Model of the Engine Exhaust System at Transonic Flight Speeds

T.H. Moulden* and J.M. Wu†
University of Tennessee Space Institute, Tullahoma, Tenn.
and
D.J. Spring‡
U.S. Army Missile Command, Redstone Arsenal, Ala.

The paper starts with a discussion of the physical nature of the transonic strong interaction flow such as is found when an exhaust jet issues from a nozzle exit plane into a near sonic freestream. Schlieren photographs of the flow are interpreted in a new light which suggests how existing theoretical models for this class of flow could be improved. Results from calculations based upon a theoretical model are then presented. Particular emphasis is placed upon the effect of changes in Reynolds number. It is shown that flows which exhibit separation at low Reynolds number may become attached as the Reynolds number is increased in which case the drag coefficient exhibits a maximum.

Nomenclature

d, D	= body diameter
h	= step height
p	= pressure
R_e	= Reynolds number
C_D	= drag coefficient
M	= Mach number
x	= coordinate along stream direction
$N1$	= 0.80 diam ratio nozzle
$N12$	= 0.45 diam ratio nozzle

Subscripts

B	= base
c	= chamber pressure
J	= jet
p	= plateau value
s	= separation condition
∞	= freestream value

Introduction

In many situations of practical interest in aerodynamics, the freestream conditions are such that the flow over a given configuration is transonic. As examples of this, it can be noted that many aircraft cruise at a Mach number which gives mixed flow over some part of the vehicle while a missile in its initial phase of flight will accelerate through the transonic region. In both of these cases the propulsive jet can have a detrimental influence on the flow over the entire configuration. This is particularly noticeable in the missile application where the engine will be operating at high thrust coefficient and a large exhaust plume is created as a consequence. The situation becomes more delicate near the sonic state when this plume can engender a separation of the flow from the missile body ahead of the nozzle exit plane. Such a separation can greatly reduce the vehicle stability and controllability just in the post launch phase.[1]

Presented as Paper 74-1175 at the AIAA/SAE 10th Propulsion Conference, San Diego, California, October 21-23, 1974; submitted October 29, 1974; revision received February 3, 1975. This work was supported under Contract DAAH01-74-C-0183 with the U.S. Army Missile Command, Redstone Arsenal, Alabama.

Index categories: Jets, Wakes, and Viscid-Inviscid Flow Interactions; Subsonic and Transonic Flow.

*Assistant Professor of Aerospace Engineering. Member AIAA.
†Professor of Aerospace Engineering. Member AIAA.
‡Research Engineer. Member AIAA.

For aircraft applications on the other hand, the consequences of the exhaust jet interaction with the main flow will not be so spectacular but are still serious. The emphasis is now more upon the interference drag penalty introduced by the engine efflux and its influence on the flow over the nacelle. An important item in this respect is the effect of Reynolds number changes on the flow and the relationship between tunnel tests and freeflight. The present paper is addressed to a discussion of these problems.

Substance of a Flow Model

Figure 1 shows a typical schlieren photograph of a jet efflux from a nozzle configuration in a transonic main stream (in this example $M_\infty = 1.1$). The components of the flow are readily seen and may be identified as follows. a) The main flow over the body develops a boundary layer on that surface. b) This boundary layer leaves the body at some point to form a free-shear layer. c) The free-shear layer proceeds downstream and suffers a confluence with the external surface of the exhaust plume. d) Shock waves can be present at various locations in the interaction region when the external flow is transonic. The actual disposition of these shocks will depend, at a given freestream condition, upon the jet thrust coefficient. Because of the transonic nature of the flow, it is found that part of the interaction region is supersonic in character while the rest of the flow is subsonic.

Surface pressure distributions on the body ahead of the base region are shown in Fig. 2 for various jet total pressure ratios. The data for this plot are taken from Ref. 2. Of par-

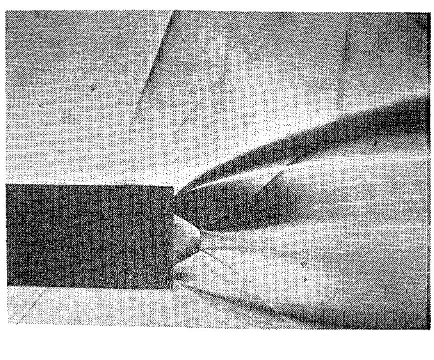

Fig. 1 Schlieren photograph of engine exhaust for high jet pressure ratio.

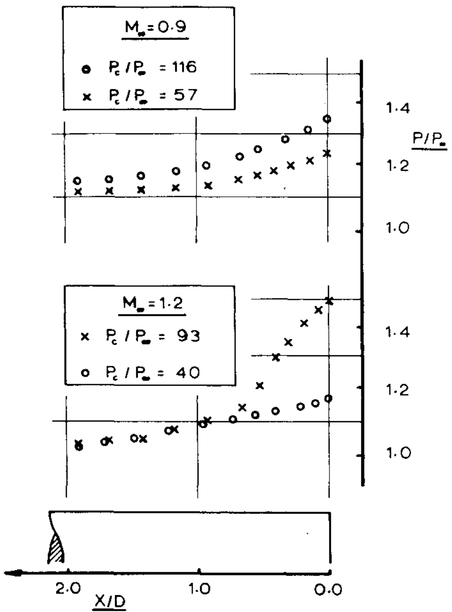

Fig. 2 **Effect of jet pressure ratio on the surface pressures ahead of the nozzle exit plane. Flow from left to right.**

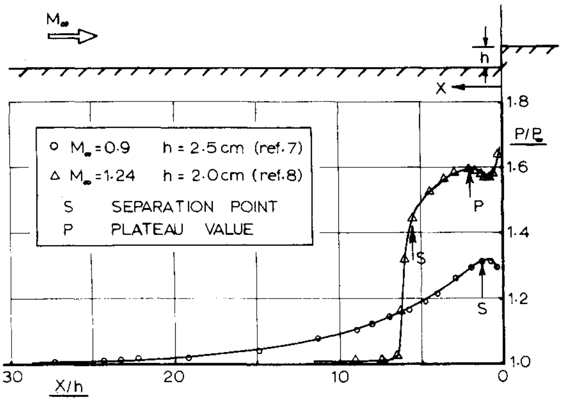

Fig. 3 **Comparison of pressure rise to separation for subsonic and supersonic approach conditions. Step height large compared to the boundary-layer thickness.**

ticular interest in this study is the spread of the upstream influence as the jet pressure ratio is increased. For missile applications, where a large jet pressure ratio is to be utilized, the extent of this upstream influence is a critical factor for the stability and controllability of the vehicle.

The difference in character of the upstream pressure rise between the subsonic and the supersonic approach velocity cases should also be noted. This point is discussed in greater detail in Ref. 3. A comparison of interest that was presented in Ref. 3 is reproduced in Fig. 3 and pertains to the flow over over a forward-facing step.

One important point must be stressed here. When the freestream Mach number is supersonic, the nature of the pressure rise on the body surface is dependent upon the magnitude of the disturbance which is creating the upstream effect. This point will again be made clear by reference to the flow over a forward-facing step. That such a discussion would be relevant to the subject in hand has been demonstrated in Ref. 1. Further evidence is put forward in Fig. 4. In this figure, the data correlation for the plateau pressure associated with the supersonic flow over forward-facing steps,[4] and its extension to transonic speeds,[3] is plotted along with the experimental data[2] for plume-induced separation. The data points are the limiting base pressure values as the jet pressure

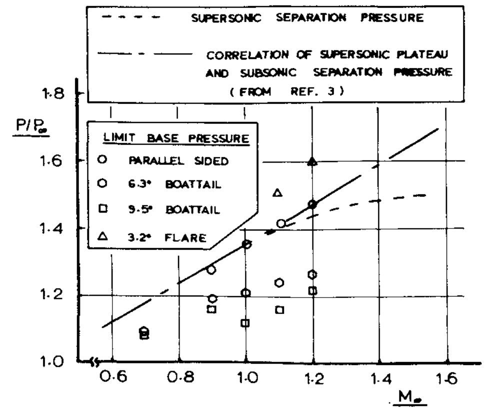

Fig. 4 **Comparison between plume-induced separation and flow over a forward-facing step.**

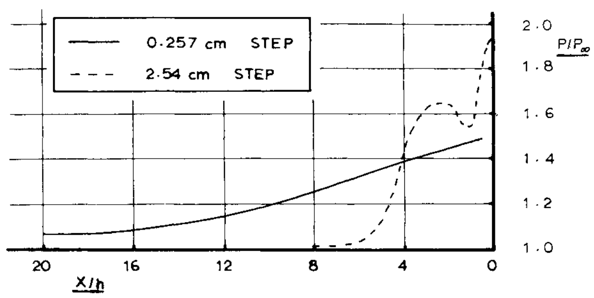

Fig. 5 **Data taken from Ref. 5 for flow over a forward-facing step.**

ratio becomes very large. Figure 7 should be referred to for the definition of this limiting base pressure. For the body with a parallel sided aft-end, it is seen that the limit pressure does in fact compare with the data correlation for the flow over forward-facing steps. For bodies with conical aft ends, however, there is a considerable divergence from this correlation as the cone angle is increased. This important three-dimensional effect will be discussed later.

Very interesting data was presented by Czarnecki and Jackson[5] concerning the influence of step size on the supersonic flow over a forward-facing step. With the approach Mach number equal to 1.61, they tested steps whose height ranged from 0.005-2.545 cm. Typical surface pressure distributions on the model ahead of the step are shown in Fig. 5. The data for the 0.237-cm step has a character much the same as the subsonic separation depicted in Fig. 3, while the 2.54-cm step shows a flow with the classical supersonic separation.

If the step produces a deflection of the external flow that is sufficiently small to be accommodated by an isentropic compression, then it is postulated that a shock wave need not occur in the pressure rise region. When this shock-free flow takes place, the classical shock-induced separation process is not evident, and the existing theories for such flows lose applicability. At the same time, classical boundary-layer theory is not able to describe a separation process under a continuous pressure rise.

Schlieren photographs are available which show the development of the flowfield as the jet pressure ratio is increased for a nozzle configuration in a body of revolution (see Fig. 6). At small jet pressure ratio (Fig. 6a), a shock wave is present in the confluence region. Then, as the jet pressure ratio is increased, this shock progresses forward and finally takes up residence as a separation shock in the classical shock-induced separation mode (Fig. 6d). For the case considered in Fig. 6, the approach Mach number is 1.1 and the body is fitted with a 9.5° boattailed aft end. In broad terms, the analogy with the step flow previously described is

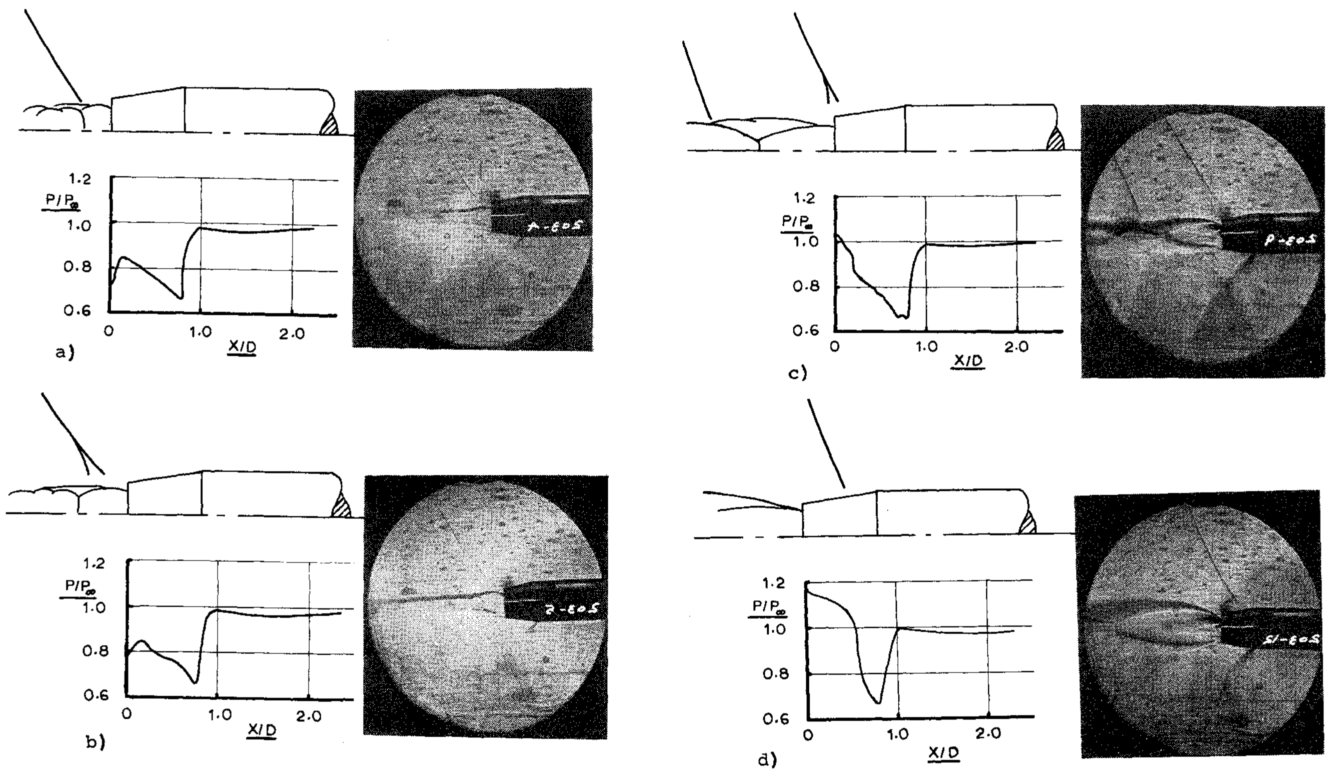

Fig. 6 Schlieren photographs and surface pressure distributions for flow over a body with a 9.5° boattail. $M_\infty = 1.1$ and various jet pressure ratios. a) $P_c P_\infty = 10.0$; b) $P_c/P_\infty = 17.9$; c) $P_c P_L = 49.9$; and d) $P_c/P_\infty = 144.0$.

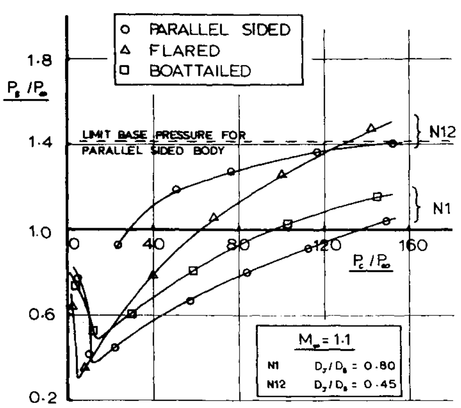

Fig. 7 Effect of aft-end conical angle on base pressure. Data from Ref. 2.

clear. At lot jet pressure ratios, the plume, being small, is analogous to the very small step flow where the separation process is enacted by an isentropic compression. Finally, at large jet pressure ratios, there exists the usual shock-induced separation.

The analogy with the step flow is only suggested as being representative of the separation process. Downstream of the separation point, the flow under discussion contains a confluence between the free-shear layer and the plume boundary. The schlieren photographs presented in Fig. 6 show the changes that take place in the flow at confluence. At low jet pressure ratio the flow is similar to a supersonic flow where confluence takes place under a wake shock. This flow progresses to a shock-free state as the jet pressure ratio is increased. This behavior is in the opposite sense to that suffered by the separation process. With the information available at the present time, there is no way of establishing whether the confluence at high jet pressure ratios takes place under subsonic or supersonic approach conditions. This fact makes the development of a theoretical model difficult.

The flow in the interaction region is further complicated by the observation (see Fig. 4) that it is quite strongly three-dimensional. That is, the limiting flow tends to a base pressure which is a function of the conical angle of the flow as well as the approach Mach number. Figure 7 plots the variation of base pressure with jet pressure ratio for three aft-end configurations and clearly shows this influence of conical angle. It is not possible to do much further analysis with this data[2] since different nozzles were employed for both the flared and boattailed configurations. It will be shown later that the nozzle geometry has a large influence on the base pressure. The whole subject of the three-dimensional nature of the flow needs further study.

Direction for Future Work

The previous section has outlined the serious difficulties that ensue when a theoretical model is put forward as a description of a separated transonic flow. In essence the problem arises because either one, or both, of the separation and reattachment regions are subsonic in nature. At this time, theories for subsonic interactive flows are not well developed. The integral approach[6] is a step towards the development of such solutions but is clearly not the final answer.

Successful theories, utilizing the classical component approach, have been developed for treating separated supersonic flows.[9] These theories have been extended to very low supersonic conditions[3,10] with some success in certain classes of problem. Further results from this theory will be given herein.

The principal difficulty that arises in the transonic region is that only part of the flow region is supersonic in character. In concept, there is no reason to suppose that component approaches would not be successful in subsonic flow, if the interaction procedure could be made to converge. However, the mechanics of selecting either subsonic or supersonic procedures would make the method of doubtful practicality in transonic flow.

It would appear that the development of numerical solutions to this type of transonic flow is the most desirable

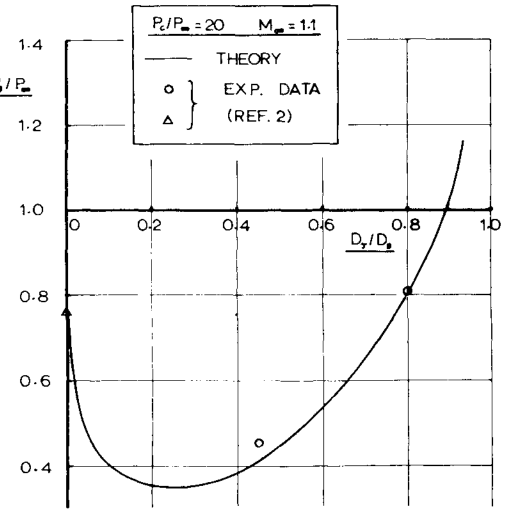

Fig. 8 Effect of nozzle diameter ratio on base pressure for a parallel-sided body.

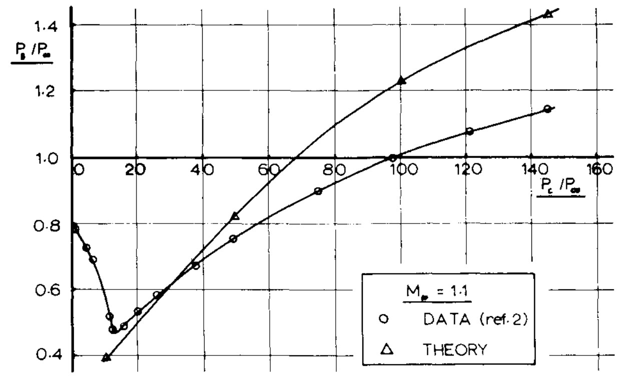

Fig. 9 Comparison between predicted base pressure and experiment. Body has a 3.2° boattail.

long term project. Sufficient experience has been gained in the past few years regarding the numerical solution for transonic potential flows that the background for viscous flow calculations is complete.

Results from the Theory

The theory employed[3,10] is based upon an interaction model that utilizes the classical component approach. It does require that the flow is supersonic throughout the interaction region. The separation process is also assumed to take place under a shock wave so that existing theories remain applicable. In view of the previous discussion, it is clear that the model under study will be restricted in its usefulness. However, it is felt that the trends predicted by the theory are valid and represent a useful study of an important practical problem.

A study has been made of the effect that the nozzle exit diameter has on the flow in the base region. A parallel-sided body was selected for this study and the ratio of the nozzle exit diameter to base diameter was varied over a wide range. Figure 8 shows the effect of this ratio on the predicted base pressure. Experimental data[2] are also included for comparison. In all cases, the nozzle exit angle was 20°. As the nozzle diameter is reduced, the base pressure tends to the value corresponding to zero mass efflux. There is, however, a minimum base pressure which occurs at some small nozzle diameter ratio. This minimum corresponds to the same ejector-type phenomenon as is observed when the pressure ratio of a given diameter nozzle is reduced. Figure 9 shows the latter effect along with the theoretical prediction for the same flow. In the case shown on Fig. 9, the body is fitted with a

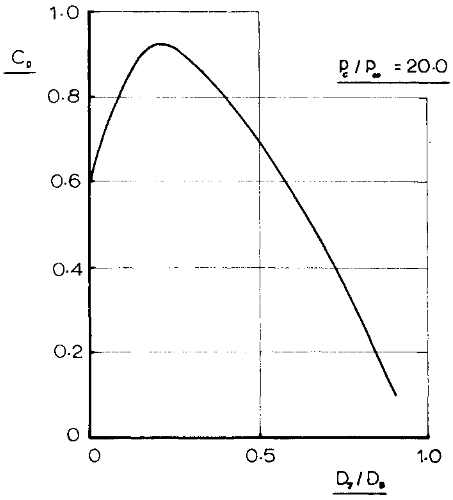

Fig. 10 Influence of nozzle diameter ratio on body drag.

3.2° boattailed aft end; again the freestream Mach number is 1.1. At the higher jet pressure ratio, the agreement between theory and experiment is not too good. This is a consequence of the observation made above concerning the three-dimensional nature of the flow when the aft end of the body is anything other than parallel sided. This effect is not included in the theory.

The changes in nozzle exit diameter cause changes in the drag coefficient which correspond to the changes in base pressure previously discussed. This aspect of the flow character is indicated in Fig. 10. Clearly, the nozzle diameter ratio must be an important parameter in any optimization study. This remark is possibly of less significance in aircraft applications where the base area tends to be small and the jet pressure ratio low.

Some remarks can be made concerning the important subject of variations in Reynolds number and the scaling from low to high Reynolds number situations. Of course, for very low Reynolds number flows, where laminar conditions predominate, vast changes can be expected as the increased Reynolds number forces the flow to be largely turbulent. In the present study, the concern is more with high Reynolds number flows and the changes that take place with further increase of Reynolds number. There is evidence[11] to indicate that under certain conditions the influence of Reynolds number can be significant. The situation is not that simple, however, since any comparison between tunnel test and free flight must contain uncertainties other than those associated with changes in Reynolds number.

The theory under discussion is able to estimate the effects of changes in Reynolds number. Certain pertinent remarks should be made before entering into a presentation of the results of such calculations. The boundary-layer theory employed in the calculation procedure incorporates a skin friction law which was obtained empirically. The data used in this skin friction correlation do not extend to extremely high Reynolds numbers. In addition to this, the conditions at the point of shock-induced separation are determined from Mager's theory[12] which does not include any influence of Reynolds number. Hence the Reynolds number only enters as a parameter in the initial boundary-layer profile at the start of the interaction. The same comment holds for the subsequent calculations through to the flow in the confluence region, and it is well known that conditions in the confluence region have a significant effect on the whole interaction.[3]

Figure 11 is a useful starting point for a discussion of the effect that Reynolds number may have on the vehicle performance. This diagram shows the variation of total drag coefficient (based upon the cross-sectional area of the body) with Reynolds number when the jet pressure ratio is held con-

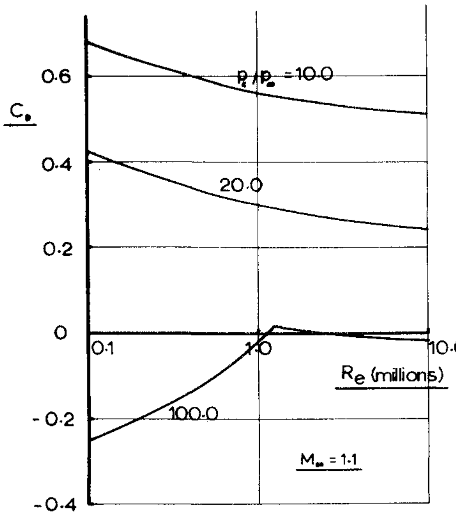

Fig. 11 Drag changes with Reynolds number for a body with a 3.2° boattail.

stant. This calculation has been performed for the same body as that referred to in Fig. 9. The freestream Mach number is 1.1.

At low jet pressure ratio, the base pressure is very low and consequently the drag is high. This can be seen from Fig. 9. Increasing the jet pressure ratio has the effect of reducing the drag at a fixed Reynolds number. For large values of jet pressure ratio, the drag is greatly reduced at Reynolds numbers below about one million because of the development of separation on the body surface. The extent of this separation is shown in Fig. 12 as a function of Reynolds number.

In the absence of separation, the drag coefficient decreases with increased Reynolds number due to the expected reduction of skin friction drag. This is shown in Fig. 13 for the case when the jet pressure ratio is ten. Also plotted in Fig. 13 are the other drag components. Because no iteration between the viscous and external flow has been attempted, the form drag is independent of Reynolds number. On the other hand, the base pressure has a small dependence upon Reynolds number due to the change in the initial velocity profile at the start of the base flow region. The major contribution to the change in total drag results from the skin friction variation with Reynolds number.

The drag decomposition becomes more complex when separation is present on the body. This is shown in Fig. 14 where the variation of the drag components with Reynolds number is plotted for the case where $p_c/p_\infty = 100$. Several points on this figure deserve comment. A separation develops on the body as the Reynolds number decreases, and this has several effects on the drag breakdown. First, there is a reduction in skin friction as the separation point moves forward on the body surface. There is also a change in the base pressure as this passes to the plateau value during the development of the separation. Once the plateau pressure is reached, further progression of the separation point with reduction of Reynolds number has no effect on the base drag. As the separation moves forward over the boattail, there is a change in the boattail form drag which reflects the increasing pressure. Once the entire boattail is influenced by the plateau pressure, this drag component will not change with further reduction of Reynolds number.

It would appear from the results of this study that the effects of changes in Reynolds number are quite orderly in cases where no flow separation is evident. In that situation, the drag of the configuration decreases with increased Reynolds number in sympathy with the reduced skin friction.

The situation is distinctly different, however, in cases where separation is indicated at low Reynolds numbers. Now the theory predicts that the drag increases with the Reynolds number. Thus the increase of form drag brought about by the reduced extent of the separated flow region is greater than the decrease in skin friction resulting from the Reynolds number change. Once the separation has completely vanished from the body, the drag decreases, as before, under the influence of the reduced skin friction. This peak in the drag curve would make the extrapolation of data impossible. Thus the data obtained at low Reynolds number with separation present lies on a curve with opposite slope to that of the unseparated high Reynolds number flow.

Throughout this study of Reynolds number effects, the transition point has been kept fixed and located at the shoulder of the forebody. While this is reasonable from the point of view of reducing the number of parameters to be varied in a given study, it is not necessarily representative of practical situations where transition is either free or arbitrarily imposed.

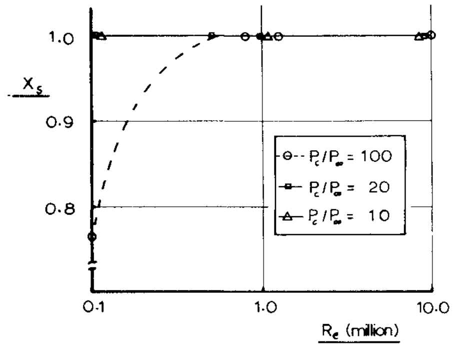

Fig. 12 Onset of separation at low Reynolds number and high jet pressure ratio.

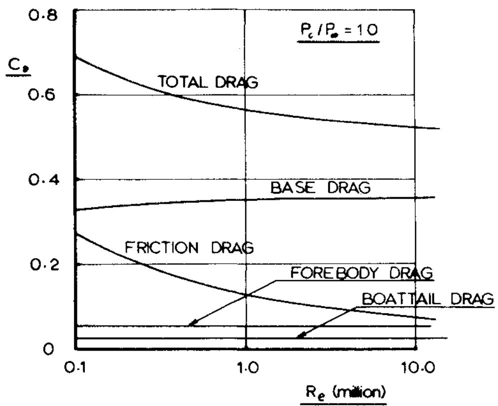

Fig. 13 Drag breakdown for the body of Fig. 11.

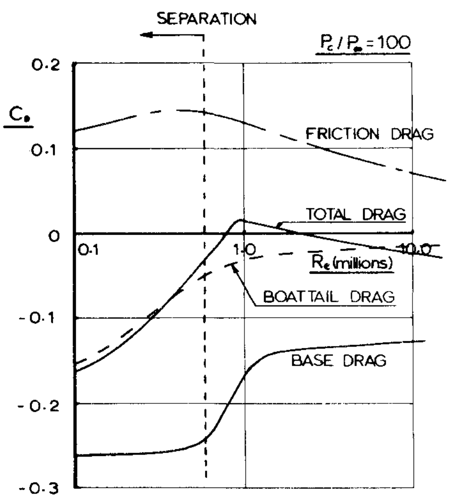

Fig. 14 Effect of Reynolds number on the drag components.

Conclusions

A study has been made of the nature of the flow in the base region of a bluff body moving at transonic velocities. The available evidence has been considered and the flow model elaborated upon. Theoretical studies have shown that the dependence of the flow on Reynolds number is quite complex when flow separation is present. In particular, the variation of drag coefficient with Reynolds number may exhibit a maximum so that extrapolation of low Reynolds number data is difficult.

References

[1]Deep, R.A., Henderson, J.H. and Brazzel, C.E., "Thrust Effects on Missile Aerodynamics," U.S. Army Missile Command RD-TR-71-9, May 1971, Redstone Arsenal, Ala.

[2]Rubin, D.V., "A Transonic Investigation of Jet Plume Effects on Base and Afterbody Pressures of Boattail and Flair Bodies of Revolution," U.S. Army Missile Command RD-TR-70-10, Oct. 1970, Redstone Arsenal, Ala.

[3]Moulden, T.H., Wu, J.M. and Spring, D.J., "On Some Problems Encountered in a Theoretical Study of the External Flow Over a Nozzle Configuration in Transonic Flight," AGARD CP 150, AGARD Specialists Meeting on Airframe/Propulsion Interference, Rome, Sept. 1974.

[4]Zukoski, E.E., "Turbulent Boundary-Layer Separation in Front of a Forward-Facing Step," *AIAA Journal,* Vol. 5, Oct. 1967, pp. 1746-1753.

[5]Czarnecki, K.R. and Jackson, M.W., "Turbulent Boundary-Layer Separation Due to a Forward-Facing Step," AIAA Paper 74-581, Palo Alto, Calif., 1974.

[6]Klineberg, J.M. and Steger, J.L., "Calculations of Separated Flows at Subsonic and Transonic Speeds," *Proceedings, Third International Conference on Numerical Methods in Fluid Dynamics,* Springer-Verlag, Berlin 1972, pp. 161-168.

[7]Wu, J.M., Moulden, T.H., Elfstrom, G.M., Reddy, K.C., Chen, C.W., Nygaard, R., Shen, L., Venghaus, H.H., and Anjaneyulu, K., "Fundamental Studies of Subsonic and Transonic Flow Separation—Summary Report," AEDC Tech. Rept.,TR-75-95 1974, Arnold Engineering Development Center, Tullahoma, Tenn.

[8]Wilson, R. and Maurer, F., "An Experimental Investigation of Turbulent Separated Boundary Layers at Low Supersonic Mach Numbers," Deutsche Forschungs and Versuchanstalt fur Luft- and Raumfarht, Munich, F.R. Germany. DRL FB 70-30, Aug. 1970.

[9]Korst, H.H., "A Theory for Base Pressure in Transonic and Supersonic Flow," *Journal of Applied Mechanics,* Vol. 23, Dec. 1956, pp. 593, 600.

[10]Wu, J.M., Moulden, T.H., and Spring, D.J., "On Some Studies of Aft-End Design and Its Influence on Plume Induced Separation in Transonic Flight," *Proceedings, 8th JANNAF Plume Technology Meeting*, Colorado Springs, Colo., July 1974.

[11]Chamberlin, R. and Blaha, B.J., "Flight and Wind Tunnel Investigation of the Effects of Reynolds Number on Installed Boattail Drag at Subsonic Speeds," AIAA Paper 73-139, Washington, D.C. 1973.

[12]Mager, A., "On the Model of the Free Shock Separated Turbulent Boundary Layer," *Journal of the Aeronautical Sciences,* Vol. 23, Feb. 1956, pp. 181, 184.