}$$

This parameter is a measure of lateral distortion of the boundary layer and so important when considering three-dimensional aspects of shock boundary layer interaction.

Local skin friction coefficient is also calculated from the definition of the friction velocity but due to compressibility involves the local total to static temperature ratio, T_{te}/T_e, at the outer edge of the layer. Thus,

$$C_f = 2\left(q_s'^2 + q_c'^2\right)\left(T_{te}/T_e\right)^{-0.5} \tag{4}$$

The boundary layer method requires the stream surface to be defined by a two-dimensional array of surface points. The points can be set in some general non-orthogonal coordinate system but need to be smoothly distributed in order to minimise numerical error. The best approach involves longitudinal distributions that march along streamlines and lateral distributions that lie on isobars.

2.4 Method of analysis

Figures 2a, 2b and 2c show part of the non-orthogonal surface grid at which the conical flow was calculated to produce data for calculating the turbulent boundary layer.

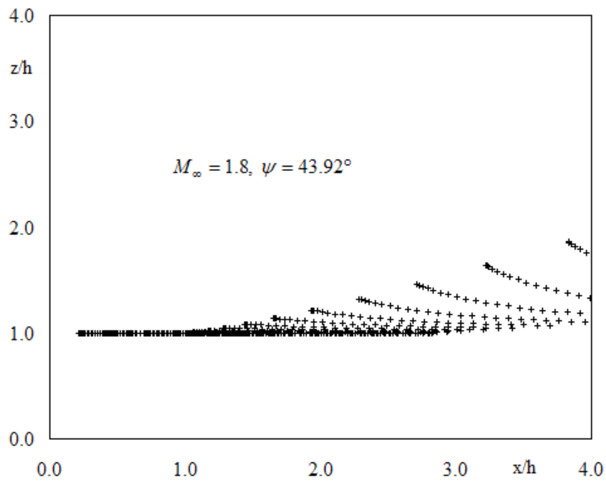


Figure 2a. Side view of grid points on stream surface

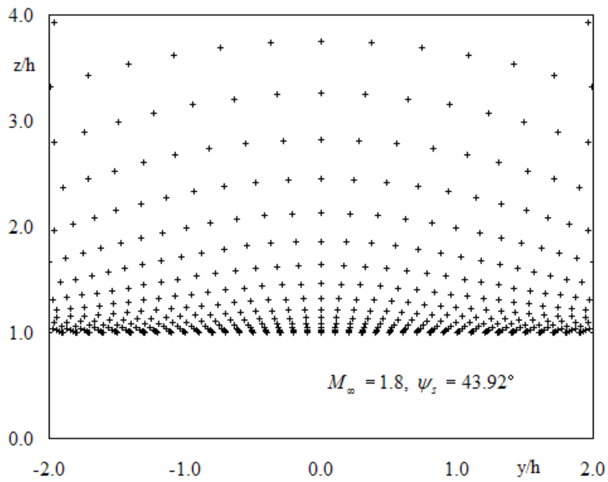


Figure 2b. Front view of grid points on stream surface

The grid points are aligned along surface streamlines and in the vicinity of the shock are concentrated in order to capture adequately the pressure rise through the shock. The grid points also lie on span-wise generators that follow the shock front at the bump leading edge.

When considering figure 2b it should be noted that the grid points do not lie on lines of constant x, so unlike figure 1b the figure does not convey the true sectional shape of the bump. Figure 2c shows that all the streamlines commence from $x = 0$ and this was chosen to provide a convenient place for prescribing the initial conditions of the boundary layer. Upstream of the shock, on the plate, the boundary layer is two-dimensional and so constant values were applied at the upstream boundary. The initial conditions used for the boundary layer are provided in table 2.

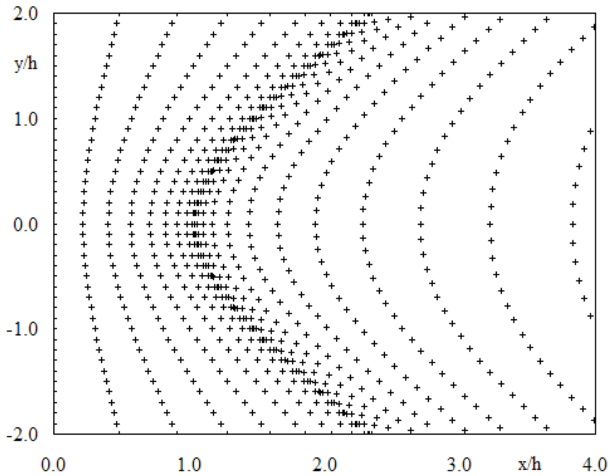


Figure 2c. Plan view of grid points on stream surface

For these initial conditions the full boundary layer development was calculated but here it is useful to focus on two key areas in order to demonstrate the nature of the design problem and value of the design method. The first area is in the region of the centreline of the bump and characterised by the boundary layer development on the symmetry plane. The second area is away from the centreline where three-dimensional effects are important and where it is necessary to focus on consideration of the possibility for boundary layer separation along the shock front.

Parameter	Value
x	0
R_θ	26,130
$\overline{H}$	1.265
β_0	0

Table 2. Boundary layer initial conditions

3 Discussion and Results

3.1 Symmetry plane results

Figure 3a shows the pressure coefficient distribution on the symmetry plane and clearly indicates the pressure rise through the shock, located at $x/h = 1.0385$. Also evident is the adverse isentropic pressure rise, downstream of the shock that shows the pressure coefficient to rise asymptotically towards the cone surface value of $C_p = 0.4441$. By $x/h = 4$, which corresponds to the end of the conical flow section of the GARTEUR test case, the pressure coefficient is quite close to the cone surface value. Upstream of the shock the pressure coefficient is at the free stream reference value.

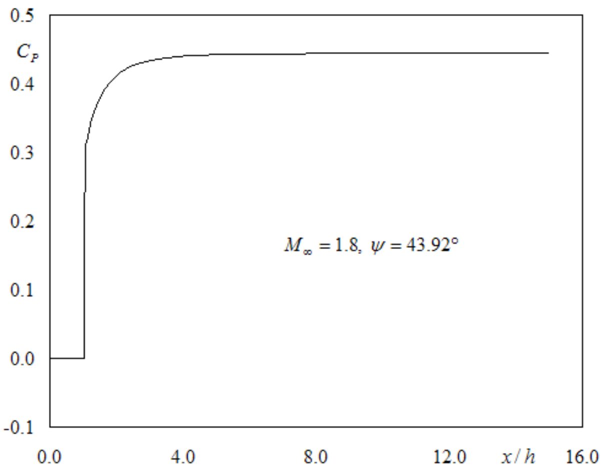


Figure 3a. Pressure coefficient on symmetry plane

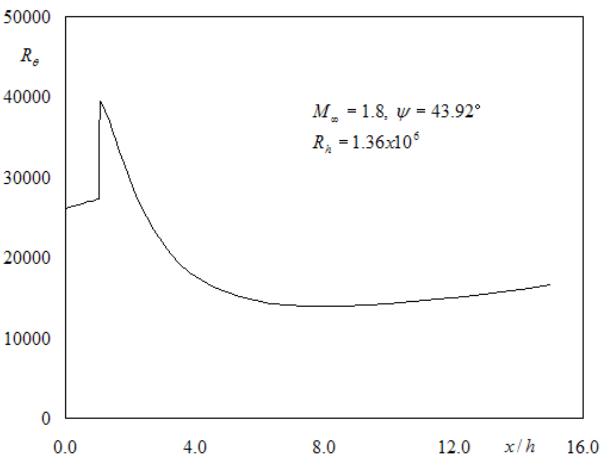


Figure 3b. Momentum thickness Reynolds number on symmetry plane

Figure 3b shows the development of the Reynolds number based on momentum thickness, R_θ. Ahead of the shock this quantity increases at an almost linear rate. Through the shock there is a sudden increase of the Reynolds number and this is followed by a reduction to a minimum value of $R_\theta = 13900$ at $x/h = 7.5$. The Reynolds number then starts to increase again at a very modest rate, but remains well below the initial value at the plate leading edge. The reduction of the Reynolds number downstream of the shock is unlike that experienced in two-dimensional adverse pressure gradient flow. The results experienced here are due to the flow diverging laterally as it moves in a radial direction away from the cone axis and can be referred to as a radial divergence effect. Thus while the momentum thickness reduces the lateral extent of the boundary layer increases such that there is an overall degradation in momentum.

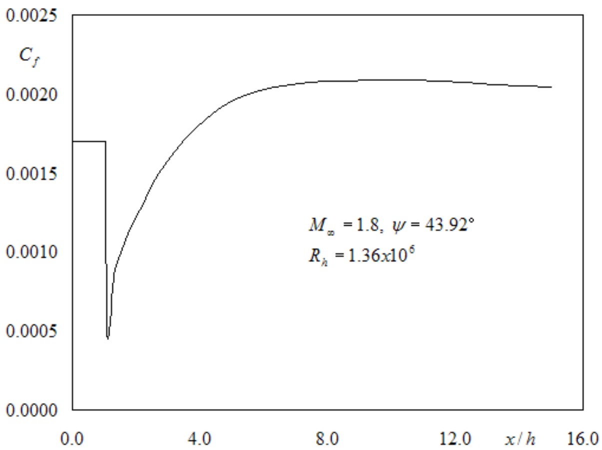


Figure 3c. Skin friction coefficient on symmetry plane

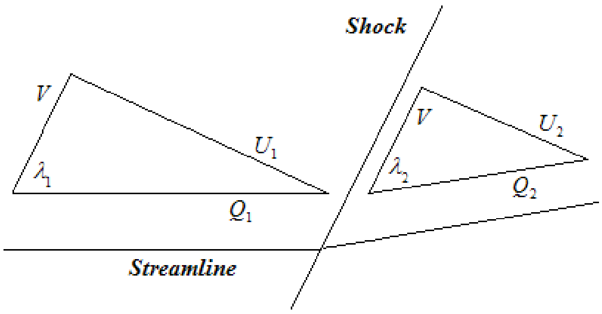
On the plane of symmetry the state of the boundary layer can be judged by the shape parameter, $\overline{H}$, and skin friction coefficient, and the development of the latter parameter is shown in figure 3c. The skin friction falls through the shock to a value of $C_f = .00045$ before increasing again to surpass the pre shock value. For this case, these results indicate that at least on the plane of symmetry the strength of the shock is insufficient to cause separation. The results also show that once through the shock, radial divergence in the boundary layer prevents the risk of separation in the region of adverse pressure gradient caused by isentropic pressure recovery.

For an air intake the consideration of separation will limit the pressure recovery achieved at the design condition. However from the calculations on the plane of symmetry it can be seen that the favourable effects of radial divergence limit the possibilities for separation and so the primary requirement is to ensure that the boundary layer negotiates the shock without separation.

3.2 Shock locus results

So far, it is demonstrated that for this class of design problem the primary consideration must be for assessing the boundary layer development through the shock. Away from the centre line, three-dimensional cross flow effects become important and so it is necessary to consider the more general boundary layer development through the shock front as illustrated in figure 1c.

The development of the boundary layer through the shock is determined by the conical flow pressure distribution and in figure 3a the condition on the symmetry plane has already been considered. Away from the symmetry plane the pressure rise through the shock remains constant but there develops a span-wise pressure gradient as the shock becomes oblique to the flow on the plate. This affects the direction of flow that impacts on three-dimensional boundary layer development and so it is convenient to consider the velocity change at the shock. Sketch 1 shows upstream and downstream velocities on the stream surface at some general point on the shock footprint away from the symmetry plane.



Sketch 1. Flow at shock in stream surface plane

In the sketch the flow is viewed on a plane that lies in the stream surface, as this is the plane in which the three-dimensional boundary layer is calculated. Ahead of the shock the plane lies on the plate, while downstream it is in the surface of the curved bump. In this boundary layer computing plane each inviscid surface streamline is deflected laterally as it passes through the shock. Through the shock the velocity component, V, in the shock plane is conserved, while the change in the component, U, normal to the shock is determined by the Rankine-Hugoniot conditions. The streamline deflection at the shock is determined by these conditions and with reference to the sketch is given by,

$$\omega = \lambda_1 - \lambda_2 \qquad (5)$$

With reference to the sketch it can be seen that at the shock,

$$Q_1 Cos\lambda_1 = Q_2 Cos\lambda_2 = V$$

Thus generally on a stream surface at a shock,

$$\omega = \lambda_1 - \cos^{-1}[(Q_1/Q_2)\cos\lambda_1] \qquad (6)$$

While from the equation for the shock footprint on the flat plate, for the right conical flow problem the angle subtended by the shock and the incoming stream direction is given by,

$$\tan\lambda_1 = \frac{dy}{dx} = \frac{x}{y}\tan^2\psi$$

Which after again using the footprint equation for the elimination of x, gives,

$$\lambda_1 = \tan^{-1}\left[\left(1+(h/y)^2\right)^{1/2}\tan\psi\right] \qquad (7)$$

This demonstrates for the GARTEUR test case that the streamline deflection through the shock is just a function of the non-dimensional lateral distance, y/h, from the plane of symmetry. The deflection has been calculated over the range $0 < y/h < 1$ and the results are presented in figure 4. At the plane of symmetry the lateral streamline deflection is zero and rises to 6.628° at $y/h = 1$, while as y/h tends towards infinity the streamline deflection asymptotically approaches the limiting value of 9.888°.

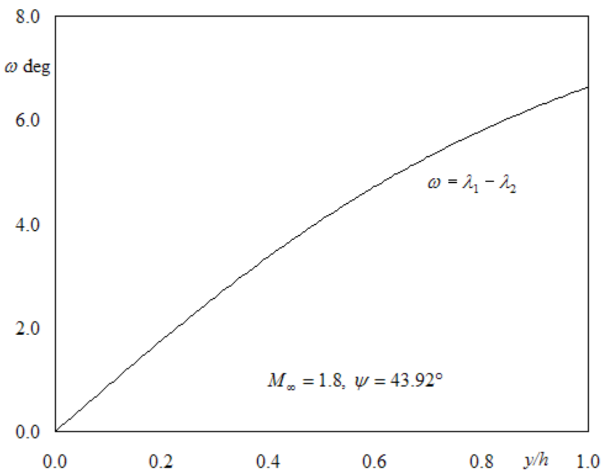


Figure 4. Inviscid streamline defection at shock

The pressure rise and other inviscid flow property changes through the shock are constant and determined solely by the free stream Mach number and equivalent cone angle. Thus for the GARTEUR test case, the streamline deflection angle is the primary independent variable for the boundary layer development through the shock. It is therefore convenient for all other independent variables to be held constant, in order to assess the impact of the primary variable. Thus the boundary layer was calculated using revised initial conditions at the shock and involved some convenient rounding to minimal significant figures of the earlier values. The revised values are recorded in table 3.

Parameter	Value
R_θ	20,000
$\overline{H}$	1.3
β_0	0

Table 3. Revised shock entry conditions

As the calculation was only required to cover the pressure rise through the shock, a much finer mesh could be considered than that described earlier for calculating the full boundary layer solution and needed for providing results on the plane of symmetry. A finer mesh was required in order to avoid numerical accuracy being an issue. To improve accuracy this approach was complemented further, by treating the shock as a discontinuity and integrating the boundary layer equations with respect to velocity rather than stream-wise displacement. Thus for the flow through the shock front a more rigorous analysis was considered than for the results shown earlier.

It was considered convenient to present the results in terms of the incremental change of the boundary layer parameters through the shock in order to show how these vary with the streamline deflection, which in itself is an incremental effect of the shock. Thus figure 5a shows how the increment in the momentum thickness Reynolds number, ΔR_θ, varies with streamline deflection. For deflections of say less than three degrees the increase is very similar to the result for $\omega = 0$, which corresponds to conditions on the bump plane of symmetry. Above three degrees there is a notable increase in the Reynolds number and above five degrees the increase becomes asymptotic and indicative of some limit. Figure 5b shows the reduction in skin friction coefficient through the shock and reveals similar characteristics.

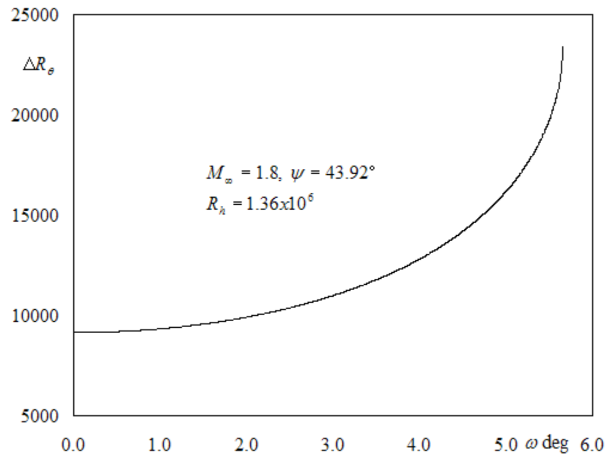


Figure 5a. Jump in momentum thickness at shock

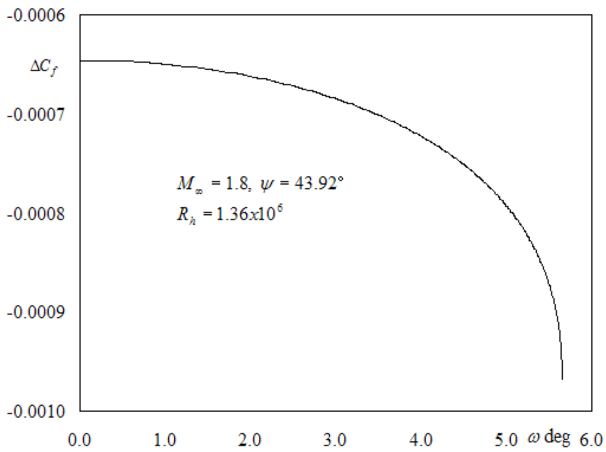


Figure 5b. Jump in skin friction coefficient at shock

Particularly interesting are the results of figure 5c, which shows the variation of the limiting streamline angle as a function of the inviscid stream deflection. Also included in the figure is the variation of λ_2 as this represents a limit for the boundary layer calculation, which involved a direct solution technique. On the plane of symmetry the limiting streamline angle is zero and this corresponds to the surface layer of the boundary layer being in the direction of the local inviscid streamline. As the limiting streamline increases, so the surface layer of the boundary layer turns towards the direction of the shock front which is the non-orthogonal lateral computing axis. Thus when $\beta_0 = \lambda_2$ the surface flow is entirely in the direction of the non-orthogonal axis and the boundary layer is highly skewed across its thickness. The significance of this condition is that it indicates when the boundary layer equations, in their direct form, become singular and this form of behaviour is associated with the onset of three-dimensional, boundary layer separation.

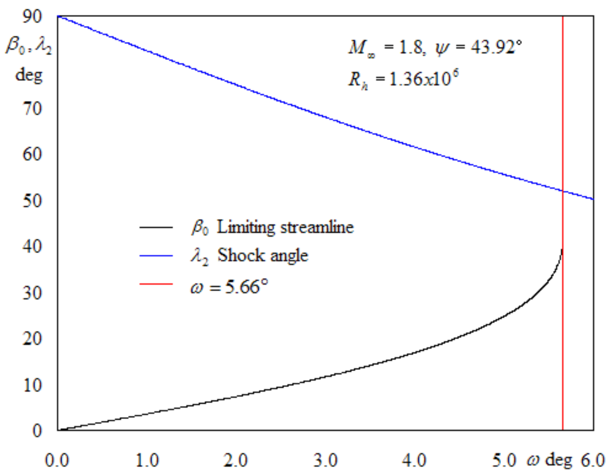


Figure 5c. Jump in limiting streamline angle at shock

From figure 5c, it can be seen that for inviscid streamline deflections of less than around three degrees the limiting streamline angle is below ten degrees. This indicates little skewing of the boundary layer with conditions close to that of the flow on the bump centre line. For higher inviscid streamline deflections the limiting streamline angle increases and the limiting condition is reached at an inviscid streamline deflection value of 5.66°, suggesting that this is the limit for attached flow for the GARTEUR test case at the conditions considered. As the flow downstream of the shock is conducive to recovery within the boundary layer this further suggests that reattachment of the boundary layer is possible on the bump surface but this is beyond the analysis considered here. If reattachment does occur then this would indicate that, as the separation is three-dimensional, it is likely associated with the formation of a vortex.

3.3 Design parameters

From the previous results and analysis it was demonstrated that the inviscid stream deflection through the shock, ω, is important when considering the shock boundary layer interaction and the likelihood of separation. For the more general case it was also shown that the angle subtended by the shock and flow direction immediately ahead of the shock, λ_1, determines the streamline deflection.

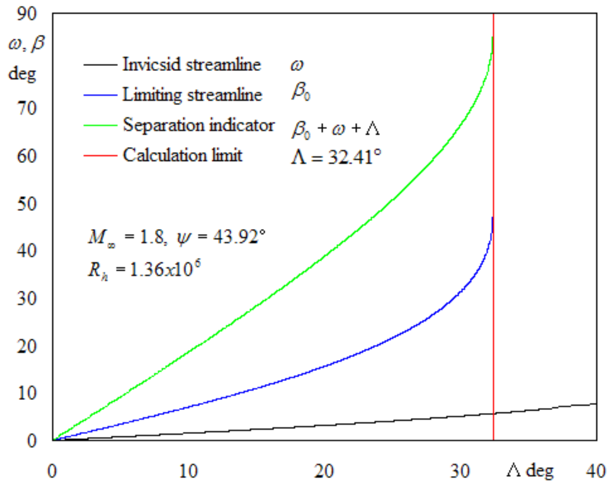


Figure 6. Jump in streamline angles at shock

While boundary layer analysis suggests that the inviscid streamline deflection is the most appropriate design parameter, it will often be more convenient to think in terms of the shock sweep as the design variable. With reference to sketch 1 the shock sweep in the plane of the plate and relative to the oncoming flow is simply defined as,

$$\Lambda = \pi/2 - \lambda_1$$

When couched in terms of shock sweep, the general equation for inviscid streamline deflection through the shock provides the link between these two parameters and gives,

$$\omega = \sin^{-1}[(Q_1/Q_2)\sin\Lambda] - \Lambda \qquad (8)$$

It will also be noted that the total deflection of the limiting streamline as it passes through the shock is given by the value,

$$\omega_0 = \beta_0 + \omega$$

While the limit condition for separation can be written as,

$$\pi/2 = \beta_0 + \omega + \Lambda = \omega_0 + \Lambda \qquad (9)$$

The significance of the limit is that for $\beta_0 > \pi/2 - \omega - \Lambda$ the flow at the surface immediately downstream of the shock would be moving towards the shock and consistent with reversed flow normal to the shock front.

Figure 6 shows the results of presenting the inviscid and limiting streamline deflection data together with the separation limit indicator as functions of shock sweep. From this the shock sweep at which separation is found to occur is $\Lambda = 32.41°$.

4 Conclusions

Three-dimensional boundary layer theory has been applied to the supersonic air inlet design problem studied by GARTEUR Action Group 34. The essential part of the inlet geometry is a stream surface that passes through a conical shock wave as generated by a right circular cone. Ahead of the shock the stream surface is equivalent to a flat plate aligned to uniform supersonic flow. Downstream of the shock the stream surface forms a bump and the bump cross section tends towards that of the right cone as the flow extends towards infinity. On the bump surface the flow is isentropic and the pressure rises asymptotically from the value at the shock towards the cone surface value. Downstream of the shock the boundary layer calculation indicates that radial flow divergence is sufficient to overcome the effects of adverse pressure gradient on the forward part of the bump surface. Through this effect the consideration of separation can be restricted to the area of shock boundary layer interaction.

It is shown that at the shock the boundary layer calculation is determined by the attitude of the incoming inviscid flow to the shock front. The attitude of the inviscid flow together with the velocity jump through the shock determines the deflection of the inviscid stream through the shock. The streamline deflection is a function of the distance from the bump centre line while the pressure rise remains constant, so for the shock boundary layer interaction it is convenient to examine the boundary layer interaction as a function of inviscid streamline deflection.

Ahead of the shock the boundary layer is two-dimensional, with limiting streamlines on the plate surface being parallel to the free stream. Through the shock these surface streamlines are deflected to a greater extent than the inviscid flow and rotate away from the bump centreline. Providing the inviscid streamline deflection is less than 5.66° the boundary layer calculation indicates that the flow remains attached. For values of streamline deflection greater than 5.66° the limiting streamlines rotate and approach the plane of the shock. For these large limiting streamline angles the boundary layer solution shows singular behaviour associated with three-dimensional separation.

As the inviscid streamline deflection increases beyond 5.66°, the related flow conditions downstream of the shock remain capable of supporting attached flow and so appear conducive to reattachment. For this reason the boundary layer calculation appears consistent with the occurrence of a vortex that develops along the foot of the shock and starts from the lateral position $y/h = .783$.

While the application considered by this paper involves a particular type of air inlet, the form of analysis that has been demonstrated can be used more generally for flows involving shock waves. This type of analysis is particularly advantageous when a shock wave is generated by a salient edge or discontinuity at a surface, as the shock position and shock strength then directly relate to the configuration geometry and can be accurately determined using Euler or shock wave theory. Once the shock position and strength are known the streamline deflection angle can be readily calculated along the shock front so providing an important design parameter and insight of the nature of the design problem.

References

[1] Seddon J and Goldsmith EL. *Intake aerodynamics*, pp 374-375, Collins Press, London, 1985.

[2] Bradbrook K. CFD study of bump inlets – interim report. British Aerospace MA&A report no BAe-WAE-AR-GEN-001347, issue 1, 1999.

[3] Ferri A. Phenomena for conical bodies – hodograph system, *Elements of aerodynamics of supersonic flows*, Macmillan Press, New York, 1949.

[4] Bower RE, Davies RS, Dowd RE, Brook J and Kachler. Design and development of pre-compression bump surfaces for use with supersonic inlets. Grumman Aircraft Corporation, Research reports 122 and 129, 1959.

[5] Sears WR (Editor). *General theory of high speed aerodynamics*, pp 721-727, Oxford University Press, London, 1955.

[6] Cross AGT. Separated flow calculations involving integral boundary layer methods and quasi-simultaneous coupling. *Conference on the prediction and exploitation of separated flow*, RAeS, London, 1989.

[7] Cross AGT. Turbulent boundary-layer methods for supersonic flow. *RTO-AVT/VKI special course on: fluid dynamics research on supersonic aircraft,* Brussels, 1998.