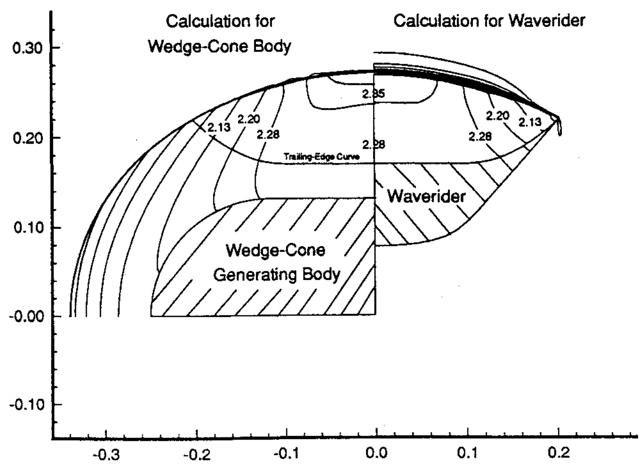


Fig. 3 Pressure contours at the exit plane.

be determined in the original generating flowfield. Despite this, excellent agreement is found between the original generating flowfield and the flowfield generated by the resulting waverider, a result that validates the design process. Also, as shown in Table 2, excellent agreement is found between the inviscid force coefficients calculated by the two methods.

Conclusions

This new class of waveriders, generated using a general nonaxisymmetric three-dimensional flowfield, shows great promise for engine-airframe integrated vehicles. This wedge-cone-derived body was found to better satisfy the requirements for forebody design than the more traditional conical waveriders. The design process was validated by calculating the flowfield about the waverider using a three-dimensional Euler solver. A high L/D value was obtained for this non-optimized waverider, which suggests that waveriders might be optimized to produce even greater aerodynamic performance, while at the same time retaining the desirable inlet flow properties.

Because it requires a three-dimensional Euler solution, the current design process is time-consuming. Future work will focus on improving the method so that the shock location can be determined more precisely. The usage of a shock-fitting method is being considered. A reduction in generating time might be found by using an osculating-cone method, however, this current process has the advantage of providing all desired flowfield information of the generating flowfield at the beginning of the design.

References

[1]O'Neill, M. K., and Lewis, M. J., "Optimized Scramjet Integration on a Waverider," *Journal of Aircraft*, Vol. 29, No. 6, 1992, pp. 1114–1121.

[2]Sobieczky, H., Dougherty, F. C., and Jones, K. D., "Hypersonic Waverider Design from Given Shock Waves," *Proceedings of the First International Hypersonic Waverider Symposium*, Univ. of Maryland, College Park, MD, 1990.

[3]Rasmussen, M. L., "Waverider Configurations Derived from Inclined Circular and Elliptic Cones," *Journal of Spacecraft and Rockets*, Vol. 17, No. 6, 1980, pp. 537–545.

[4]Corda, S., and Anderson, J. D., Jr., "Viscous Optimized Hypersonic Waveriders Designed from Axisymmetric Flow Fields," AIAA Paper 88-0369, Jan. 1988.

[5]Vasta, V. N., Thomas, J. L., and Wedan, B. W., "Navier-Stokes Computations of Prolate Spheroids at Angle of Attack," AIAA Paper 87-2627, Aug. 1987.

[6]Hsu, A., and Lytle, J., "A Simple Algebraic Grid Adaptation Scheme with Applications to Two- and Three-Dimensional Problems," AIAA Paper 89-1984, April 1989.

Visualization of Choked Supersonic Flow-Through Engine Nacelles

Eddie Irani*
Wichita State University, Wichita, Kansas 67260-0044

Introduction

SUPERSONIC wind-tunnel models of complete aircraft are usually of small scale, due to the small size of most supersonic wind tunnels. The size of engine nacelles on wind-tunnel models of supersonic transport aircraft can be quite small, these nacelles thus operate at low Reynolds number. Often such nacelles are of the flow-through type and the low Reynolds number internal flow through these nacelles may cause substantial boundary-layer buildup, resulting in a choked flow condition. The choked internal flow will cause a detached bow shock on the nacelle that could adversely affect the flow on the model, leading to inaccurate flow structure predictions.[1]

The present tests were carried out to identify the flow about a particular truncated cone-type nacelle with a smooth, constant cross section, flow-through channel, and a sharp leading edge. The effects of choked flow through the nacelle on the surrounding flow were desired. In addition, it was hoped that any difference in exit flow structure between the choked and unchoked internal flow cases could be identified. Four configurations were considered, an unchoked internal flow, a frictionally choked internal flow, and two cases of physically restricting the flow through the nacelle. The nacelle was tested at Mach 1.94 and a model Reynolds number of 2.5×10^6. The nacelle was 63.5 mm long and had a length-to-i.d. ratio of 10.

Model Description

An axisymmetric supersonic engine nacelle-pylon model with a smooth, circular flow-through channel was designed and fabricated (Fig. 1a). The model had a constant diameter hole along the flow axis to simulate the flow channel. The nacelle was shaped as a truncated cone. It was 63.5 mm in length and had a flow-through channel diameter of 6.35 mm, giving a length-to-i.d. ratio of 10. Leading edges on the model were 0.025 mm thick. The pylon was extended and flared into a standard sting that attached to the tunnel C-mount. The axis of rotation for angle-of-attack changes was fixed relative to the tunnel and located at the leading edge of the model.

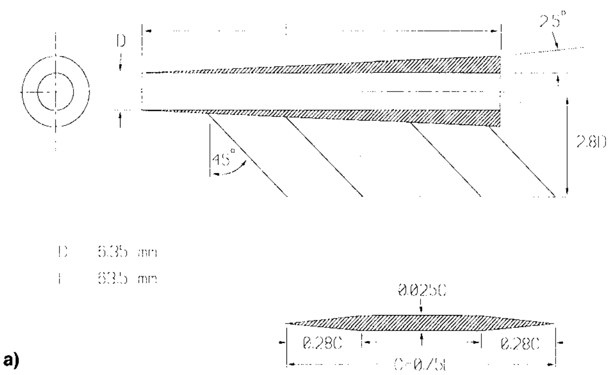
Four model configurations were tested. The first was the original unmodified model as described previously. The second was the original model coated on the inside wall with no. 80 grit to create a constant diameter channel with high friction, hence, causing friction-induced choked flow. Third, a 13-mm-long, 4.8-mm-i.d. tube was inserted in the smooth flow-through channel 6.5 mm downstream of the leading edge (Fig. 1b) to restrict the internal flow and obtain a detached bow shock. Fourth, a coarse nylon screen with 0.5-mm strands and about 50% porosity was added just upstream of the tube in the third configuration to further restrict the flow.

Facilities

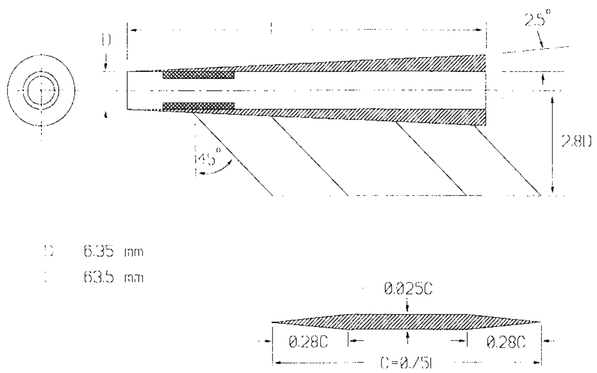
The tests were carried out at Wichita State University's 230 by 230 mm test section supersonic wind tunnel. This is a blowdown facility in which the Reynolds number may be

Received May 24, 1993; revision received Sept. 21, 1994; accepted for publication Feb. 21, 1995.

*Graduate Research Assistant, Department of Aerospace Engineering. Member AIAA.



a)



b)

Fig. 1 a) Schematic view of supersonic engine nacelle-pylon model and b) schematic drawing of model with tube restriction to induce a detached bow shock.

changed by adjusting the flow stagnation pressure. The tunnel Mach number may also be varied for Mach 2, 3, and 4 flows by changing throat blocks.

Tests

The model was tested at Mach 1.94 and a Reynolds number based on the model chord of 2.5×10^6. The shock patterns on the engine nacelle model were recorded using schlieren photographs. A standard schlieren setup was used for the tests with a vertical knife edge to show the structure of the shocks[2] (Figs. 2 and 3). The shock patterns were recorded using a 35-mm SLR camera on 3200 ASA black and white film.

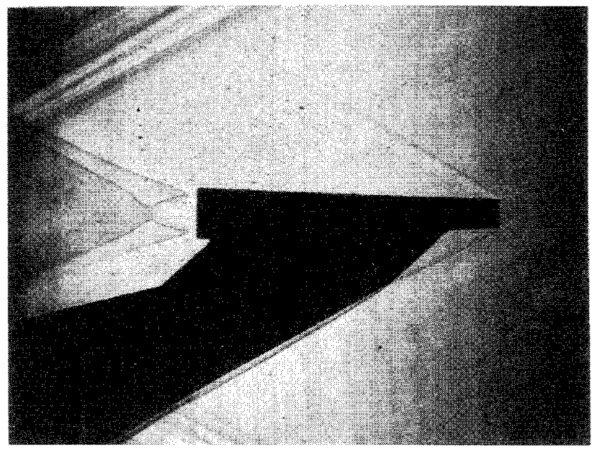
The first configuration was tested at angles of attack α of -2, 0, $+2$, and $+4$ deg. The other three configurations were only tested at an angle of attack of 0 deg.

Results

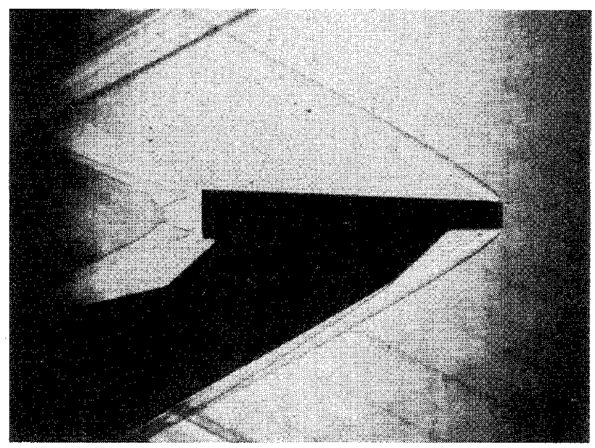
Inlet Model with Smooth Constant Area Flow-Through Channel

For configuration 1, the engine nacelle model displayed an attached shock at the leading edge and exit flow patterns characteristic of an underexpanded flow; this exit flow should be at a low supersonic Mach number, approximately Mach 1.36 (Fig. 2a). The leading edges on the model were about 0.025 mm thick and thicken behind this at a rate of 2.5 deg so that the leading-edge shock is relatively weak. It is also attached to the leading edge, indicating unchoked flow through the engine nacelle, giving a capture area ratio[3] of 1.

At the trailing edge both the flow outside the engine nacelle and the flow through the engine nacelle (internal flow) expand into the base area, the outside flow turning towards the engine centerline and the internal flow turning away from the centerline. This internal flow deflects the outside flow, creating an oblique shock. The flow expanding along the centerline speeds up and requires a normal shock on the centerline to slow down. At this point the outside flow again needs to turn back and this generates another oblique shock (Fig. 2a). This shock pattern does not repeat, however, and the flow becomes



a)



b)

Fig. 2 a) Configuration 1, smooth through-flow channel. Flow is from right to left, α = 0 deg and b) configuration 2, through-flow channel coated with no. 80 grit, α = 0 deg.

a straight horizontal jet with the mixing lines between the inside and outside flow clearly visible.

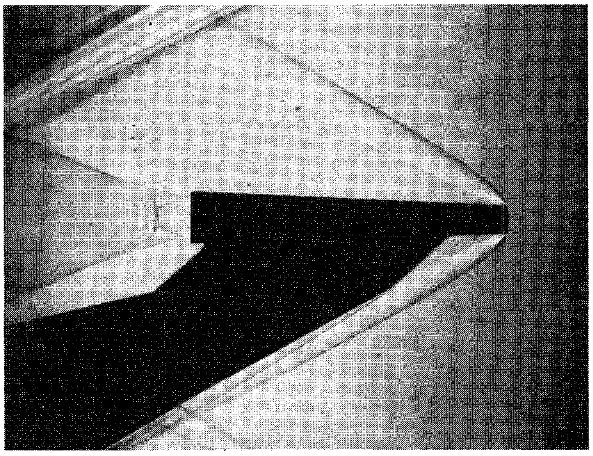
At different angles of attack, there was only one barely discernible difference: the lower part of the leading-edge shock appeared to be stronger at an angle of attack of $+4$ deg and progressively weakened with decreasing angle of attack. At the trailing edge, the flow for all angles of attack was observed to have the same character. Using a smooth channel assumption, a pressure drop of 32% and mass flow rate of 0.0144 kg/s was calculated through the nacelle.[4]

Inlet Model Modified for Choked Flow

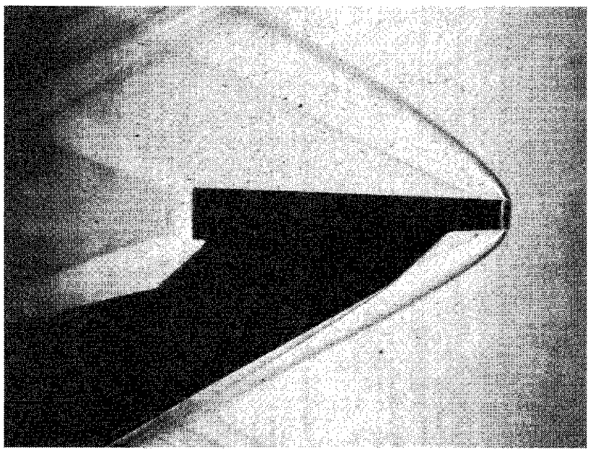
As expected, the bow shock was detached for configuration 2, indicating that the flow had been choked due to the friction caused by the no. 80 grit (Fig. 2b). The exit flow was, surprisingly, almost the same as in the case of the unchoked original model. The only indication of this choked flow was the detached bow shock and it's curved downstream path, note the separation between the lower shock and the model mount in Figs. 2a and 2b. Exit flow velocities could not be measured due to the small scale of the model and limitations of the test project. Assuming an exit flow of Mach one,[5] we can approximate a capture area ratio of 0.52, a 76% drop in pressure through the nacelle, and a mass flow rate of 0.0075 kg/s.

Inlet Model Modified for Detached Shock

To explore the effects of physically restricting the flow through the nacelle without changing inlet or exit geometry, the nacelle model was modified as indicated in the Model Description section. In both cases the leading-edge shock is detached and the exit flow is considerably different from that of the original model (Figs. 3a and 3b).



a)



b)

Fig. 3 a) Configuration 3, smooth through-flow channel with tube restriction, α = 0 deg and b) configuration 4, through-flow restricted by a tube and screen, α = 0 deg.

The exit flow in the first case, configuration 3, with only the tube restriction, is most certainly supersonic due to the converging–diverging nozzle effect of the restriction (Fig. 1b), about Mach 1.8. The exit flow in Fig. 3a shows the multiple shocks required to slow the internal flow down to freestream conditions. Assuming supersonic internal flow, a capture area ratio of 0.73 and mass flow rate of 0.0099 kg/s can be calculated.

Figure 3b shows the second case, configuration 4, with a screen added upstream of the tube. Here, evidently the flow lost enough stagnation pressure through the screen so that the flow within the nacelle and at the exit stays subsonic.

Conclusions

A supersonic engine nacelle with a flow-through channel was tested at a Mach number of 1.94 at a model length Reynolds number of 2.5×10^6: with a smooth constant diameter flow-through channel, with a rough constant diameter flow-through channel, and two cases where the channel was restricted. The test conclusions are as follows:

1) In the smooth constant diameter channel case the flow was unchoked and the bow shock was attached.

2) With a rough constant diameter channel, the flow was choked, resulting in a detached bow shock, but an exit flow structure similar to the unchoked case.

3) The two cases where the flow channel was restricted, first with a tube and second with the tube and a screen, the bow shock was detached and the exit flow was unlike case 1 or 2.

4) It can be concluded that for a particular model geometry, freestream Mach number, and pressure the exit flow structure of an adiabatic flow-through channel is only dependent on the exit flow Mach number.

References

[1]Irani, E., and Ellis, D. R., "Supersonic Wind-Tunnel Visualization of Shock Structures on Flow Through Engine Nacelles," National Institute for Aviation Research, Rept. 92-16, Wichita, KS, Sept. 1992.

[2]Goldstein, R. J., "Optical Systems for Flow Measurement: Shadowgraph, Schlieren, and Interferometric Techniques," *Fluid Mechanics Measurements*, Hemisphere, Washington, DC, 1983, pp. 377–422.

[3]Pope, A., and Goin, K. L., "Calibration and Use of Supersonic Tunnels," *High Speed Wind Tunnel Testing*, Wiley, New York, 1965, pp. 383, 384.

[4]Shapiro, A. H., "Flow in Constant-Area Ducts with Friction," *The Dynamics and Thermodynamics of Compressible Fluid Flow*, Wiley, New York, 1953, pp. 162–173.

[5]Anderson, J. D., Jr., *Modern Compressible Flow with Historical Perspective*, McGraw–Hill, New York, 1990, pp. 85–91.

Origin of Computed Unsteadiness in the Shear Layer of Delta Wings

Miguel R. Visbal* and Raymond E. Gordnier*
U.S. Air Force Wright Laboratory,
Wright–Patterson Air Force Base, OH 45433-7913

Introduction

EXPERIMENTAL observations[1–4] of the shear layer emanating from the leading edge of delta wings have revealed the existence of both steady and unsteady vortical substructures somewhat reminiscent of the classical Kelvin–Helmholtz instability found in plane shear layers. Previous computations by the present authors[5] for a 75-deg sweep delta wing at low Reynolds number also showed the presence of unsteady, three-dimensional, vortical structures in the shear layer. Frequencies were commensurate with experiments[1,2] as well as with those obtained from an inviscid linear two-dimensional stability analysis.

Recently, it has been suggested[3,4] that the unsteady type of shear-layer instability on delta wings (with which this Note is solely concerned) is simply caused in the experiments by flow disturbances inherent to the experimental setup. Given the sensitivity of shear layers to natural disturbances, this is a reasonable explanation in the experimental situation where environmental and surface disturbances are present. However, it raises the question as to the origin of the unsteadiness observed in the computations in which no deliberate forcing of the shear layer is applied. If one assumes the spatially developing, three-dimensional shear layer above the delta wing to be convectively unstable, a continuous forcing is required for the shear-layer roll-up process to persist. The purpose of this Note is to elucidate the origin of the computed unsteadiness previously reported.[5]

Results and Discussion

In the interpretation of the computed and experimental work cited earlier, emphasis has been placed almost exclu-

Received Aug. 4, 1994; revision received Feb. 14, 1995; accepted for publication Feb. 24, 1995. This paper is declared a work of the U.S. Government and is not subject to copyright protection in the United States.

*Aerospace Engineer, CFD Research Branch, Aeromechanics Division. Senior Member AIAA.