Journal of Sound and Vibration (1997) **203**(1), 75–85

THE DEVELOPMENT OF FLOW DISTORTIONS AT THE FAN FACE FOR AN AXISYMMETRIC SUPERSONIC INLET

K. Yamamoto* and W. F. Ng

Department of Mechanical Engineering, Virginia Polytechnic Istitute and State University, Blacksburg, VA, 24061-0238, *U.S.A.*

(*Received* 11 *September* 1995, *and in final form* 30 *August* 1996)

As a parallel effort to complement the experimental study on the aeroacoustics of an axisymmetric supersonic inlet, a computational study was conducted to provide a better understanding of the mechanism that generates the circumferential flow distortion at the fan face. A three-dimensional, compressible Navier–Stokes code using a finite volume method was used to calculate the flow field of the inlet with the struts located near the fan face. At 60% design fan speed, the computational results revealed a large region of three-dimensional separation on the centerbody downstream of the inlet throat. The influence of the struts on this separated flow resulted in a strong secondary flow pattern, creating a region of large circumferential distortion at the fan face near the centerbody. While the distortion caused by the strut wake shows significant reduction in a distance of half a strut chord downstream from the trailing edge of the strut, the distortion created by the secondary flow is much more persistent, and shows little sign of decay as it is converted downstream.

1. INTRODUCTION

As one in a series of studies [1, 2] on the fan noise reduction for a new generation of supersonic commercial transport, an experimental study is presented in Miller and Ng [3] to investigate the effect of choking on the aeroacoustics of an axisymmetric mixed compression supersonic inlet. It was found that in the configuration where the centerbody of the supersonic inlet was fully retracted, a large circumferential distortion existed near the centerbody between the struts. Such circumferential distortion can reduce the stall margin of the fan. In addition, as discussed in Miller and Ng [3], the unsteady blade loading due to a fan blade rotating through a circumferential distortion will result in an increase in noise generation. This companion paper is an attempt to explain the development of flow distortions at the fan face through the use of Computational Fluid Dynamics (CFD) simulation.

Most of the previous research work in the numerical simulations of supersonic inlets has been performed with the inlets under supersonic cruise speed [4–6]. Very few numerical studies were conducted to investigate the aerodynamics of supersonic inlets at takeoff and landing conditions. The work by Detwiler *et al*. [2] used a three-dimensional, viscous CFD code to investigate the development of the flow in the present inlet model due to the opening of the auxiliary inlet doors. However, for simplicity, the struts were not modelled

† Present address: Aeroengihe Division, National Aerospace Laboratory, Tokyo, Japan.

0022–460X/97/210075 + 11 $25.00/0/sv960763

in the calculation. As will be shown later in this paper, the struts play an important role in the generation of secondary flow and distortions at the fan face. Thus, in the present work, struts are included in the modelling. The focus of the research is to use numerical simulation as a tool to explore the details of the flow field. The only work in the literature that pertains to the present research is the experimental work by Lakshminarayana [7], in which a simple, axisymmetric, subsonic inlet was studied to investigate the effect of secondary flow around the struts and its influence on the inlet distortion and noise generation. The study showed that the secondary flow, which was generated by the imbalance between the pressure gradient and the centrifugal force in the boundary layer around the strut, had an appreciable effect on the noise generation. Similar secondary flow is expected to appear in the present inlet model, and the results from Miller and Ng [3] clearly indicate the existence of a large distortion near the centerbody at the fan face.

The objective of this numerical study was to provide a better physical understanding of the development of flow distortion in the inlet due to the presence of the struts through the use of CFD techniques. It should be emphasized that the purpose of the calculation was to provide a qualitative analysis of the flow field and to supplement flow data that would otherwise be very difficult to obtain in the experiment.

A brief description is given on the inlet model and the numerical method. Computational results are then used to describe the flow features and the development of flow distortions. The results reveal that the flow was distorted not only by the strut wake but also by a strong secondary flow around the strut, caused by a boundary layer separation on the centerbody. It is also shown that the strut wake decayed rapidly in a distance of half a strut chord downstream of the strut, but the distortion due to secondary flow persisted much further downstream.

2. INLET MODEL

The inlet used was a model of the axisymmetric mixed compression type inlet [8, 9], commonly referrred to as the "P-inlet". It was a representative inlet design for supersonic civil transports. The inlet model had four struts for the centerbody support. The maximum thickness to chord ratio for the strut is 17%. The detail of the experimental models are shown in reference [3]. Only the configuration with the centerbody fully retracted was considered here.

The configuration of the inlet model used in this numerical study is shown in Figure 1. The experimental model incorporated a fan simulator downstream of the struts to provide a characteristic engine noise signal. The fan face was located 5% strut chord downstream of the trailing edge of the strut. In the numerical model, the inlet was extended by about

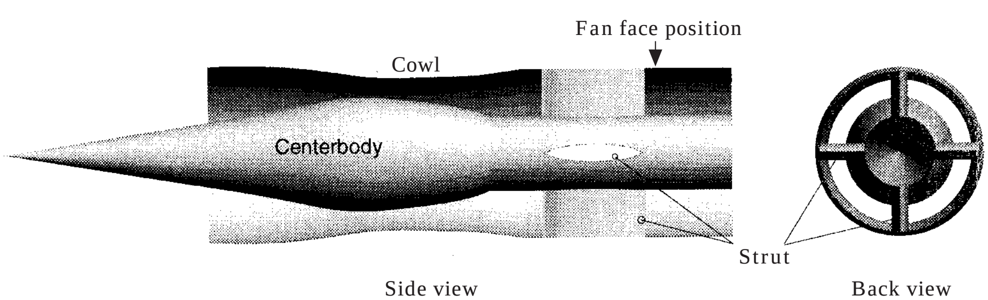


Figure 1. The inlet model.

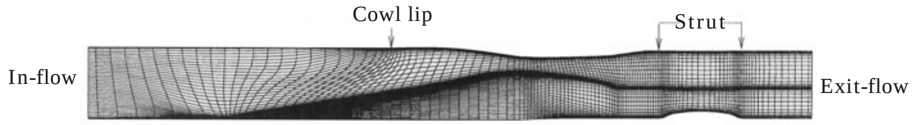


Figure 2. The computational grid.

80% chord length of the strut from the fan face as shown in Figure 1, and the back pressure was specified at this exit plane.

The calculated flow condition corresponds to 50 000 rpm (60% design fan speed), where the flow was almost choked at the throat and significant inlet distortion was observed at the fan face station. The use of this high throat Mach number is an attempt to reduce the fan noise radiation by the "choking effect", as reported in Miller and Ng [3].

3. COMPUTATIONAL METHOD

3.1. FINITE VOLUME FORMULATION

A cell-centered finite volume method for structured grids was used to solve the three-dimensional Reynolds-averaged Navier–Stokes equations. The basic scheme for convection terms was similar to the MUSCL type scheme proposed by Chakravarthy and Szema [10]. The viscous terms were calculated by using Gauss's divergence theorem [11]. To reduce the error caused by large grid stretching, the first and second gradients of the conservative variables in each finite volume cell were evaluated on the physical space. The accuracy of the discretization in space was third order for convection and second order for viscous terms. A differentiable flux limiter [12] was used to ensure the Total Variation Diminishing (TVD) property of the scheme.

For the time integration scheme, a planar Gauss–Seidel relaxation method [13, 14] was used to take advantage of the diagonal dominance property of the TVD scheme [11]. Steady solutions could then be obtained relatively rapidly, with large time steps (the maximum CFL number was about 17 000). The integration within the sweeping plane was calculated by a diagonal-dominant ADI method [13]. The radial direction was used for sweeping in the present calculation. A residual reduction of four orders of magnitude was obtained in 700 steps by this method.

The Baldwin–Lomax algebraic turbulence model [15] was used with a modification for corner flows [16], in which the turbulence properties y_{max}, F_{max}, etc. were calculated from the nearest wall. The wake of the strut was also calculated with this modification. The first grid points from the wall surface were located within the viscous sublayer ($y^+ < 5$). A numerical study was conducted and showed that the effect of boundary layer transition had a minimal impact on the flow features being studied. Thus, the entire boundary layer on the surface was assumed to be turbulent in the calculation.

3.2. GRID

Due to symmetry, it was only necessary to model a quarter of the inlet. A grid size of $37 \times 37 \times 131$ was used in the calculation, as shown in Figure 2. The grid was generated by an elliptic differential equation method [17]. This allows efficient grid points usage, with good orthogonality near the walls. The grid points were clustered around the flow separation region upstream of the struts. A grid study was performed and confirmed that the results were grid independent.

3.3. BOUNDARY CONDITIONS

The inflow and exit boundaries are shown in Figure 2. The exit boundary extended well beyond the current fan face station and allowed an assessment to be made on the advantage of placing the fan further downstream from the struts. (This will be discussed later, in the discussion section). The usual physical boundary conditions for internal flows were used in the present calculation: uniform stagnation pressure and temperature at the inflow boundary, uniform static pressure at the outflow boundary, and the no-slip condition at all solid walls. The static pressure at the outflow was adjusted so that the average throat Mach number from the calculation matched that of the experiment. The outer boundary upstream of the cowl lip was treated as a slip wall to separate the outer flow. The slip wall boundary condition and the in- and out-flow boundary conditions were calculated by a characteristic numerical method [18] to obtain higher physical accuracy. The wall boundary conditions and symmetric boundary conditions were treated implicitly in the iteration for faster convergence.

3.4. COWL LIP FLOW SEPARATION

In this calculation, no attempt was made to model the details of the small flow separation at the cowl lip, as observed in the experiment described in Miller and Ng [3]. Measurements had shown that for the present experiment, the cowl lip flow separation was small and its influence was confined to the cowl boundary layer [19]. To simulate flow separation at the cowl lip accurately, the outer flow has to be calculated with the internal flow simultaneously. This is reserved for a future effort.

4. RESULTS

4.1. FAN FACE MACH NUMBER DISTRIBUTIONS

To establish the credibility of the numerical simulation, the Mach number contour from the experiment was compared with the calculation at the fan face station. The results are presented below, in Figure 4, which shows that qualitatively the two are in good agreement.

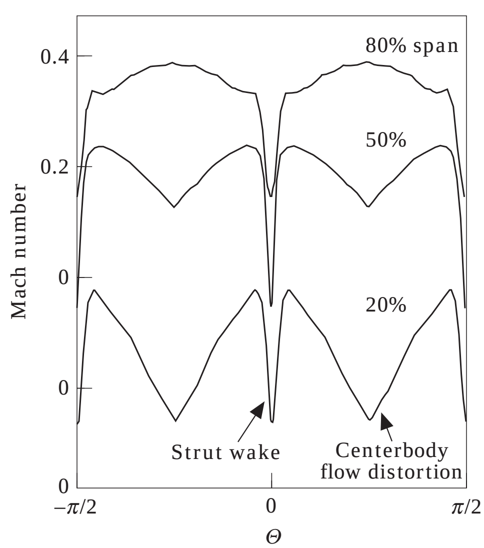


Figure 3. Circumferential Mach number distributions at three span heights at the fan face.

Three prominent flow features, as labelled in Figure 4, are identified from both the experiment and the computation: (1) the strut wake; (2) centerbody flow distortion and (3) corner flow distortion. The second flow feature will be described in the following subsection in more detail. The third flow feature is created by a boundary layer separation at the corner of the strut tip. This flow separation was caused by the adverse static pressure gradient downstream of the strut mid-chord. In Figure 4 it is shown that the strut wake in the experiment appears to be thicker compared to the computation, perhaps due to the coarseness of the measurement grid. The comparison also shows a small discrepancy in the size and shape of the cowl boundary layer. This was probably caused by the small cowl lip separation in the experiment, which was not modelled in the computation. In general, it is shown in Figure 4 that there is sufficient fidelity in the numerical simulation to justify its use as a tool to understand qualitatively the development of the flow distortion in the inlet.

In Miller and Ng [3], the importance of the circumferential flow distortion on the generation of fan noise had been discussed. All three flow features indicated in Figure 4 were sources of circumferential distortion. In Figure 3 is shown the circumferential Mach number distribution at the fan face at three spanwise locations (20%, 50% and 80% span from the centerbody). At all three locations, the wake of the strut was clearly visible as a sharp spike. The corner flow distortion appeared as a thicker wake at the 80% span location. At the 20% span location, the centerbody flow distortion showed a significant magnitude. This distortion was as deep as the strut wake but occupied a wider circumferential range. The magnitude of this distortion is such that it will have as much an effect on the noise generation as the strut wake. This centerbody flow distortion was also found in an investigation of the P-inlet [20], although the inlet model size was almost five times larger than the present model. The development of this flow distortion at the fan face will be examined in detail next. It is hoped that a better understanding of the flow physics may lead to better inlet design in the future.

4.2. DEVELOPMENT OF FLOW DISTORTION ON THE CENTERBODY

In order to identify the origin of the centerbody flow distortion at the fan face, it is necessary to examine the Mach number distributions along meridional cuts of the inlet. In Figures 5(a) and 5(b) are shown the Mach number distributions on the plane along the strut chord and on the plane at the center between two struts, respectively. It can be seen from Figure 5 that the throat of the inlet was almost choked, as it was in the experiment. The most important feature in Figures 5(a) and 5(b) appeared in the subsonic diffuser section downstream of the throat, where the flow was decelerated, resulting in an adverse pressure gradient on the boundary layers. Along a plane at the center between struts, a massive flow separation appeared in the diffuser section (Figure 5(b)), but no sign of such massive flow separation existed along a plane of the strut chord (Figure 5(a)). A very low Mach number region due to massive separation, covering about 40% of the span of the inlet is shown in Figure 5(b). It began at a short distance downstream from the throat and persisted along the remaining length of the inlet, passing the strut into the fan station and beyond.

Further to elucidate the development of the flow distortion in the subsonic diffuser section, the Mach number distributions around the struts on four cross-sectional planes are presented in Figure 6. This figure clearly shows that the centerbody flow distortion at the fan face was originated upstream of the struts. The first cross-sectional plane in Figure 6 shows the initiation of the three-dimensional flow separation on the centerbody, due to flow deceleration in the subsonic diffuser section. As discussed below, when this distortion due to the separated flow was convected downstream, it was amplifed by the secondary flow in the passage between two struts.

In Figure 7 are presented the stream of ribbons near the centerbody surface, showing the three dimensional flow structure of the flow separation and the existence of secondary flow around the struts. The recirculation in the flow separation had a twin vortex structure (stream ribbon 1 in Figure 7), which created the circumferential distortion upstream of the struts, as shown in the first cross-sectional plane in Figure 6. The low momentum fluid from the upstream boundary layer (stream ribbons 2 and 3), due to the twin vortices, deviated considerably from the main stream, moved above the twin vortices and formed the core of the distortion downstream. In Figure 7 it is shown that stream ribbon 3, which was convected from the upstream boundary layer fluid located closer to the centerbody, was pushed more toward the center between the two struts than stream ribbon 2. This flow pattern between the struts is typical of secondary flow around struts [7]. However, in the present situation, this secondary flow was amplified much more due to the presence of the flow separation just upstream of the struts. Indeed, comparing the boundary layer flow along the cowl surface with that on the centerbody (Figure 6), the absence of massive flow separation upstream of the strut leading edge for the flow along the cowl surface resulted in an exit flow with much less three-dimensionality.

Figure 8 further explains the mechanism of the twin vortices and the secondary flow development by comparing the static pressure and secondary velocity vectors on a cross-section near the leading edge of the strut. The influence on the flow due to the potential flow effect of the leading edge of the strut, coupled with the existing upstream flow separation on the centerbody, resulted in a large pressure gradient, as shown in Figure 8(a). The pressure distribution had a low pressure core located near the centerbody in between the two struts, and a high pressure region near the strut leading edge at about 70% span from the centerbody. As shown in Figure 8(b), this pressure gradient drove the low momentum fluid in and around the flow separation toward the center of the low pressure core. Low momentum fluid was then either convected upstream in the recirculation to form the twin vortices or convected downstream outside of the recirculation to form the core of the distortion.

In summary, the highly distorted flow near the centerbody at the fan face was due to the co-existence of two elements. The first was the presence of massive flow separation on the centerbody downstream from the throat. The second was the presence of the strut near this separated flow. The close proximity of the struts to this separated flow had a strong influence on the secondary flow generated in the strut passage, resulting in a highly distorted flow field at the fan face. Based on this observation, it appears that there are two possible way to minimize this distortion at the fan face. One way is to eliminate the flow separation on the centerbody downstream from the throat. If this cannot be achieved, then another way to minimize this distortion is to place the strut downstream from the re-attachment point of this flow separation. Either of these will minimize the secondary flow development and hence will reduce the circumferential flow distortion at the fan face.

5. DISCUSSION

In the previous section a detailed description is given on the formation of circumferential distortions at the fan face. With the understanding of the flow physics, a follow-up question to ask would be the following: Given an opportunity to redesign the inlet, what would be the least practical distance between the strut trailing edge and the fan face in order to avoid the large circumferential distortions? The present computational model extended almost one strut chord length downstream from the trailing edge (Figure 1), and

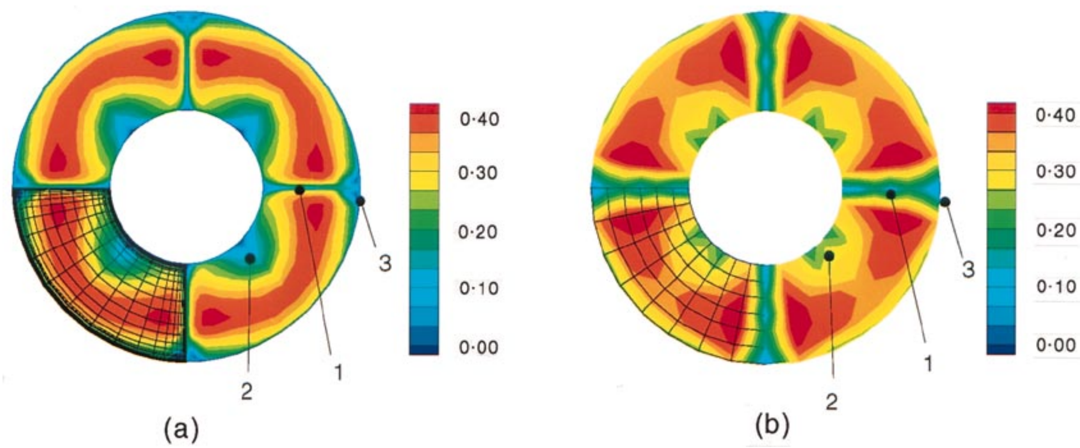


Figure 4. A comparison of Mach numbers at the fan face: the computational and measurement grids are also shown. 1, strut wake; 2, centerbody flow distortion; 3, corner flow distortion. (a) Computation; (b) experiment.

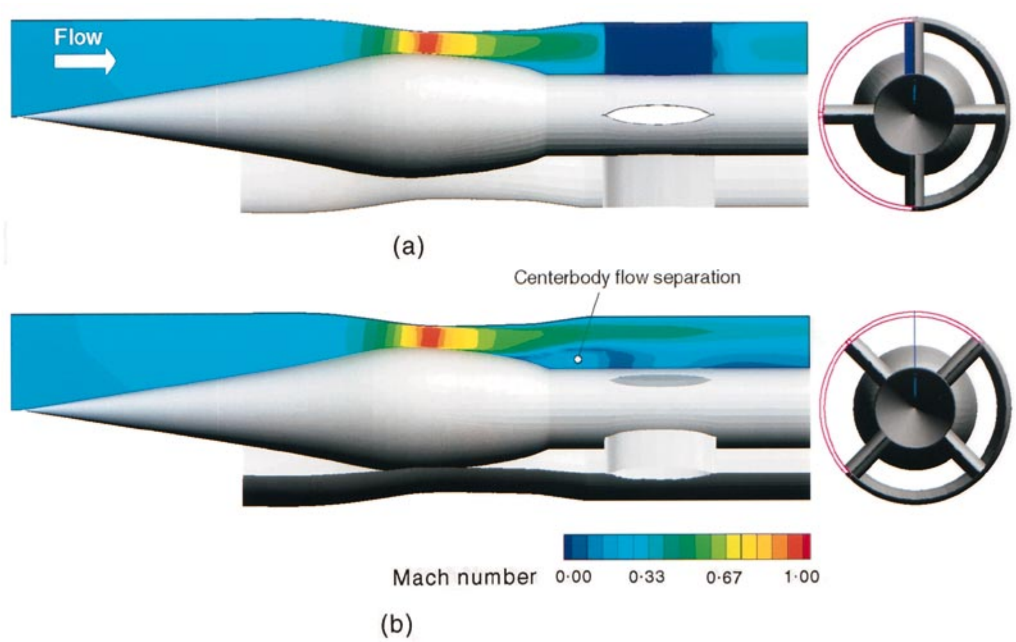


Figure 5. The Mach number distribution on meridional planes. (a) Plane along strut chord; (b) plane between struts.

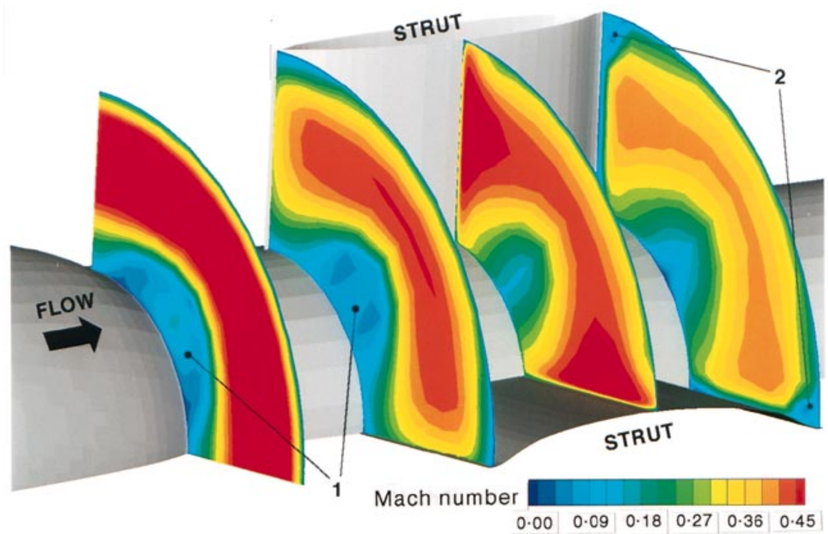


Figure 6. The Mach number distribution on cross-sections around struts. 1, Centerbody flow distortion; 2, corner flow distortion.

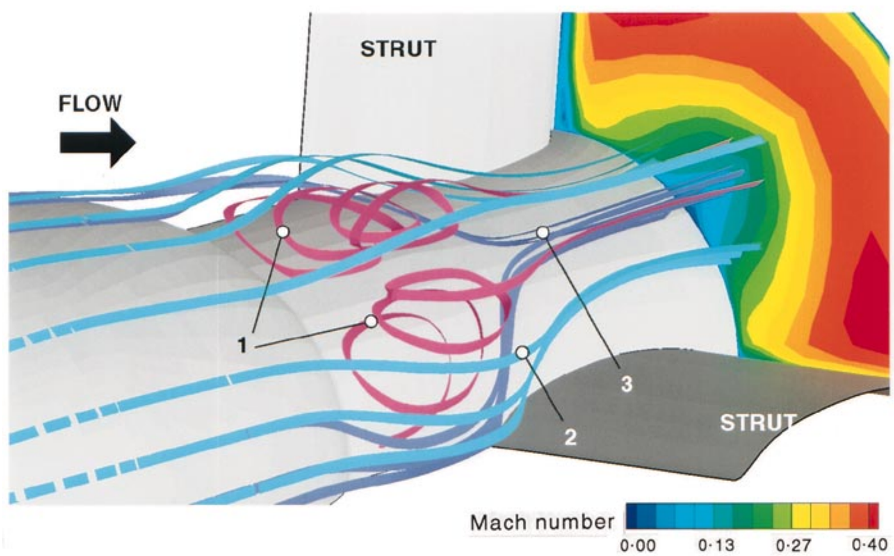
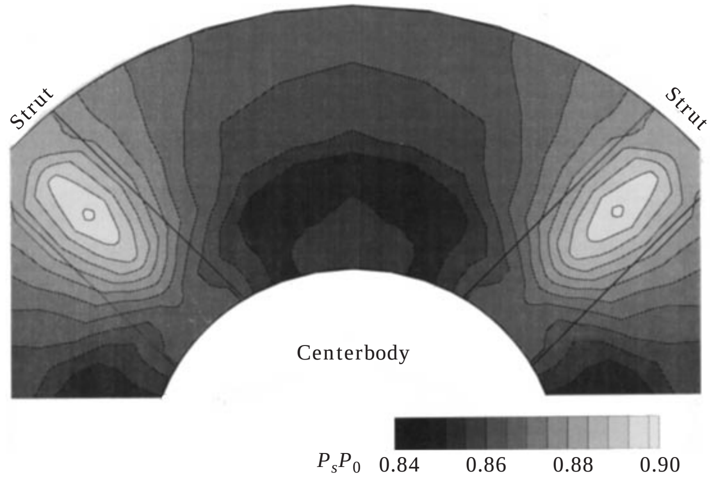
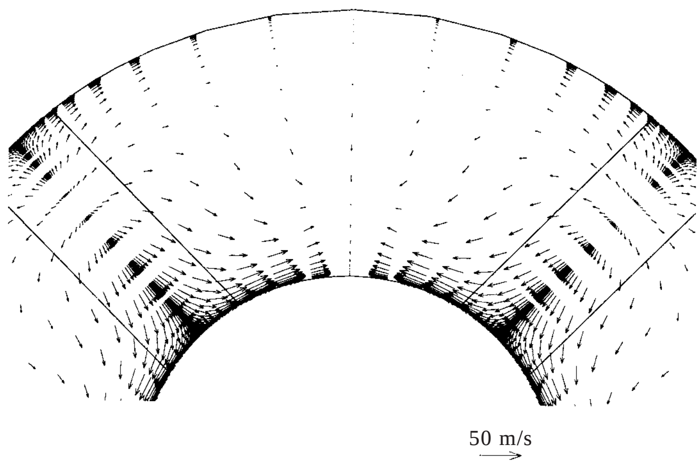


Figure 7. Stream ribbons near the centerbody and the Mach number distribution at the trailing edge. 1, Twin vortices; 2, stream from upstream, outer boundary layer; 3, stream from upstream, inner boundary layer.



(a)



(b)

Figure 8. The static pressure distribution and the secondary flow on a cross-section immediately upstream of the strut leading edge. (a) Static pressure, (b) velocity vectors.

computational results can be used to investigate the decay of the strut wake and the centerbody flow distortion as they progress downstream.

5.1. DECAY OF THE STRUT WAKE DISTORTION

In Figure 9 is shown the circumferential Mach number distribution at 20%, 50% and 80% spans of the strut, at an axial distance about 80% sturt chord length downstream from the original fan face station. The strut wake, which had a prominent effect on the distribution at the original fan face station, is still visible. However, compared to Figure 3, it had decayed rapidly in a distance of less than one chord length. At all three spanwise locations, the Mach number defects at the wake center had almost reduced by a factor of three. At the 80% span location, the distortion due to the corner flow separation was convected to both sides of the wake, as indicated in Figure 9, and also merged with the boundary layer on the cowl.

The rapid diffusion of the strut wake was not only driven by viscous forces, but also enhanced by the axial pressure gradient that existed in the wake immediately behind the

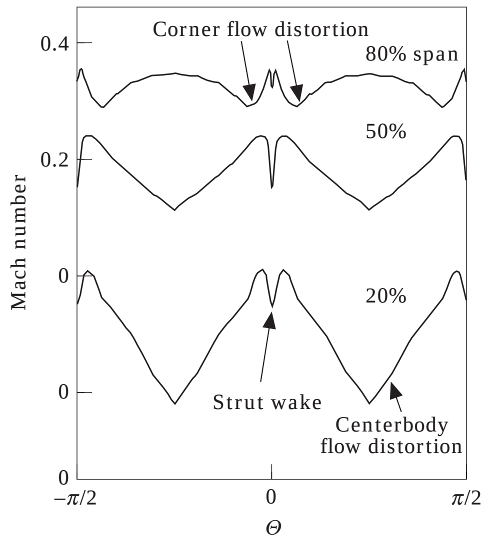


Figure 9. Circumferential Mach number distributions at three span heights at about one strut chord downstream.

strut trailing edge. In Figure 10 are shown the variations of the axial velocity in the wake center, as well as the static pressure at the same location, as a function of downstream distance (normalized by the strut chord length) from the trailing edge, taken at the mid-span location. It is shown in Figure 10 that, in the wake center, there was a sharp gradient in the static pressure in the axial direction immediately behind the trailing edge of the strut. The pressure then levelled off at about 35% strut chord length from the trailing edge. This pressure gradient in the wake center was caused by the pressure recovery at the trailing edge of the thick strut. Due to the existence of this pressure gradient, it is shown in Figure 10 that the wake center velocity was accelerated rapidly and reached an asymptotic value in less than 50% of the strut chord length downstream. It is believed that this acceleration was responsible for the rapid decay of the strut wake that appeared in Figure 9. Figure 10 also has an important implication in that much of the decay of the strut wake occurred in less than a half chord length dowstream. (This can also be given

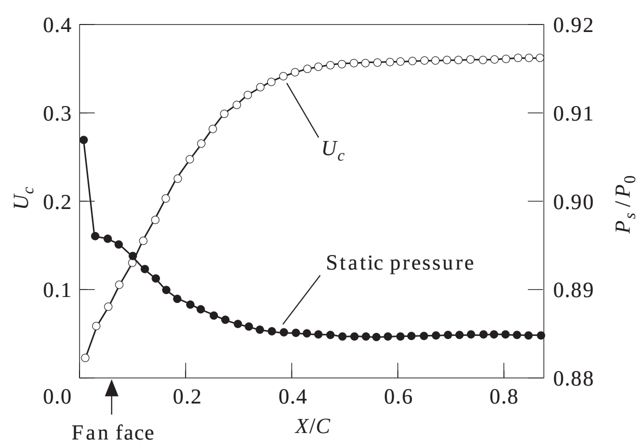


Figure 10. Variation of the wake centerline velocity and static pressure in axial direction: 50% span height.

in terms of strut thickness; half strut chord length is equivalent to three strut thicknesses in this experimental model). Thus, in order to minimize the circumferential distortion due to strut wake, the fan face should be placed at least a half chord distance away from the trailing edge of the strut.

5.2. REDUCTION OF THE CENTERBODY DISTORTION

Figure 9 also reveals that the distortion due to the centerbody flow separation is much more persistent. Comparing Figure 3 and 9, at 20% and 50% span locations, the centerbody flow distortion shows only little diffusion. Unlike the decay of the strut wake, there is no other mechanism besides turbulence mixing to diffuse the flow. At this axial location the distortion became the most significant circumferential distortion. This implies that the circumferential distortion due to the centerbody separation cannot be reduced by simply placing the fan at a position further downstream from the strut. Instead, the only way to reduce the circumferential distortion from the centerbody flow separation, as discussed in the previous section, is to attack the origin of the problem; that is, to minimize the boundary layer flow separation on the centerbody downstream of the throat, caused by the rapid expansion of the flow area. To accomplish this, the translating centerbody can be positioned such that optimal cross-sectional flow areas along the inlet can be achieved in order to employ the effect of ‘‘soft choking’’ [3], and that the flow separation on the centerbody can also be kept to a minimum. Another way to minimize the distortion due to the centerbody flow separation is to place the strut downstream from the re-attachment point of the separated flow on the centerbody. From the standpoint of the inlet pressure recovery, it seems that to minimize the centerbody boundary layer separation is a more practical approach.

It is important to point out again that in this investigation the centerbody position was chosen in order to achieve a high Mach number at the throat for choking to occur. As shown in Miller and Ng [3], choking does have an effect in attenuating forward radiated fan noise. However, the chosen configuration of the centerbody location also leads to an increase in circumferential distortion at the fan face, which could increase the noise generation and decrease the pressure recovery.

Based on this study, some general recommendation can be made regarding the aeroacoustic design of axisymmetric supersonic inlets. It is desirable to place the fan face at least about a half strut chord downstream from the strut trailing edge. This will allow the strut wake to decay and reduce its effect on noise generation considerably. While choking should be employed at the throat to attenuate noise propagation, care must be taken not to do this at the expense of generating additional circumferential distortion at the fan face, such as that caused by the effect of the strut on the centerbody separated flow. Perhaps as a compromise, ‘‘soft choking’’ (where the Mach number at the throat is high enough to making noise attenuation appreciable) can be employed without causing other undesirable effects on the inlet performance.

While it may be arguable that the accuracy of the CFD solution may be influenced by several other factors, especially by the turbulence model, it is believed that the qualitative flow features as described in this paper are much less sensitive to these factors. This paper has also demonstrated the utility of CFD in enhancing the understanding of the governing flow physics in this problem.

6. CONCLUSIONS

A numerical study was performed to investigate the development of circumferential distortion at the fan face in a supersonic inlet due to the presence of support struts. A three-dimensional Navier–Strokes code was used to provide a qualitative anaysis of the internal flow field of an axisymmetric, mixed compression, supersonic inlet, where the support struts were located immediately upstream of the fan face. This configuration represented one possible set of inlet conditions corresponding to the aircraft on approach landing, where the centerbody is fully retracted in an attempt to employ choking at the throat as a means to reduce noise radiation. Three flow features that caused circumferential distortion at the fan face were identified: (1) strut wake; (2) corner flow separation and (3) flow separation on the centerbody upstream of the strut. Of particular interest is that the distortion due to the flow separation on the centerbody was amplified by the secondary flow between the struts. The study also showed that by relocating the fan face to about half chord length downstream from the original configuration, the distortion caused by the strut wake can be reduced by a factor of almost three. However, the distortion caused by the separated flow on the centerbody is much more persistent, and showed little sign of attenuation as it is convected downstream. The understanding of the mechanism of this centerbody flow distortion suggested that it is important to re-attach the centerbody flow separation in some distance upstream of the strut, as well as to minimize its size, in order to reduce the circumferential distortion at the fan face.

ACKNOWLEDGMENTS

This research was performed at VaTech while the first author was on research leave from the National Aerospace Laboratory, Tokyo, Japan. Additional support was provided by the Department of Mechanical Engineering at Virginia Tech.

REFERENCES

1. W. E. Nuckolls and W. F. Ng 1995 *Transactions of the American Society of Mechanical Engineers*, *Journal of Engineering for Gas Turbines and Power* **117**(2), 237–244. Fan noise reduction from a supersonic inlet during simulated aircraft approach.
2. K. Detwiler, Z. Yuan and W. F. Ng 1995 *Journal of Sound and Vibration* **184**, 853–870. Experimental and numerical studies of the aeroacoustics of axisymmetric supersonic inlets.
3. K. C. Miller and W. F. Ng 1997 *Journal of Sound and Vibration* **300**, 63–73. Effects of choking on the aeroacoustics of an axisymmetric supersonic inlet.
4. T. J. Benson and B. H. Anderson 1983 *AIAA Paper* 83-0540. Validation of a three-dimensional viscous analysis of axisymmetric supersonic inlet flow fields.
5. J. Shigematsu, K. Yamamoto, K. Shiraishi and A. Tanaka 1990 *AIAA Paper* 90-2135. A numerical investigation for supersonic inlet using implicit TVD scheme.
6. B. H. Anderson and K. Kapoor 1994 *AIAA Paper* 94-2812. A study on bifurcated transitioning S-ducts for high speed inlet application.
7. B. Lakshminarayana, D. E. Thompson and R. Trunzo 1982 *AIAA Paper* 82-0125. Nature of strut of inlet guide vane secondary flows and their effect on turbomachinery noise.
8. D. B. Smeltzer and N. E. Sorensen 1972 *NASA TN D*-6971. Tests of a mixed compression axisymmetric inlet with large transonic mass flow at Mach numbers 0·6 to 2·65.
9. J. L. Koncsek and J. Syberg 1972 NASA *CR*-1977. Transonic and supersonic test of a Mach 2·65 mixed-compression axisymmetric intake.
10. S. R. Chakravarthy and K. Y. Szema 1985 *AIAA Paper* 85-1703. An Euler solver for three-dimensional supersonic flows with subsonic pockets.
11. S. R. Chakravarthy and K. Y. Szema, U. C. Goldberg, J. J. Gorski and S. Osher 1985 *AIAA Paper* 85-0165. Application of a new class of high accuracy TVD schemes to the Navier–Stokes equations.
12. W. K. Anderson, J. L. Thomas and B. van Leer 1985 *AIAA Paper* 85-0122. A comparison of finite volume flux vector splittings for the Euler equations.
13. J. Bardina and C. K. Lombard 1985 *AIAA Paper* 85-1193. Three-dimensional CSCM method for the compressible Navier–Stokes equations with application to a multi-nozzle exhaust flowfield.

14. J. R. Edwards 1995 *AIAA Paper* 95-0208. Development of an upwind relaxation multigrid method for computing three-dimensional, viscous internal flows.
15. B. S. Baldwin and H. Lomax 1978 *AIAA Paper* 78-257. Thin layer approximation and algebraic model for separated turbulent flows.
16. C. M. Hung and R. W. MacCormack 1978 *American Institute of Aeronautics and Astronautics Journal* **16**(10), 1090–1096. Numerical solution of three-dimensional shock wave and turbulent boundary layer interaction.
17. K. Yamamoto 1995 *AIAA Paper* 95-0178. A flexible elliptic grid generation method and application to turbomachinery flows.
18. H. Deconinck and R. Struys 1988 *Numerical Methods for Fluid Dynamics III*, 464–470. Consistent boundary conditions for cell centered upwind finite volume Euler solvers.
19. A. Pande 1994 *Master's Thesis*, *Virginia Polytechnic Institute and State University*, *Blacksburg*, *Virginia*. Effect of struts on the aeroacoustics of axisymmetric supersonic inlets.
20. J. F. Wasserbauer, R. W. Cubbison and C. J. Trefny 1983 *AIAA Paper* 83-1414. Low speed performance of a supersonic axisymmetric mixed compression inlet with auxiliary inlets.