PREDICTION OF COMPRESSIBLE FLOW FOR A 3D AFTERBODY USING LINEAR AND NON-LINEAR TURBULENCE MODELS

Reazul G.M. Hasan
Computational Modelling Section, Health & Safety Laboratory,
Buxton, Derbyshire SK17 9JN, U.K.

Abstract Three-dimensional RANS calculations are presented for transonic flow past a non-axisymmetric nozzle/afterbody typical of those advocated for fighter airplanes. Full details of the geometry have been modelled and the flow domain included nozzle flow and jet exhaust. Calculations were conducted for free-stream Mach numbers of 0.6 and 0.94 using two turbulence models namely, the standard k-ε model and the high-Re quadratic model of Speziale. For the lower Mach number case, the flow is well behaved and none of the models predict any kind of separation. However, for the Mach number of 0.94, the flow over the afterbody is massively separated and the Speziale model shows some improvement in the prediction of shock location and surface pressure coefficient. The effect of the turbulence models is most noticeable in normal stress anisotropy, but the validity of the predictions could not be checked due to lack of experimental data. This observation also highlights that the mean flow dynamics is not very much dominated by the normal stresses. The flow field in the jet nozzle is not influenced by turbulence models highlighting the essentially inviscid nature of the flow in this region. The paper also highlights the urgent need for more elaborate experimental data for this kind of flow.

Keywords: Transonic flow, afterbody, eddy-viscosity model.

INTRODUCTION

The numerical prediction of flow over advanced military aircraft afterbodies is a challenging task due to the complexity of the geometry, the flow regime of interest and the associated influence of the turbulence model on the shock-induced separation. Although the design of afterbody of modern highly manoeuvrable fighter aircraft is vital for drag reduction, the design methodology is still predominantly based on experimental and empirical techniques. The well-developed CFD methodologies which have been widely used in many other complex engineering appliances, remain to be fully utilized for this class of flow. One difficulty for increasing confidence in CFD predictions of these flows is the fact that very little reliable experimental data (along with well documented geometry) is available in the literature. As a result, CFD practitioners and developers find it extremely difficult to justify their numerically generated data to aircraft designers outside of the (CFD) community.

In this paper, we present calculations of the flow around a jet/afterbody geometry of length L shown in Fig. 1. The cross-sectional shape of the afterbody model is nearly rectangular with rounded edges and is typical of those advocated for advanced fighter aeroplanes. Internally, there is a converging-diverging nozzle designed to produce a jet plume at constant nozzle pressure ratio (ratio of jet total pressure, p_{tj} to free-stream static pressure p) of 4.0 and a design exit Mach number of 1.6. The experimental data (Putnam and Mercer, 1986) consisted of pressure tappings along axial lines for the inside and outside nozzle section of the afterbody for two free-stream Mach numbers of 0.6 and 0.94. For the lower Mach number case, pitot pressure measurements were also carried out in the plume. However, like most other afterbody experimental data reported in the literature, no velocity or turbulence data was available. It was found that the free-stream Mach number of 0.94 was high enough to cause shock-induced separation on the top surface. This particular afterbody model has been considered in a number of research exercises such as Compton(1990, 1996), AGARD (1995), mainly due to the availability of detailed geometrical information.

The work reported here has been carried out during a joint university-industry validation exercise called the VoTMATA (Validation of Turbulence Models for Aerospace and Turbomachinery Applications) which involved Loughborough University, UMIST, BAE Systems, DERA, ARA and Rolls Royce. A key feature of the collaboration was the use of identical numerical meshes and a collective scrutiny of differences arising from various computational approaches. This interactive approach increases the confidence in the validity of the conclusions derived from the results.

TURBULENCE MODELS

The turbulence models used in this study are the standard *k*-ε model of Launder and Spalding (1974) and the high-Re quadratic model of Speziale (1987). The first one appears to be the most widely used model among CFD practitioners. Very briefly, the linear *k*-ε model derived its name from the Boussinesq linear stress-strain relationship, whereas the other model embodies some quadratic terms that take care of the normal stress anisotropy. The differences can be more clearly observed by considering a simplified expression of the anisotropy tensor **a** up to the second order (Apsley and Leschziner, 1998):

$$\mathbf{a} = -2\, f_\mu\, C_\mu\, \mathbf{s}$$
$$+ \beta_1 (\mathbf{s}^2 - 1/3\ s_2\, \mathbf{I}) + \beta_2 (\mathbf{ws} - \mathbf{sw}) + \beta_3 (\mathbf{w}^2 - 1/3\ w_2\, \mathbf{I})$$

where,

$$\mathbf{a}=(a_{ij}),\ \mathbf{s}=(S_{ij})/\tau \text{ and } \mathbf{w}=(W_{ij})/\tau$$

Here, **s** and **w** are the mean strain and vorticity non-dimensionalised by the time scale of turbulence; for example, for the two equation *k*-ε model $\tau=k/\varepsilon$. In the above expressions, for any tensor,

$\{T\}=\text{trace}(T)$ and $\mathbf{I}=(\delta_{ij})$.

s_2 and w_2 are the dimensionless strain and vorticity invariants given by:

$s_2 = s_{ij}s_{ij}$; $w_2 = -w_{ij}w_{ij}$

The coefficient f_μ takes care of the near wall sublayer and in the current high-Re formulations it is set to be unity. For the Speziale's (1987) model the coefficients (β_1, β_2, β_3) are all empirical constants and are equal to (0.054, 0.054, 0.0). If the βs are all set to be zero, then the equation reduces to a linear relation. More details about the models can be found in the above references.

NUMERICAL ISSUES

Figure 2 shows the details of the computational grid used near the nozzle section and also shows the co-ordinate system. X is taken to be the axial distance normalised by the model length L. X=1.0 means the end plane of the afterbody. All other distances such as Y, Z, R are also normalised by L. For computational efficiency, we have assumed symmetry for both the vertical and horizontal planes and only a quarter of the domain was considered. In the present calculation, unlike other previous work (AGARD, 1995), the complete geometrical detail has been considered including the small (~1mm) lip thickness between the inner and outer nozzle cross-section. The computational domain extended both upstream and downstream of the model and the whole domain was divided into four blocks: block one ahead of the body, block two coincident with the afterbody, block three is the downstream plume section and block four is the inside nozzle. The grid topology was H-O with one-to-one connectivity between the blocks. A total of about 750,000 non-uniform grid cells have been employed for the calculations. The value of y+ for the first grid point off the wall was 40 or less for the outside surface of the afterbody and 100 or less for the internal nozzle. Introduction of the upstream block helped to avoid the uncertainty about the inlet boundary condition for the flow. The locations of the far field and downstream boundary were sufficiently far in the sense that they had no influence on the flow development near the afterbody nozzle. As shown in Fig. 2, body-fitted hexahedral grids were generated by using an in-house computer program developed for handling complex geometries. The results are believed to be grid independent and further details would appear in a future paper (Hasan et al. 2001b).

Because the free-stream is subsonic (0.6 and 0.94), Riemann invariants for a one-dimensional flow were used to calculate the primitive variables ρ, u, v, w and p at the computational domain inflow. At the outflow boundary where the flow is a mixture of the jet exhaust and the free stream, all gradients were set to zero regardless of the free stream conditions. On the far field boundary, away from the surface, the flow was considered to be tangent to the domain. At the jet inflow, experimental total pressure and total temperature were specified. Static pressure was extrapolated to the boundary from the interior of the computational domain. Finally, no-slip and adiabatic wall boundary conditions were imposed on the body surface. The turbulence intensity at the inlet, *k* was taken to be 0.01% of the free-stream mean kinetic energy and the length scale determining quantity ε was specified such that $\nu_t/\nu=0.1$. These values have little influence on the flow development and has been extensively studied for the 2D bump test case (Hasan and McGuirk, 2001a).

Calculations have been carried out using a non-orthogonal, collocated cell-centred finite-volume pressure based solution procedure employing the concept of retarded pressure (McGuirk and Page, 1990). The convection scheme was second order limited by a TVD procedure. Two turbulence models have been used. The standard linear k-ε model (Launder and Spalding, 1974) and the quadratic model of Speziale (1987). Both of these models are high Reynolds number models and hence standard wall functions have been used to bridge the gap between the wall and the first grid node.

RESULTS AND DISCUSSION

Calculations were carried out for two free-stream Mach numbers of 0.6 and 0.94. The jet total pressure ratio was about 4 for both the cases and hence the flow development inside the convergent-divergent nozzle was almost identical. However, the flow over the afterbody displayed completely different features for the two Mach numbers. For both the cases, the flow accelerates as the fluid moves towards the aft of the model. For the lower Mach number case, the flow is well behaved and there is no indication that the flow separates on any of the surfaces. On the other hand, for the 0.94 Mach number case, the flow accelerates to supersonic speed which is high enough to cause a shock and due to interaction with the slower moving fluid in the thickened boundary layer, a separation occurs at about X~0.90. This is a massive separation, most pronounced on the vertical surface and extends partly even to the side walls. Fig. 3a-b show the qualitative view of the of the overall flow for the higher Mach number case. The contours on the afterbody nozzle surface represent the magnitude of the streamwise velocity (U) next to the surface. This, together with the velocity vectors (only a few vectors are shown for clarity) on the vertical plane, show the location of the shock on the afterbody and the vertical and longitudinal extent of the separation. The exit Mach number from the convergent-divergent nozzle is about 1.6 and it interacts with the separated flow coming off the afterbody external surface resulting in a very complex shear layer in the plume section of the domain.

Fig. 4 compares the pressure variation (static divided by jet total) of the internal nozzle surface along the vertical (Y=0) and horizontal (Z=0) planes. The flow here is almost inviscid, with an extremely thin boundary layer and hence showing no sensitivity to turbulence model. As can be seen, the pressure fluctuations have been poorly captured by the current pressure-based algorithm and is likely to be resolved much better by a compressible density-based method (Hasan et al. 2001b). It is possible that indirect calculation of the pressure in a pressure-based code associated with Rhie and Chow smoothing and density blending may be responsible for this behaviour.

Detailed comparisons with the available surface pressure data along the five axial measuring lines are shown in Fig. 5a-b. For the lower Mach number case, both the models perform well and the quality of predictions is nearly similar. The differences are very small, except along line 3 which is believed to be not due to turbulence model. In fact, line 3 is along the sharp edge of the afterbody and hence represents a poor location for profile comparison. For the higher Mach number case, the use of quadratic model slightly improves the predictions. In particular the shock location and recovery are better resolved.

Extensive pitot pressure measurements were available for the jet exhaust plume for the free-stream Mach number of 0.60. Unfortunately no such measurements were made for the separated higher Mach number case. Comparisons with the experimental data are given in Fig. 6 at three streamwise locations i.e., at X=1.0, 1.08 and 1.16. The first set of profiles ϕ=(0-15) deg are near the vertical plane of symmetry, those at ϕ~25 are in a plane that cuts through the top surface of the nozzle, those at 65 deg are in a plane that intersects the nozzle at approximately the corner and those at ~90 deg are near the horizontal plane of symmetry. Overall, the turbulence models produce very little difference in the results except at further downstream stations where the quadratic model shows slightly superior shear layer resolution. Compared to the experimental data, the results are very close to experiments at X=1.0. These results are clearly superior to some of the calculations reported in AGARD (1995) due to the fact that the full geometry including the small thickness has been modelled in the current exercise. The quality of predictions become poorer at downstream locations. Two characteristics are particularly significant. First, the effect of weak shock cells inside the nozzle has not been captured at all and this is consistent with Fig. 4. The experimental pitot pressure at the centre line shows an oscillating pattern between the three axial stations, which the current pressure based algorithms failed to predict. However, the density based codes were found to capture this phenomenon (Hasan et al. 2001a). The other characteristic is the poor agreement at ~65 deg location. The spreading rate is found to be severely underpredicted. One reason for such poor prediction could be due to grid topology, which is O-type in the cross flow plane and may not allow accurate resolution of the viscous terms in the corners. Turbulence model may also be responsible for this discrepancy. However, as will be shown in the future paper, even the advanced eddy-viscosity models including a cubic model with strain dependent C_μ have failed to resolve the shear layer in this region. Hence the results do highlight the importance of investigating the flow with Differential Stress Models.

Finally, some details of the flow development are given in Fig. 7. Although the mean velocity and shear stresses are very close to each other for the two models, the magnitude of the normal stresses are resolved quite differently. However, the accuracy of the results could not be judged due to lack of experimental data. These however, highlight the fact that dynamics of the flow is not dominated by stress anisotropy.

CONCLUSIONS

One of the gross but important findings from the work is that as long as the flow remains attached, the performance of the eddy-viscosity models are of reasonable quality at least for the flow variables considered here. Hence such models may be used for design purposes with fairly good confidence level. For the higher Mach number case, both of the models display delayed shock location and poor recovery, with the Speziale’s model performing only slightly better. The Speziale’s model shows some difference in resolving the normal stress anisotropy but the effect of this on other flow parameters such as pressure does not seem to be very influential.

The comparisons with the plume pressure data for the unseparated lower Mach number case are in general good, but show big discrepancy in the highly sheared region. This is probably due to grid topology or a limitation of the eddy- viscosity models. The current pressure based algorithm seems to smear out the weak shock cells inside and outside of the convergent-divergent nozzle.

Finally, the study also highlighted the urgent need to carry out more experiments to obtain reliable velocity and turbulence data. It is also important to obtain more pressure data to confirm the existence of pressure fluctuations inside the convergent-divergent nozzle. In fact, the comments about shock cells are rather speculative given the very sparse location of the pressure tappings. The merit or limitation of a turbulence model can only be judged by analysing the whole flow field including pressure, velocity and stress. Skin friction data are also necessary to justify the use of more expensive low-Re models for this kind of flow.

ACKNOWLEDGEMENT

The work was carried out in the Aeronautical and Automotive engineering department of Loughborough University and the author gratefully acknowledges the supoort provided by the department, in partcular by Prof. J.J.McGuirk. The financial support provided by the UK Engineering and Physical Sciences Research Council, BAE Systems, Rolls- Royce and the UK Defence Evaluation and Research Agency are also gratefully acknowledged. Thanks are also due to a number of people in the VoTMATA project from whom the author was immensely benefited.

REFERENCES

AGARD AR-318, "Aerodynamics of 3D Aircraft Afterbodies", (1995).

Apsley, D. D. and Leschziner, M. A., "A New Low-Re Non-linear Two-equation Turbulence Model for Complex Flows", *Int. J. Heat Fluid Flow*, **19**, 209-222 (1998).

Compton III, W. B., "Comparison of Turbulence Models for Nozzle-afterbody Flows with Propulsive Jets", NASA TP 3592 (1996).

Compton III W.B. and Abdol-Hamid K.S., "Navier-Stokes Simulations of Transonic Afterbody Flows with Jet Exhaust", AIAA paper 90-3057 (1990).

Hasan, R.G.M. and McGuirk, J.J., "Assessment of Turbulence Model Performance for Transonic Flow Over an Axisymmetric Bump", *Aero. J.*, **105**, n. 1043, 17-31 (2001a).

Hasan, R.G.M., Apsley, D.D., McGuirk, J.J. and Leschziner, M.A., "A Turbulence Model Study of Separated 3D Jet/ afterbody Flow", (in review) (2001b).

McGuirk J.J. and Page G.J., "Shock Capturing Using a Pressure-correction Method", *AIAA J.*, **28**, n. 10, 1751-1757 (1990).

Launder B.E. and Spalding, D.B., "The Numerical Computation of Turbulent Flows", *Comp. Meths. Appl. Mech. Eng.*, **3**, 269-289 (1974).

Putnam, L.E. and Mercer C.E., "Pitot-pressure Measurements in Flow Fields Behind a Rectangular Nozzle with Exhaust Jet for Free-stream Mach Numbers of 0.0, 0.60 and 1.20", NASA TM-88990 (1986).

Speziale C.G., "On Non-linear K-l and K-ε Models of Turbulence", *J. Fluid Mech*., **178**, 459-475 (1987).