#6097 6.1

COMPUTATION OF DEVELOPING FLOW THROUGH A SQUARE CROSS-SECTIONED S-DUCT

Researcher: H. Li
Supervisors: B.E. Launder and H. Iacovides

1. OBJECTIVES

Aircraft engine inlet ducts are often S-shaped, particularly in single engined military aeroplanes. In the design of such intakes it is important to be able to predict the distribution of the flow delivered to the engine face as well as the pressure losses through the duct. The flow development is affected by curvature effects acting directly on the mean flow and also indirectly by modifying the turbulent field. The group's initial computations of S-duct flows were performed with a 3-D semi-elliptic code employing an Algebraic Stress Turbulent Model over the duct core region and a low-Re ASM model within the near-wall sub-layer. The main features of the flow development were correctly reproduced by this first computation but the pressure drop through the second bend was under-predicted. This objective of the present effort has been to try to improve the quality of the initial predictions by extending the ASM model over the sub-layer regions and also by employing a fully elliptic solver.

2. APPROACH

The conversion of the fully-elliptic code to a semi-elliptic one is relatively straight forward. For reasons of computational economy the low-Re ASM model proposed by Iacovides and Launder [1] was further simplified so that the near-wall length scale was prescribed. The resulting model was first used to predict 2-D flow through curved channels (Fig. 1). These 2-D tests allowed on evaluation of the model's ability to represent the effects of streamline curvature on near-wall turbulence. The simplified low-Re ASM model was subsequently used in the computation of 3-D flow through a square S-duct consisting of two identical bends of 22½° angle and curvature ratio $R_c/D=14$ (Fig. 2). The bulk Reynolds number was 48000.

3. RESULTS

Predicted profiles of the streamwise velocities used in the 2-D curved channel are compared with the data of Ellis and Joubert [2]. Use of the near-wall ASM model brings the predictions significantly closer to the measurements. Streamline curvature is here sufficiently great that even across the buffer region its effects are significant; the near-wall ASM model employed is able to reproduce this effect.

In the case of the S-bend introduction of the near-wall ASM model did not have any significant effect on the mean velocity predictions which, as shown in Figure 3, are already in good agreement with Taylor et al's [3] measurements. Use of the fully elliptic code did improve the static pressure predictions and, as shown in Figure 4, the use of the ASM model in the near-wall regions brings the predicted static pressure distribution in close agreement with the data.

4. CONCLUSIONS

The simplified low-Re ASM developed in this study results in more realistic and satisfactory predictions of curved channel flows and S-ducts, than the near-wall 1-equation model.

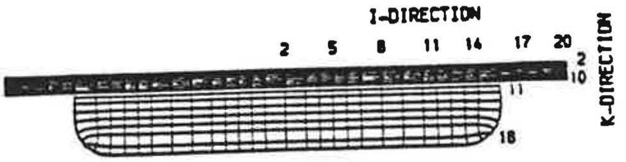


Fig. 1: Computation grid for the engine

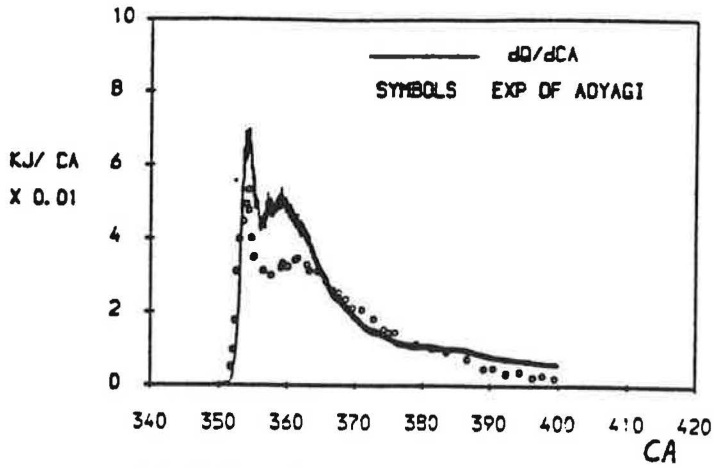


Fig. 2: Heat release rate ($A = 3 \times 10^{13}$)

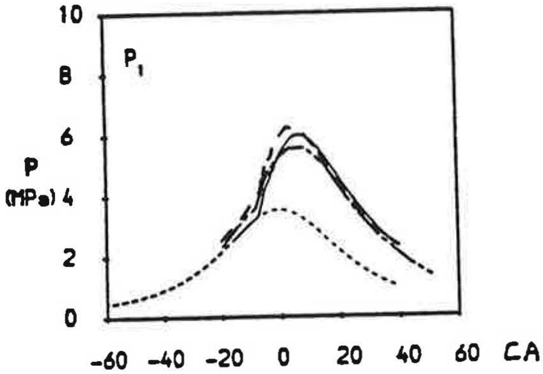


Fig. 3: Comparison of pressure diagrams

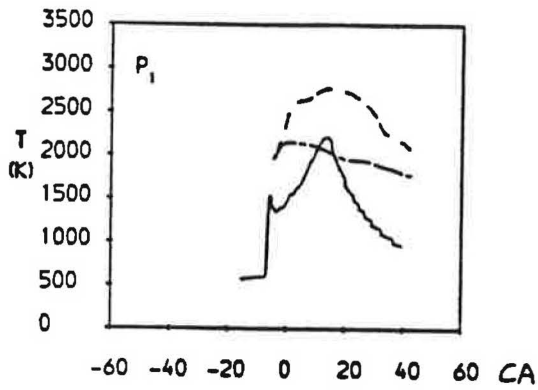
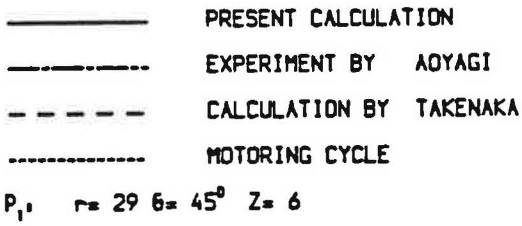


Fig. 4: Comparison of temperature diagrams



REFERENCES

[1] Ahmadi-Befrui, B., Gosman, A.D., Issa, R.I. and Watkins, A.P. "EPISO - an implicit non-iterative solution procedure for the calculation of flows in reciprocating engine chambers". *Comp. Meth. Appld. Mech. and Engineering*, 79:249–279, 1990.

[2] Wang, D.M. *Modelling spray wall impaction phenomena and combustion processes of diesel engines.* PhD thesis, University of Manchester, Faculty of Technology, 1992.

[3] Aoyagi, Y., Kamimoto, T., Matsui, Y. and Matsuoka, S. "A gas sampling study on the formation processes of soot and NO in a DI diesel engine". SAE Paper 800254, 1980.

[4] Takenaka, Y., Aoyagi, Y., Tsuji, Y. and Joko, I. "3D numerical simulation of fuel injection and combustion in DI diesel engines". SAE Paper 890668, 1989.