COMPUTATION OF FLOW THROUGH AN S-BEND OF CIRCULAR CROSS-SECTION

Researcher: N.I. Abou-Haidar
Supervisors: H. Iacovides and B.E. Launder
Sponsor: British Aerospace

1. OBJECTIVES

This study is also part of our overall investigation of turbulence modelling aspects of flows through S-shaped intake ducts of aircraft engines. During the course of these investigations flows through ducts of both circular and square cross-section have been examined. Different cross-sectional geometries can lead to the development of different flow features. It is therefore important to investigate the effectiveness of turbulence models in both situations. Furthermore, the computation of circular bends is numerically easier to compute because of the need to resolve only *one* near-wall region. In the computation of circular S-bend flows it is thus possible, within the resources currently available, to implement near-wall turbulence models which require high near-wall grid densities.

2. APPROACH

Computations have been performed using both a 3-D semi-elliptic and 3-D fully elliptic code. In both cases the flow equations are solved in a toroidal coordinate system. Within the fully turbulent region of the pipe, which covers around 97% of the pipe radius, an ASM closure was used to model the effects of turbulence. Within the viscosity dominated near-wall sublayer three low-Re turbulence models were tested: an effective viscosity one-equation model, an effective viscosity two-equation model and a low-Re ASM model. The resulting computations reveal the strong influence exerted by the near-wall regions in such flows and the level of near-wall modelling required for the reliable prediction of S-bend flows.

3. RESULTS

The test case considered concerns the flow through the S-bend measured by Taylor et al [1]. The S-bend consists of two identical curved sections of 22.5° of arc and curvature ratio (Rc/d) = 7. The initial ASM/1-eqn computations reproduced the measured velocity and pressure distribution within the first bend but failed to return the correct flow behaviour through the second bend. As shown in Figure 1, the successive refinements in the near-wall model eventually lead to a realistic representation of the velocity field at the S-bend exit plane, indicating the importance of the near-wall regions in such flows and the need to account for the effects of near-wall turbulence anisotropy. The same conclusions can also be drawn from the comparisons of the static pressure distribution through the S-bend shown in Figure 2. The effective viscosity near-wall models and, in particular, the one-equation fail to reproduce the effects of the change in the direction of curvature on near-wall turbulence. A full account of this research has been published in [2].

4. CONCLUSIONS

1. In S-bend flows the anisotropy in the near-wall turbulence exerts a significant influence on the overall flow development.

2. In order to produce acceptably accurate flow predictions, a second-moment near-wall closure is necessary.

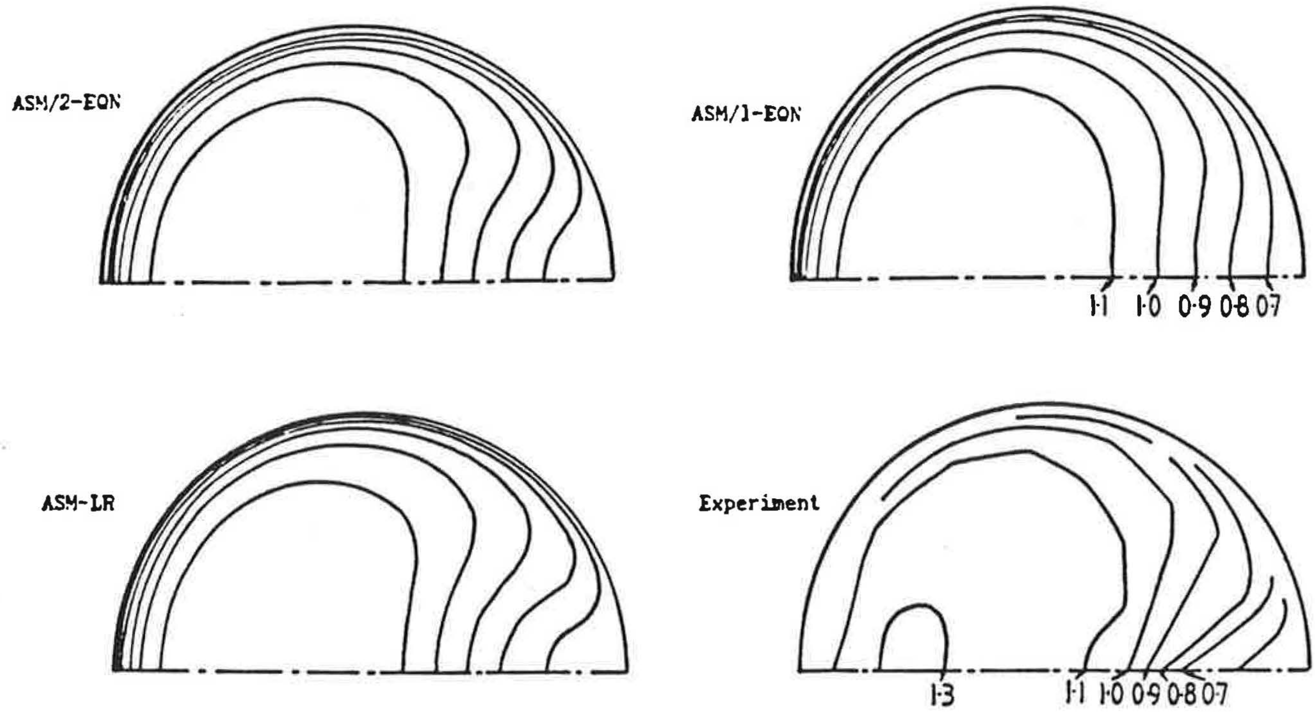

Figure 1 Distribution of the streamwise velocity at the S-bend exit plane

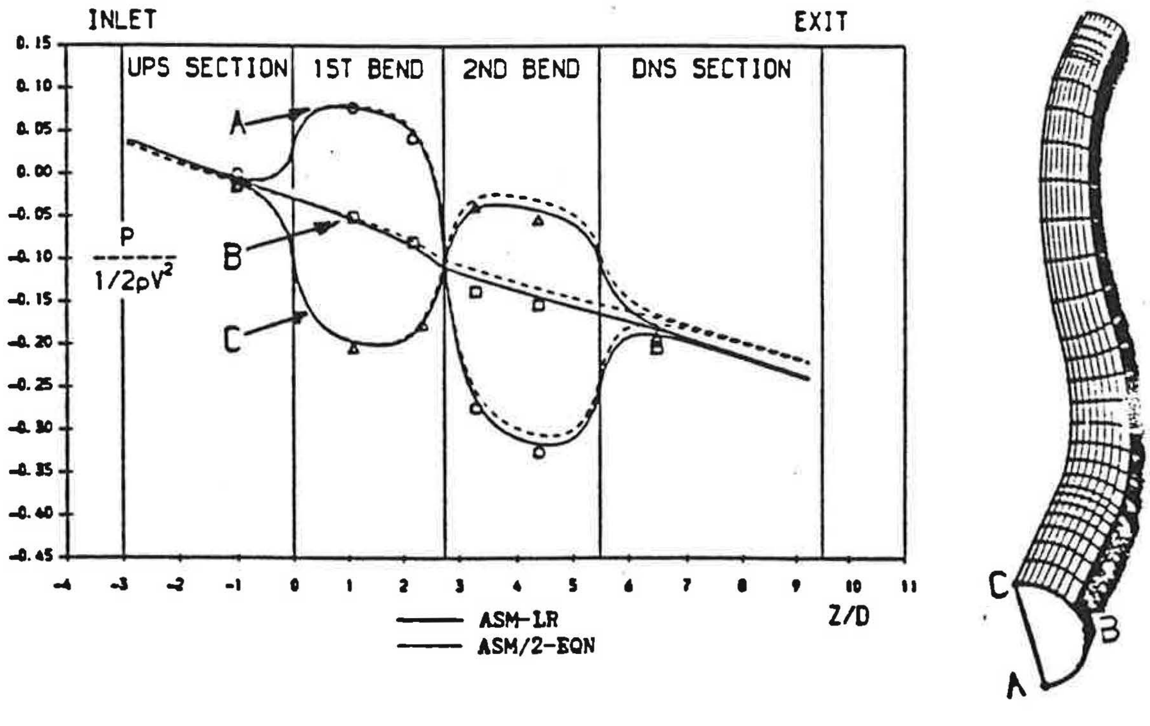

Figure 2 Streamwise distribution of the wall static pressure

REFERENCES

1. Taylor, A.M.K.P., Whitelaw, J.H. and Yianneskis, M., (1982), Report FS/82/7, Imperial College, Fluids Section (NASA CR 3759).

2. Abou-Haidar, N.I., Iacovides, H. and Launder, B.E., AGARD Conf. Proc. 510, CFD Techniques for Propl. Appl., 77th Symp., San Antonio, Texas, May 1991.